NASA-CR-204254

CALSPAN - UB RESEARCH CENTER
Buffalo, New York 14225

"Further Developments of BEM for Micro and Macromechanical Analyses of Composites"

BEST-CMS
(Boundary Element Software Technology - Composite Modeling System)
User's Manual

Final Technical Report
NASA Grant No. NAG3-1483

April 1993 - March 1997

Prepared for:
NASA Lewis Research Center
Cleveland, Ohio

Prepared by:
P. K. Banerjee, Principal Investigator*
D. P. Henry*
D. A. Hopkins**
R. K. Goldberg**

*Computational Engineering Mechanics Laboratory, Dept. of Civil Engineering, SUNY Buffalo

**NASA Lewis Research Center, Cleveland, OH

TABLE OF CONTENTS

1.0 | INTRODUCTION

BEST-CMS (Boundary Element Solution Technology - Composite Modeling System) is an advanced engineering system for the micro-analysis of fiber composite structures. **BEST-CMS** is based upon the boundary element program BEST3D which was developed for NASA by Pratt and Whitney Aircraft and the State University of New York at Buffalo under contract NAS3-23697. **BEST-CMS** presently has the capabilities for elastostatic analysis, steady-state and transient heat transfer analysis, steady-state and transient concurrent thermoelastic analysis and elastoplastic and creep analysis. The fibers are assumed to be perfectly bonded to the composite matrix, or in the case of static or steady-state analysis, the fibers may be assumed to have spring connections, thermal resistance, and/or frictional sliding between the fibers and the composite matrix. Additional features of the program are listed in Table 1.1, and new features are listed in Table 1.2.

The primary objective of this User's Manual is to provide an overview of all **BEST-CMS** capabilities, along with detailed descriptions of the input data requirements. In the next chapter, a brief review of the theoretical background is presented for each analysis category. Then, Chapter 3 discusses the key aspects of the numerical implementation, while Chapter 4 provides a tutorial for the beginning **BEST-CMS** user. The heart of the manual, however, is in Chapter 5, where a complete description of all data input items is provided. Within this chapter, the individual entries are grouped on a functional basis for a more coherent presentation. Chapter 6 includes sample problems and should be of considerable assistance to the novice. Chapter 7 includes capsules of a number of fiber-composite analysis problems that have been solved using **BEST-CMS**. This chapter is primarily descriptive in nature and is intended merely to illustrate the level of analysis that is possible within the present **BEST-CMS** system. Chapter 8 contains a detail description of the BEST-CMS Neutral File which is helpful in writing an interface between BEST-CMS and any graphic post-processor program. Finally, all pertinent references are listed in Chapter 9.

TABLE 1.1
GENERAL FEATURES OF BEST-CMS

- **For efficient use of disk space, all temporary files created by BEST-CMS are stored in single precision.**

- Conforming element approach to provide inter-element continuity of the field variables, along with efficient solutions

- Substructured Regions (Super-elements) to permit multiple materials and more efficient solutions

- Automatic adaptive numerical integration schemes

- Cyclic and planar symmetry

- Local or global boundary condition specification

- Sliding, frictional spring and resistance-type interfaces

- Block banded solver routines

- Restart capability for low cost re-analysis

- Free-format, keyword-driven input

- Automatic error checks of input data

- Automatic check of equilibrium and heat balance

Table 1.2

What is new for BEST-CMS Version 4.0

Micro-surface modeling of fibers - This feature allows greater detail in modeling of the fibers and the fiber/matrix interface without any additional modeling effort from the user. It is very effective in modeling curved fibers and fiber weaves, and more result information is output for post-processing so that the fiber can be analyzed in greater detail. (See **GMR card in Section 5.3.1.)

Frictional Contact on micro-surface interfaces - The nonlinear frictional contact was extended for use with micro-surfaces. (See INTErface card in Section 5.3.5.)

Sampling surface - The user can define elements for which result information will be output. The information is output to the neutral file for use in post-processing. (See SAMP SURF card in Section 5.3.7.)

Rigid Boundary Conditions - (See RIGID and FORCE cards in Section 5.5.5.)

Standard inserts with elliptical and variable cross sections - (See POINT card in Section 5.3.5.)

Larger Size Problems can be analyzed in BEST-CMS 4.0. A file called PARAM was introduced which contains all critical parameters used in BEST-CMS. This allows the user to customize the size of BEST-CMS to suit their specific requirements by simply changing the values in this file before compiling the computer code. Definitions of all parameters are included in this file and two sample parameter files have been provided (PARAM.3K, PARAM.6K).

Improvements in numerical integration have been made to allow problems with small dimensions (of the order of 10^{-3}).

Improvements have been made in the calculation of stresses.

A Makefile is provided for compiling BEST-CMS with the recommended compiler options for five different UNIX workstations (HP, SGI, IBM, SUN, DEC).

Performance has been enhanced on most workstations and some memory leaks and other impurities in earlier versions of BEST-CMS have been eliminated.

| 2.0 | THEORETICAL BACKGROUND |

| 2.1 | INTRODUCTION |

The mathematical background of the boundary element method has been known for nearly 100 years. Indeed some of the boundary integral formulations for elastic, elastodynamic, wave and potential flow equations have existed in the literature for, at least, 50 years. With the emergence of digital computers the method had begun to gain popularity as the 'boundary integral equation method', 'panel method', or 'integral equation method' during the sixties. The name was changed to the 'Boundary Element Method' by Banerjee and Butterfield [1981] in 1975, so as to make it more appealing to the engineering analysis community. Since then a number of textbooks and advanced level monographs have appeared which give a very comprehensive and thorough account of the existing literature on the method.

In this section, the different types of ceramic composites analyses incorporated in **BEST-CMS** are briefly outlined.

2.2	STEADY STATE HEAT CONDUCTION

AND ELASTOSTATIC BEM FORMULATION

The conventional boundary integral equations for elastostatic and steady-state heat conduction analyses are used in deriving a boundary element formulation for the analysis of ceramic composite structures. The boundary integral equation written for a point in the interior of the composite matrix is modified by adding to it the boundary integral equations of each insert written at the same point in the composite matrix. This eventually eliminates the displacement (or temperature) variables on the insert-matrix interface from the system, and therefore, reduces the total number of equations required for a solution of the system.

The BEM formulation for the steady-state heat conduction analysis of ceramic composites is identical to the elastostatic formulation, and therefore, the two formulations will be derived as one.

Boundary Integral Equation Formulations

The direct boundary integral equation for the displacement (or temperature) at a point ξ inside an elastic composite matrix is

$$C_{ij}(\xi)u_i(\xi) = \int_S \left[G_{ij}^O(x,\xi)t_i^O(x) - F_{ij}^O(x,\xi)u_i^O(x) \right] dS(x)$$
$$+ \sum_{n=1}^N \int_{S^n} \left[G_{ij}^H(x,\xi)t_i^H(x) - F_{ij}^H(x,\xi)u_i^H(x) \right] dS^n(x) \qquad (2.2.1)$$

$$i,j = 1,2,3 \text{ for elastostatics}$$
$$i,j = 1 \text{ for heat conduction}$$

where

G_{ij}, F_{ij} are the fundamental solutions of the governing differential equations of the ceramic matrix of infinite extent

C_{ij} are constants determined by the geometry at ξ

u_i, t_i are displacements and tractions (or temperature and flux)

S, S^n are the surfaces of the outer boundary of the matrix and the nth hole (left for fiber), respectively

N is the number of individual insert fibers

Superscripts O and H identify the quantities on the outer surface of the matrix and the quantities on the surface of the hole, respectively.

The conventional boundary integral equation for displacement (or temperature) can also be written for each of the N insert fibers. For the displacement (or temperature) at a point ξ inside the nth insert we can write

$$C_{ij}^{I}(\xi)u_i(\xi) = \int_{S^n} \left[G_{ij}^{I}(x,\xi)t_i^{I}(x) - F_{ij}^{I}(x,\xi)u_i^{I}(x) \right] dS^n(x) \qquad (2.2.2)$$

$$i,j = 1,2,3 \text{ for elastostatics}$$
$$i,j = 1 \text{ for heat conduction}$$

G_{ij}^{I}, F_{ij}^{I} are the fundamental solutions of the nth insert

C_{ij}^{I} are constants determined by the geometry at ξ in insert n

u_i^{I}, t_i^{I} are displacement and tractions (or temperature and flux) associated with the nth insert

S^n the surface of the nth insert

We next examine the interface conditions between the composite matrix and the insert. For a perfect bond the displacement (or temperature) of the matrix and the displacement (or temperature) of the inserts are equal and the tractions (or fluxes) along the interface are equal and opposite.

$$u_i^{H}(x) = u_i^{I}(x) \qquad (2.2.3a)$$

$$t_i^{H}(x) = -t_i^{I}(x) \qquad (2.2.3b)$$

In elastostatics, when the elastic modulus of the insert is much greater than the modulus of the composite matrix, the Poisson ratio of the insert can be assumed equal to that of the matrix with little error (no approximation is required for the heat conduction analysis). Therefore, upon consideration of the surface normals at the interface and examination of the F_{ij} kernels, we can write the following relation for the nth insert

$$F_{ij}^{I}(x,\xi) = -F_{ij}^{H}(x,\xi) \qquad (2.2.3c)$$

Substitution of equations (2.2.3) into equation (2.2.2) yields the following modified boundary integral equation for insert n.

$$C_{ij}^{I}(\xi)u_i(\xi) = \int_{S^n} \left[-G_{ij}^{I}(x,\xi)t_I^{H}(x) + F_{ij}^{H}(x,\xi)u_i^{H}(x) \right] dS^n(x) \qquad (2.2.4)$$

Finally adding the N insert equations (2.2.4) to equation (2.2.1) and cancelling terms, yields the modified boundary integral equation for the composite matrix

$$\bar{C}_{ij}(\xi)u_i(\xi) = \int_{S} \left[G_{ij}^{O}(x,\xi)t_i^{O}(x) - F_{ij}^{O}(x,\xi)u_i^{O}(x) \right] dS(x)$$
$$+ \sum_{n=1}^{N} \int_{S^n} \bar{G}_{ij}^{n}(x,\xi)t_i^{H}(x)dS^n(x) \qquad (2.2.5)$$

where

$$\bar{G}_{ij}^{n}(x,\xi) = G_{ij}^{H}(x,\xi) - (G_{ij}^{I}(x,\xi))^n$$

$\bar{C}_{ij}(\xi)$ are constants dependent on the geometry for a point ξ on the outer boundary and $\bar{C}_{ij}(\xi) = \delta_{ij}$ for a point ξ in the interior of the body.

Analytic Integration Around an Insert

The boundary element discretization of equations (2.2.4) and (2.2.5) in the conventional manner [Banerjee and Butterfield (1981)] requires a very fine discretization about the hole/insert. Alternatively, a new formulation is introduced in this report for the efficient modeling and analysis of holes/inserts using what the authors refer to as 'Insert Elements'. The inserts are defined with Insert Elements by describing the centerline of the (curvilinear, tubular) insert with nodal points; defining the connectivity of the nodal points; and specifying the radius of the insert at each of these nodal points. Internally the program generates the surface of the insert and the hole in which the resulting displacements (or temperatures) and tractions (or fluxes) are described using a trigonometric circular shape function in the circumferential direction and a curvilinear shape function of any order in the longitudinal direction (the present work employs both linear and quadratic shape functions for this purpose). A long hole (which is allowed to vary in diameter) can be described by a number of the insert elements connected end to end, and any insert element not connected to another is assumed, by the program, to be closed at the end by a circular disc.

Using the concept of the insert element, the essential part of the formulation is the conversion of the two-dimensional surface integration of the insert (and of the hole) to a one-dimensional integration. By performing a semi-analytical integration on the surface of the hole (or insert) the numerical integration is significantly reduced. In equation (2.2.5) the integral under the summation is the integral associated with the hole which is to be modified. To facilitate an analytic integration in the circumferential direction, the three-dimensional kernel functions are first expressed in local coordinates with the center of the coordinate system coinciding with the center of the insert/hole and the z axis aligned with the centerline of the insert. The relative translation ξ'_i is added to the field coordinate ξ_i and the rotation is applied using the appropriate vector transformation.

$$\xi_i = a_{ij}\bar{\xi}_j + \xi'_i$$

where a_{ij} are the direction cosines between the axis of the local and global coordinate systems and the bar indicates a local variable.

The integration point x_i for a ring can now be expressed in cylindrical coordinates relative to the center of the hole/insert as

$$x_1 = R\cos\theta$$

$$x_2 = R\sin\theta$$

$$x_3 = 0$$

where R represents the radius of the insert, i.e., $R = (x_1^2 + x_2^2)^{1/2}$.

The normal vectors are transformed by

$$n_1 = n_r cos\theta$$

$$n_2 = n_r sin\theta$$

$$n_3 = n_z$$

where n_r and n_z represents the normals of the side of the hole in local coordinates and are dependent on the change in the radius of the insert/hole. On the side of a straight hole $n_r = 1$ and $n_z = 0$, and on the flat surface closing the end of the hole/insert $n_r = 0$ and $n_z = 1$.

Next a circular shape function is employed to approximate the variation in the traction (or flux) about the circumference of the hole/insert. The circular shape function is multiplied and integrated with the three-dimensional G_{ij}^n kernel, allowing the nodal values of traction (or flux) to be brought outside the integral. The shape function is expressed as

$$t_i = M^\gamma t_i^\gamma \quad \text{(summation over } \gamma \text{ is implied, } \gamma = 1, 2, 3)$$

where

$$M^1(\theta) = \frac{1}{3} + \frac{2}{3}cos\theta$$

$$M^2(\theta) = \frac{1}{3} + \frac{\sqrt{3}}{3}sin\theta - \frac{1}{3}cos\theta$$

$$M^3(\theta) = \frac{1}{3} - \frac{\sqrt{3}}{3}sin\theta - \frac{1}{3}cos\theta$$

and t_i^γ is the nodal traction (or flux).

A modified circular shape function is used in the integration over the end of the hole to insure continuity of traction at the center of the end surface. The modified shape function is expressed as:

$$\bar{M}^\gamma = aM^\gamma + b/3 \qquad \gamma = 1, 2, 3$$

$$a = r/R \qquad\qquad b = (R - r)/R$$

where

R	is the radius of the hole at the end,
r	is the location of the integration (Gauss) point as it sweeps from $r = 0$ to $r = R$, and
M^γ	is the circular shape function defined above.

The traction must also be transformed between the local and the global systems by

$$t_i = a_{ik}\bar{t}_k$$

or

$$\bar{t}_j = a_{mj}t_m$$

Since flux is a scalar quantity, no transformation is necessary in the heat conduction analysis.

The last term in equation (2.2.5) can now be analytically integrated in the circumferential direction. For the mth hole the two integrals involved can be expressed as

$$\int_{S^m} \bar{G}_{ij}^n(x,\xi)t_i^H(x)ds^m(x) = \int_{C^m} a_{jk} \int_0^{2\pi} \bar{G}_{\ell k}^{local}(R,\theta,z,\bar{\xi})M^\gamma Rd\theta a_{i\ell}t_i^\gamma dC^m$$

$$= \int_{C^m} \bar{G}_{ij}^\gamma(R,z,\bar{\xi})t_i^\gamma dC^m(z)$$

where the indicated integration over C^m is now a one-dimensional curvilinear integration along the hole and $\bar{G}_{ij}^H$ represent the analytically integrated hole/insert kernels. Note, since the transformation vector a_{ik} is the independent of angle θ, it may be taken outside the $d\theta$ integration. Similar analytic integration is also performed in equation (2.2.4).

The final kernel functions of the hole/insert obtained from the analytical integration are very lengthy and therefore are not presented in this report. They contain functions of elliptical integrals which in general are expressed numerically by common series approximations. For a range of input values (coordinate locations), several higher order elliptical integral functions were found to produce incorrect numerical results. To overcome this problem, several new series were derived using a best fit polynomial approximation (as a function of the modulus of elliptic integrals) using values of the integrals calculated by a very accurate numerical integration in the circumferential direction.

The derivation of the insert/hole kernels corresponding to the strain equation are accomplished from the displacement equation (2.2.5) by differentiation and application of the strain-displacement equations. The stress equation is then found using Hooke's Law. Due to the complexity of the resulting insert/hole kernels, the authors perform the required differentiation before the analytic circumferential integration.

Finally we note, an insert which has curvature along its length will differ in surface area about the circumference on the curved portion of the insert. This is neglected in the formulation since the analytical integration is performed on an axisymmetric ring in which the surface area is constant about the circumference. This error, however, is small and disappears completely on a straight tubular insert which is most commonly encountered. We should also mention that the inserts should not intersect the outer surface of the body or intersect other inserts. This minor restriction can be ignored if results at these locations are not of interest. As long as the inserts do not coincide with nodal points of other elements the errors will be localized and will not affect the overall boundary element solution.

Numerical Implementation

The integral representations of the previous section are extremely accurate statements of the ceramic composite problem, however, approximations such as finite discretization

and numerical integration are necessary in order to obtain a solution to non-trivial problems. The goal of the numerical implementation of the present formulation is to obtain the most accurate and efficient implementation possible.

Discretization

After the analytical integration in the circumferential direction is complete, an insert in a three-dimensional solid can be modeled as a two-dimensional curvilinear line element with a prescribed radius at each longitudinal node. In the present work, linear and quadratic shape functions are utilized in modeling the geometry and field variables along the insert element as well as the boundary elements on the outer surface of the 3-D body. In the discretized form the displacement boundary integral equation for an elastic body containing inserts (equation (2.2.4)) can be expressed for a single insert as

$$C_{ij}^I(\xi)u_i(\xi) = -\sum_{p=1}^{P}\left[\int_{C^p} G_{ij}^I(x,\xi)N^\gamma(\eta)dC^p\right]t_i^\gamma$$
$$+\sum_{p=1}^{P}\left[\int_{C^p} F_{ij}^I(x,\xi)N^\gamma(\eta)dC^p\right]u_i^\gamma \qquad (2.2.6)$$

where

P is the number of line elements, and

$N^\gamma(\eta)$ represents a shape function over the curvilinear line element. Summation over γ is implied.

t_i^γ and u_i^γ are nodal values of traction (or flux) and displacement (or temperature) on the surface of the hole, respectively.

In a similar manner, equation (2.2.5) can be discretized using one- and two-dimensional shape functions in the following manner.

$$\bar{C}_{ij}(\xi)u_i(\xi) = \sum_{q=1}^{Q}\left[\int_{S^t} G_{ij}^O(x,\xi)L^\beta(\eta_1,\eta_2)dS^q\right]t_i^\beta$$
$$-\sum_{q=1}^{Q}\left[\int_{S^t} F_{ij}^O(x,\xi)L^\beta(\eta_1,\eta_2)dS^q\right]u_i^\beta \qquad (2.2.7)$$
$$+\sum_{p=1}^{P}\left[\int_{C^p} \bar{G}_{ij}(x,\xi)N^\gamma(\eta)dC^p\right]t_i^\gamma$$

where

Q is the number of surface elements on the outer surface of the composite matrix in the region, and

$L^\beta(\eta_1,\eta_2)$ represents a two-dimensional shape function. Summation over γ and β is implied.

It is important to note that the displacement and traction (or temperature and flux) on an insert/hole varies in the longitudinal as well as the circumferential direction, i.e., for displacement (or temperature).

$$u_j = M^\alpha N^\gamma u_j^{\alpha\gamma}$$

The circular shape function M^α has been analytically integrated into the kernel functions of equations (2.2.6) and (2.2.7). The ends of the inserts are assumed to be a flat surface and a one-dimensional numerical integration is carried out in the radial direction. The coefficients obtained from the integration over the end are lumped with their respective coefficients from the integration of the side of the insert.

Equations for stress, strain, or flux at points in the interior of a body can be discretized and integrated in a similar manner.

Numerical Integration

The complexity of the integral in the discretized equation necessitates the use of numerical integration for their evaluation. The steps in the integration process for a given element is outlined below:

1. Using appropriate Jacobian transformations, a curvilinear line element or surface element is mapped on a unit line or on a flat unit cell, respectively.

2. Depending on the proximity between the field point (ξ) and the element under consideration, there may be element subdivision and additional mapping for improved accuracy.

3. Gaussian quadrature formulas are employed for the evaluation of the discretized integral over each element (or sub-element). These formulas approximate the integral as a sum of weighted function values at designated points. The error in the approximation is dependent on the order of the (Gauss) points employed in the formula. To minimize error while at the same time maintaining computational efficiency, optimization schemes are used to choose the best number of points for a particular field point and element (Watson, 1979).

4. When the field point coincides with a node of the element being integrated, the integration becomes singular. In this case, the value of the coefficients of the F_{ij} kernel corresponding to the singular node cannot be calculated accurately by numerical integration. Instead, after the integration of all elements is complete, this value is determined so as to satisfy a rigid body displacement of the body (Banerjee and Butterfield, 1981).

Assembly of Equations

After the derivation of the modified boundary integral equations and the analytical circumferential integration of the kernel functions, the next critical step in the formulation is the assembly of the inserts in the system equations. Here, efficiency is of utmost importance. The approach to writing an efficient algorithm is to keep the number of system

equations to a minimum by eliminating all unnecessary unknowns from the system. The strategy is to retain in the system only traction variables on the matrix-insert interface. This is in contrast to a general multi-region problem where both displacement and tractions are retained on an interface. The elimination of the displacement on the interface is achieved through a backsubstitution of the insert equations in the system equations which are made up exclusively from equations written for the composite matrix (on the outer surface and on the surface of the holes). The procedure is described below.

Equation (2.2.7) is used to generate a system of equations for nodes on the outer surface of the composite matrix and for nodes on the surface of the holes containing the inserts. Written in matrix form we have

On the Matrix Outer Surface: $\quad \mathbf{G^O t^O} - \mathbf{F^O u^O} + \bar{\mathbf{G}} \mathbf{t^H} = 0$ (2.2.8a)

On the Matrix Hole Surface: $\quad \mathbf{G^O t^O} - \mathbf{F^O u^O} + \bar{\mathbf{G}} \mathbf{t^H} = \mathbf{I u^H}$ (2.2.8b)

where

$\mathbf{t^O}$ and $\mathbf{u^O}$ are traction and displacement (or flux and temperature) vectors on the outer surface of the composite matrix

$\mathbf{t^H}$ and $\mathbf{u^H}$ are traction and displacement (or flux and temperature) vectors on the hole

$\mathbf{I}$ is the identity matrix

$\mathbf{G^O}$ and $\mathbf{F^O}$ matrices contain coefficients from the integration over the outer boundary.

$\bar{\mathbf{G}}$ matrix contains coefficients integrated about the hole/insert

Our goal is to eliminate $\mathbf{u^H}$ from the system. To this end, equation (2.2.6) is written for every node on an insert, collocating slightly outside the boundary of the insert [at a distance of $(1.25)*($insert radius$)$] where $C^I_{ij}(\xi) = 0$.

$$\mathbf{F^{I2} u^I} = \mathbf{G^{I2} t^I}$$

Superscript **I2** identifies the equations written at points located slightly outside the boundary of the inserts.

Noting $\mathbf{u^H} = \mathbf{u^I}$ and $\mathbf{t^H} = \mathbf{t^I}$ we have

$$\mathbf{F^{I2} u^H} = -\mathbf{G^{I2} t^H}$$ (2.2.9)

Post multiplying equation (2.2.8b) by the $\mathbf{F^{I2}}$ matrix in equation (2.2.9) yields

$$\mathbf{F^{I2} G^O t^O} - \mathbf{F^{I2} F^O u^O} + \mathbf{F^{I2} \bar{G} t^H} = \mathbf{F^{I2} u^H}$$ (2.2.10)

Equation(2.2.9) can now be set equal to equation (2.2.10) and the final form of the system is derived.

On Outer Surface: $\quad \mathbf{G^O t^O} - \mathbf{F^O u^O} + \bar{\mathbf{G}} \mathbf{t^H} = \mathbf{O}$

On Hole: $\mathbf{F^{I2}G^Ot^O - F^{I2}F^Ou^O + (F^{I2}\bar{G} + G^{I2})t^H = O}$ (2.2.11)

At every point on the outer surface, either the traction or the displacement (or the temperature or the flux) is specified and on the surface of the hole only the tractions (or fluxes) are retained. Therefore, the number of equations in the system are equal to the final number of unknowns, and hence, the system can be solved. Thereafter, equation (2.8b) is used to determine the displacement on the matrix-insert interface.

It should be noted that since the displacement (or temperature) about a particular hole is present only in the insert equation corresponding to that hole, backsubstitution can be performed one insert at a time in a more efficient manner than backsubstitution of all inserts at once. Further, nowhere in the assembly process is a matrix inversion necessary. This efficient assembly process was made possible due to the unique formulation of the modified boundary integral equations developed earlier in this chapter.

When the composite matrix is divided into a multi-region model, the above insert assembly is performed for each region independently. Thereafter, equilibrium and compatibility conditions are invoked at common interfaces of the substructured matrix composite. After collecting together the known and unknown boundary quantities, the final system can be expressed as

$$\mathbf{A^b x = B^b y}$$ (2.2.12)

where

x is the vector of unknown variables at outer boundary unknown tractions (or fluxes) along the hole/insert interface

y is the vector of known variables on the outer boundary of the composite matrix, and

$\mathbf{A^b, B^b}$ are the coefficient matrices

Standard numerical procedures are used to solve the unknowns in equation (2.2.12). Details are described in the computer development section.

Once the unknowns are determined, the resulting displacements (or temperatures) on the matrix-insert interfaces can be found using equation(2.2.8b) rendering a complete boundary solution for both the inserts and composite matrix.

Interior Quantities

Once all of the displacements and tractions (or temperature and fluxes) are known on the matrix outer surface and on the matrix-insert interface, interior quantities of displacement, stress and strain (or temperature and flux) can be determined at any point in the composite matrix or insert. For displacement (or temperature), either the conventional boundary displacement (or temperature) integral equation (2.2.1) or (2.2.2) can be employed or alternatively the modified equations (2.2.3) or (2.2.5) can be used.

Equations for strains can be derived from the forementioned displacement equations and the strain-displacement relations. Thereafter, equations for stress are obtained by substituting the resulting strain equations into Hooke's law.

The resulting equations, however, are not only invalid on the surface, but also difficult to evaluate numerically at points close to it. For points on the surface, the stresses can be calculated by constructing a local Cartesian coordinate system with the axes 1 and 2 directed along the tangential directions and the axis 3 in the direction of the outward normal. The stresses $\bar{\sigma}_{ij}$ referred to these local axes (indicated by overbars) are then given by:

$$\bar{\sigma}_{11} = \frac{\nu}{1-\nu}\bar{t}_3 + \frac{E\nu}{1-\nu^2}\left(\bar{\varepsilon}_{11} + \bar{\varepsilon}_{22}\right) + \frac{E}{1+\nu}\bar{\varepsilon}_{11}$$

$$\bar{\sigma}_{12} = \bar{\sigma}_{21} = \frac{E}{2(1+\nu)}\bar{\varepsilon}_{12}$$

$$\bar{\sigma}_{22} = \frac{\nu}{1-\nu}\bar{t}_3 + \frac{E\nu}{1-\nu^2}\left(\bar{\varepsilon}_{11} + \bar{\varepsilon}_{22}\right) + \frac{E}{1+\nu}\bar{\varepsilon}_{22} \qquad (2.2.13)$$

$$\bar{\sigma}_{33} = \bar{t}_3$$

$$\bar{\sigma}_{32} = \bar{\sigma}_{23} = \bar{t}_2$$

$$\bar{\sigma}_{31} = \bar{\sigma}_{13} = \bar{t}_1$$

where E is the Young's modulus, $\bar{\varepsilon}_{ij}$ defines the components of the strains in the local axes system and $\bar{t}_i$ are the traction on the boundary. This method of evaluating the stresses on the surface was originally devised by (Rizzo and Shippy, 1968).
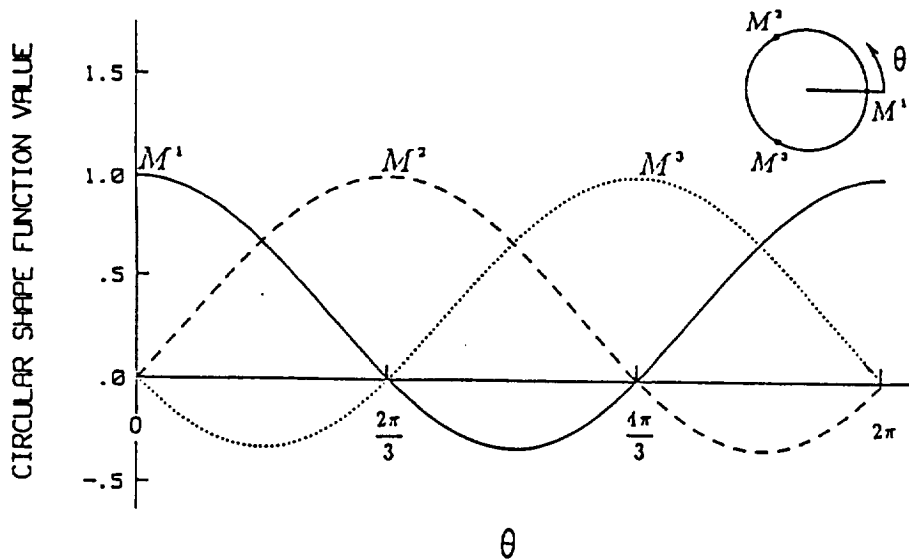
Fig. 2.1 Value of the 3-Nodal Circular Shape Function about the Fiber/Hole
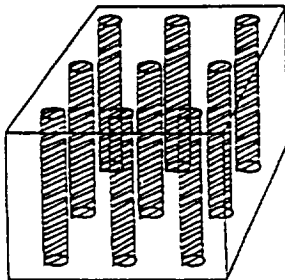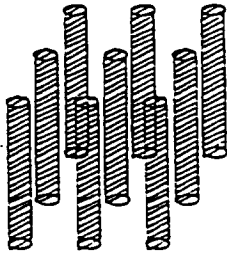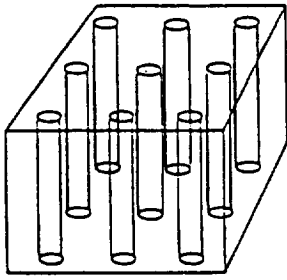
$$C_{ij}(\xi)u_i(\xi) = \int_S \left[G_{ij}^M(x,\xi)t_i^O(x) - F_{ij}^M(x,\xi)u_i^O(x) \right] dS(x)$$

$$+ \sum_{n=1}^N \int_{S^n} \left[G_{ij}^M(x,\xi)t_i^H(x) - F_{ij}^M(x,\xi)u_i^H(x) \right] dS^n(x)$$

$$C_{ij}^F(\xi)u_i(\xi) = \int_{S^n} \left[G_{ij}^F(x,\xi)t_i^F(x) - F_{ij}^F(x,\xi)u_i^F(x) \right] dS^n(x)$$

$$\bar{C}_{ij}(\xi)u_i(\xi) = \int_S \left[G_{ij}^M(x,\xi)t_i^O(x) - F_{ij}^M(x,\xi)u_i^O(x) \right] dS(x)$$

$$+ \sum_{n=1}^N \int_{S^n} \bar{G}_{ij}(x,\xi)t_i^H(x)dS^n(x)$$

where

$$\bar{G}_{ij}(x,\xi) = G_{ij}^H(x,\xi) - G_{ij}^F(x,\xi)$$

$i,j = 1,2,3$ for elastostatics
$i,j = 1$ for heat conduction

Fig. 2.2 Boundary Integral Equation Formulation for Fiber Composite Materials

The boundary element formulation for the steady-state uncoupled thermoelastic analysis of perfectly bonded ceramic composite structures is similar to the elastostatic/heat conduction BEM formulation of Section 2.2. The primary difference is the inclusion of one-way coupling terms which, for the present formulation, requires the solution of the heat conduction equation prior to solving the thermoelastic equation set. This, however, is favorable since solving two uncoupled subsystems independently is more efficient than solving the system as a whole.

Boundary Integral Equation Formulation

Once again, the conventional boundary integral equation is the starting point for the uncoupled thermoelastic ceramic formulation. The displacement/temperature for a point ξ inside the elastic composite matrix is

$$C_{ij}(\xi)u_i(\xi) = \int_S \left[G^O_{ij}(x,\xi)t^O_i(x) - F^O_{ij}(x,\xi)u^O_i(x) \right] dS(x)$$

$$+ \sum_{n=1}^N \int_{S^n} \left[G^H_{ij}(x,\xi)t^H_i(x) - F^H_{ij}(x,\xi)u^H_i(x) \right] dS^n(x) \qquad (2.3.1)$$

$$i,j = 1,2,3,4$$

where

G_{ij}, F_{ij} are the uncoupled thermoelastic fundamental solutions of the governing differential equations of the ceramic matrix of infinite extent [Dargush, 1987]

$\quad C_{ij}$ are constants determined by the geometry at ξ

$\quad u_i, t_i$ are displacements and tractions for $i = 1,2,3$, and temperatures and fluxes for $i = 4$

$\quad S, S^n$ are surfaces of the outer boundary of the matrix and the nth hold (left for fiber), respectively

$\quad N$ is the number of individual insert fibers

Superscripts O and H identify the quantities on the outer surface of the matrix and the quantities on the surface of the hole, respectively.

The conventional boundary integral equation for displacement/temperature can also be written for each of the N insert fibers. For the displacement/temperature at a point ξ inside the nth insert we can write

$$C^I_{ij}(\xi)u_i(\xi) = \int_{S^n} \left[G^I_{ij}(x,\xi)t^I_i(x) - F^I_{ij}(x,\xi)u^I_i(x) \right] dS^n(x) \qquad (2.3.2)$$

$$i,j = 1,2,3,4$$

G_{ij}^I, F_{ij}^I are the fundamental solutions of the nth insert

C_{ij}^I are constants determined by the geometry at ξ in insert n

u_i^I, t_i^I are displacement and tractions ($i = 1, 2, 3$) and temperature and fluxes ($i = 4$) associated with the nth insert

S^n the surface of the nth insert

We next examine the interface conditions between the composite matrix and the insert. For a perfect bond the displacement/temperature of the matrix and the displacement/temperature of the inserts along the interface are equal and the tractions/fluxes are equal and opposite.

$$u_i^H(x) = u_i^I(x) \tag{2.3.3a}$$

$$t_i^H(x) = -t_i^I(x) \tag{2.3.3b}$$

Substitution of equations (2.3.3) into equation (2.3.2) yields the following modified boundary integral equation for insert n.

$$C_{ij}^I(\xi)u_i(\xi) = \int_{S^n} \left[-G_{ij}^I(x,\xi)t_I^H(x) + F_{ij}^H(x,\xi)u_i^H(x) \right] dS^n(x) \tag{2.3.4}$$

Finally adding N insert equations (2.3.4) to equation (2.3.1) and canceling terms, yields the modified boundary integral equation for the composite matrix

$$\bar{C}_{ij}(\xi)u_i(\xi) = \int_S \left[G_{ij}^O(x,\xi)t_i^O(x) - F_{ij}^O(x,\xi)u_i^O(x) \right] dS(x)$$
$$+ \sum_{n=1}^N \int_{S^n} \left[\bar{G}_{ij}^n(x,\xi)t_i^H(x) - \bar{F}_{ij}^n(x,\xi)u_i^H(x) \right] dS^n(x) \tag{2.3.5}$$

where

$$\bar{G}_{ij}^n(x,\xi) = G_{ij}^H(x,\xi) - \left(G_{ij}^I(x,\xi) \right)^n \tag{2.3.6}$$

$$\bar{F}_{ij}^n(x,\xi) = F_{ij}^H(x,\xi) + \left(F_{ij}^I(x,\xi) \right)^n \tag{2.3.7}$$

$\bar{C}_{ij}(\xi)$ are constants dependent on the geometry for a point ξ on the outer boundary and $\bar{C}_{ij}(\xi) = \delta_{ij}$ for a point in the interior of the body.

Assembly of Equations

The approach to writing an efficient algorithm is to keep the number of system equations to a minimum by eliminating all unnecessary unknowns from the system. Once again the strategy used is to retain in the system only traction/flux variables on the matrix-insert interface. This is in contrast to a general multi-region problem where both displacement/temperatures, and tractions/fluxes are retained on an interface. The elimination of the displacement/temperatures on the interface is achieved through a backsubstitution of the insert equations into the system equations which are made up exclusively

from equations written for the composite matrix (on the outer surface and on the surface of the holes). The procedure is described below.

Equations (2.3.4) and (2.3.5) can be discretized and integrated as described in Section 2.2. The thermoelastic and heat conduction equations are integrated simultaneously. Therefore, using equation (2.3.4) a system of equations can be generated for the nodal points on the discretized composite matrix model. The equations for the nodes on the outer boundary of the matrix and for the nodes on the hole surface can be written separately in matrix notation as:

On the Matrix Outer Surface:
$$G^O t^O - F^O u^O + \bar{G} t^H - \bar{F} u^H = 0 \qquad (2.3.8a)$$

On the Matrix Hole Surface:
$$G^O t^O - F^O u^O + \bar{G} t^H - \bar{F} u^H = I u^H \qquad (2.3.8b)$$

where

t^O and u^O are traction/flux and displacement/temperature vectors on the outer surface of the composite matrix

t^H and u^H are traction/flux and displacement/temperature vectors on the hole

I is the identity matrix

G^O and F^O matrices contain coefficients from the integration over the outer boundary. $\bar{G}$ and $\bar{F}$ matrices contain coefficients integrated about the hole/insert

Our goal is to eliminate u^H from the system. To this end, equation (2.3.4) is written for every node on an insert, collocating slightly outside the boundary of the insert [at a distance of (1.25)∗ (radius of insert)] where $C^I_{ij}(\xi) = 0$.

$$G^{I2} t^I - F^{I2} u^I = 0$$

Superscript I2 identifies the equations written at points located slightly outside the boundary of the inserts.

Noting $u^H = u^I$, and $t^H = -t^I$ we have

$$F^{I2} u^H = -G^{I2} t^H \qquad (2.3.9)$$

Post multiplying equation (2.3.8b) by the F^{I2} matrix in equation (2.3.9) yields

$$F^{I2} G^O t^O - F^{I2} F^O u^O + F^{I2} \bar{G} t^H - F^{I2} \bar{F} u^H = F^{I2} u^H \qquad (2.3.10)$$

Equation (2.3.9) can now be set equal to equation (3.10) and the final form of the system is derived.

On Outer Surface:
$$G^O t^O - F^O u^O + \bar{G} t^H - \bar{F} u^H = 0 \qquad (2.3.11a)$$

On Hole:
$$F^{I2} G^O t^O - F^{I2} F^O u^O + (F^{I2} \bar{G} + G^{I2}) t^H - F^{I2} \bar{F} u^H = 0 \qquad (2.3.11b)$$

Matrix $\bar{\mathbf{F}}$ of equation (2.3.11) are coefficients derived from the integration of the $\bar{F}_{ij}$ kernel defined in equation (2.3.7) as

$$\bar{F}_{ij} = F_{ij}^H + F_{ij}^I$$

In the elastostatic/heat conduction formulation of Section 2.2 this kernel is zero according to equation (2.2.3c). This allowed the displacement/temperature vector $\mathbf{u}^H$ to be removed from the formulation leaving an equal number of unknowns and equations for which a solution can be obtained. In the present formulation, if we once again assume the Poisson ratio of the insert to be the same as the composite matrix, all but three of the sixteen components in the $\bar{F}_{ij}$ kernel vanish. F_{41}, F_{42}, and F_{43} are non-zero when the coefficient of thermal expansion of the insert is not the same as the matrix. Examining the $\bar{F}_{ij}$ kernel more closely the following simplification is possible.

$$\bar{F}_{ij}(u,\xi)u_i^H(x) = \frac{\alpha^M - \alpha^I}{\alpha^M} \begin{bmatrix} 0 & 0 & 0 & F_{41}^H \\ 0 & 0 & 0 & F_{42}^H \\ 0 & 0 & 0 & F_{43}^H \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1^H \\ u_2^H \\ u_3^H \\ u_4^H \end{Bmatrix} \qquad (2.3.12)$$

where

u_1^H, u_2^H, u_3^H are the displacement components of the hole

u_4^H are the temperature component of the hole

$F_{41}^H, F_{42}^H, F_{43}^H$ are components of the F_{ij}^H kernel on the hole surface of the matrix, and

α^I, α^μ are the coefficients of thermal expansion of the insert and the matrix, respectively.

From the above equation it is clear the displacement about the hole/insert once again vanishes from the formulation, however, the temperature θ^H ($\theta^H = u_4^H$) is present as a coupling term, in the displacement equations (components 1,2 and 3 of the kernel) and vanishes in the temperature equation (component 4). Equation (2.3.11) may therefore be uncoupled and written as follows.

Heat conduction system

$$\text{Outer Surface:}\quad \mathbf{G}^\circ\mathbf{t}^\circ - \mathbf{F}^\circ\mathbf{u}^\circ + \bar{\mathbf{G}}\mathbf{t}^H = 0 \qquad (2.3.13a)$$

$$\text{Hole Surface:}\quad \mathbf{F}^{I2}\mathbf{G}^\circ\mathbf{t}^\circ - \mathbf{F}^{I2}\mathbf{F}^\circ\mathbf{u}^\circ + (\mathbf{F}^{I2}\bar{\mathbf{G}} + \mathbf{G}^{I2})\mathbf{t}^H = 0 \qquad (2.3.13b)$$

Thermal elastic system

$$\text{Outer Surface:}\quad \mathbf{G}^\circ\mathbf{t}^\circ - \mathbf{F}^\circ\mathbf{u}^\circ + \bar{\mathbf{G}}\mathbf{t}^H = \bar{\mathbf{B}}\theta^H \qquad (2.3.13c)$$

$$\text{Hole Surface:}\quad \mathbf{F}^{I2}\mathbf{G}^\circ\mathbf{t}^\circ - \mathbf{F}^{I2}\mathbf{F}^\circ\mathbf{u}^\circ + (\mathbf{F}^{I2}\bar{\mathbf{G}} + \mathbf{G}^{I2})\mathbf{t}^H = \bar{\mathbf{B}}^M\theta^H \qquad (2.3.13d)$$

where

$$\bar{\mathbf{B}}\theta^H = \bar{\mathbf{F}}\mathbf{u}^H \qquad (2.3.14a)$$

and

$$\bar{\mathbf{B}}^{\mathbf{M}}\theta^{\mathbf{H}} = \mathbf{F}^{\mathbf{I2}}\bar{\mathbf{F}}\mathbf{u}^{\mathbf{H}} \tag{2.3.14b}$$

In the heat conduction system the number of unknowns is equal to the number of equations. The heat conduction system can be solved and the nodal temperatures on the hole determined using an uncoupled form of equation (2.3.8b). With the nodal temperatures on the hole known, the number of unknowns in the thermoelastic equation system is reduced to exactly the number of equations, and therefore, this system can be solved. With the use of equation (2.3.8b) all remaining nodal boundary variables can be determined on the matrix-insert interface rendering a complete solution to the boundary value problem. Displacement, temperature, traction, flux, stress, or strain measures can now be determined at any point on the boundary or in the interior of the body as described in Section 2.2.

2.4	INTERFACE CONNECTIONS BETWEEN
	THE FIBER AND THE MATRIX

In order to accurately analyze a ceramic composite structure the interaction between the fiber and the composite matrix must be properly modeled. Interface phenomena such as perfect bonding, progressive debonding (with gap openings), frictional slipping, and plastic behavior along the matrix-insert interface must be included. The failure mode of an interface is dependent on the state of stress at the interface. Therefore, the general mode of failure will be nonlinear and irreversible rendering a path dependent, quasistatic analysis.

In this section the steady-state uncoupled thermoelastic boundary element formulations of Section 2.3 is rederived in a form suitable for the inclusion of nonlinear interface connections. In addition to the perfectly bonded interface, two new types of interface connections are presented in a general form so as to be interchangeable upon input by the user. Finally the assembly of the numerically integrated BEM equations is presented and an incremental algorithm for their solution is described. The elastostatic and steady-state heat conduction formulations are obtained from the uncoupled thermoelastic formulation.

Boundary Integral Equation Formulation

The conventional boundary integral equation for displacement/temperature is the starting point for the steady-state uncoupled thermoelastic ceramic formulation with non-linear matrix-insert interface connections. The displacement/temperature for a point ξ inside the elastic composite matrix is

$$C_{ij}(\xi)u_i(\xi) = \int_S \left[G_{ij}^O(x,\xi)t_i^O(x) - F_{ij}^O(x,\xi)u_i^O(x) \right] dS(x)$$

$$+ \sum_{n=1}^N \int_{S^n} \left[G_{ij}^H(x,\xi)t_i^H(x) - F_{ij}^H(x,\xi)u_i^H(x) \right] dS^n(x) \qquad (2.4.1)$$

$$i,j = 1,2,3,4$$

where

G_{ij}, F_{ij} are the fundamental solutions of the governing differential equations of the ceramic matrix of infinite extent [Dargush, 1987]

$\quad C_{ij}$ are constants determined by the geometry at ξ

$\quad u_i, t_i$ are displacements and tractions

$\quad S, S^n$ are the surfaces of the outer boundary of the matrix and the nth hole (left for fiber), respectively

$\quad N$ is the number of individual insert fibers

Superscripts O and H identify the quantities on the outer surface of the matrix and the quantities on the surface of the hole, respectively.

The conventional boundary integral equation for displacement can also be written for each of the N insert fibers. For the displacement/temperature at a point ξ inside the nth insert we can write

$$C_{ij}^I(\xi)u_i(\xi) = \int_{S^n} \left[G_{ij}^I(x,\xi)t_i^I(x) - F_{ij}^I(x,\xi)u_i^I(x) \right] dS^n(x) \qquad (2.4.2)$$

$$i,j = 1,2,3,4$$

G_{ij}^I, F_{ij}^I are the fundamental solutions based on material properties of the nth insert

$\quad C_{ij}^I$ are constants determined by the geometry at ξ in insert n

u_i^I, t_i^I are displacement and tractions associated with the nth insert

$\quad S^n$ the surface of the nth insert

We next examine the interface conditions between the composite matrix and the insert. The difference between the displacement/temperature of the insert and the displacement/temperature of the insert is defined by a variable d_i. The traction/flux on the hole is equal and opposite to the traction/flux on the insert. Therefore

$$u_i^H(x) = u_i^I(x) + d_i(x) \qquad (2.4.3a)$$

$$t_i^H(x) = -t_i^I(x) \qquad (2.4.3b)$$

Substitution of equations (2.4.3b) into equation (2.4.2) yields the following modified boundary integral equation for insert n.

$$C_{ij}^I(\xi)u_i(\xi) = \int_{S^n} \left[-G_{ij}^I(x,\xi)t_I^H(\xi) - F_{ij}^I(x,\xi)u_i^I(x) \right] dS^n(x) \qquad (2.4.4)$$

Finally adding N insert equations (2.4.4) to equation (2.4.1) and invoking equation (4.3a), yields the modified boundary integral equation for the composite matrix

$$C_{ij}(\xi)u_i(\xi) + \sum_{n=1}^{N} \left(C_{ij}^I(\xi) \right)^n u_i(\xi) = \int_{S} \left[G_{ij}^O(x,\xi)t_i^O(x) - F_{ij}^O(x,\xi)u_i^O(x) \right] dS(x)$$

$$+ \sum_{n=1}^{N} \int_{S^n} \left[\bar{G}_{ij}^n(x,\xi)t_i^H(x) - F_{ij}^H(x,\xi)d_i(x) - \bar{F}_{ij}^n(x,\xi)u_i^I(x) \right] dS^n(x) \qquad (2.4.5)$$

where

$$\bar{G}_{ij}^n(x,\xi) = G_{ij}^H(x,\xi) - \left(G_{ij}^I(x,\xi) \right)^n$$

$$\bar{F}_{ij}^n(x,\xi) = F_{ij}^H(x,\xi) + \left(F_{ij}^I(x,\xi) \right)^n$$

For a point in the interior of the matrix

$$C_{ij}(\xi) = \delta_{ij} \quad \text{and} \quad \left(C_{ij}^I(\xi) \right)^n = 0 \quad \text{for all } n \qquad (2.4.6a)$$

For a point in the interior of the kth insert

$$C_{ij}(\xi) = 0 \quad \text{and} \quad \left(C_{ij}^I(\xi)\right)^n = \begin{cases} 0 & \text{for} \quad n \neq k \\ \delta_{ij} & \text{for} \quad n = k \end{cases} \tag{2.4.6b}$$

For a point on the outer boundary of the matrix $(C_{ij}^O(\xi) = C_{ij}(\xi))$

$C_{ij}^O(\xi)$ is dependent on the geometry at ξ, and

$$\left((C_{ij}^I(\xi)\right)^n = 0 \qquad \text{for all } n \tag{2.4.6c}$$

For a point on the kth matrix-insert interface $(C_{ij}^H(\xi) = C_{ij}(\xi))$

$$C_{ij}^H(\xi) + \left((C_{ij}^I(\xi)\right)^k = \delta_{ij}$$
$$\left((C_{ij}^I(\xi)\right)^n = 0 \quad \text{for} \quad n \neq k \tag{2.4.6d}$$

For a point ξ on the kth matrix-insert interface the left-hand side of equation (2.4.5) may be rewritten using equation (2.4.3a) and (2.4.6d) as

$$C_{ij}^H(\xi)u_i^H(\xi) + \left(C_{ij}^I(\xi)\right)^k u_i^I(\xi) + \sum_{\substack{n=1 \\ n \neq k}}^N \left(C_{ij}^I(\xi)\right)^n u_i(\xi)$$

$$= C_{ij}^H(\xi)u_i^I(\xi) + C_{ij}^H(\xi)d_i(\xi) + \left(C_{ij}^I(\xi)\right)^k u_i^I(\xi) = \delta_{ij}u_i^I(\xi) + C_{ij}^H(\xi)d_i(\xi) \tag{2.4.7}$$

Interface Constitutive Relationships

The interface constitutive relationships of ceramic composite materials are, in general, nonlinear. This requires that their BEM solution be found using an incremental (quasistatic) algorithm. Therefore, the linear BEM equations of the previous section are interpreted as incremental relations and the interface constitutive relationships are defined in terms of the incremental displacements/temperatures $\dot{u}_j$ and the incremental tractions/fluxes $\dot{t}_i$ as

$$\dot{t}_i = k_{ij}^{ep}\dot{d}_j \tag{2.4.8a}$$

$$i,j = 1 \text{ to } 3 \text{ for elastostatics}$$
$$i,j = 1 \text{ for heat conduction}$$
$$i,j = 1 \text{ to } 4 \text{ for uncoupled thermoelasticity}$$

where k_{ij}^{ep} is the nonlinear constitutive matrix dependent on the current state of stress,

$$\dot{d}_j = \dot{u}_j^H - \dot{u}_j^I, \tag{2.4.8b}$$

$$\dot{t}_i = \dot{t}_i^I = -\dot{t}_i^H \tag{2.4.8c}$$

The elastostatic constitutive relationships that are derived in this section are expressed using a local coordinate system where the first component corresponds to the direction normal to the interface and the second and third components correspond to arbitrary tangential directions on the matrix-insert interface. The local constitutive matrix is transformed to the global coordinate system when incorporated in the BEM equations as follows

$$\left(k_{ij}^{ep}\right)^{\text{global}} = a_{\ell i}\left(k_{\ell m}^{ep}\right)^{\text{local}} a_{mj}$$

where a_{mj} is the directional cosine transformation tensor.

Linear Spring Interface Connection

The general form of the linear spring interface connection may be expressed as (k_{ij}^e replaces k_{ij}^{ep})

$$t_i = k_{ij}^e d_j \tag{2.4.9}$$

In the current elastostatic implementation the special case $k_{11}^e = k_n$, $k_{22}^e = k_{33}^e = k_t$, and $k_{ij}^e = 0$ for $i \neq j$, is assumed (see Figure 2.4.1). k_n is the spring constant in the normal direction and k_t is the spring constant in the tangential direction. The general form does not pose any more difficulty in implementation then the special case, but the additional generality is usually not required in practice. The limiting form as $k_n \to \infty$ and $k_t \to \infty$ approaches the case of a perfect bond, and $k_n = k_t = 0$ corresponds to a completely debonded interface. $k_n \to \infty$ and $k_t = 0$ corresponds to a sliding interface with a perfect connection in the normal direction. Results using the present formulation with values $k_n = k_t = (E^{\text{matrix}} + E^{\text{insert}}) * 10^6$ compared very closely to results obtained for the same problem assuming a perfect bond. Similar comparisons were found to hold true for problems with $k_n = k_t = 0$ verses $E^{\text{insert}} = 0$ (hole solution).

In the heat conduction implementation $k_{11} = -k$ where k is the thermal conductivity, $k = 0$ does not permit heat flow across the interface which is analogous to an insulated hole and $k \to \infty$ is analogous to a perfect bond between the insert and the matrix.

The k_{ij} for uncoupled thermoelastic analysis is derived by combining the elastostatic and heat conduction k_{ij} to form a completely uncoupled k_{ij} matrix $k_{11} = k_n, k_{22} = k_{33} = k_t, k_{44} = -k$, and $k_{ij} = 0$ for $i \neq j$.

Spring-Friction Nonlinear Interface (Coulomb Friction)

An interface model shown in Figure (2.4.2) exhibits a linear spring behavior normal to the interface when the normal tractions are in compression. Linear spring resistance is also observed in the tangential direction when the principal tangential traction is below the slip limit defined by the coulomb friction criteria. When the slip limit is reached the traction that the interface can sustain in the tangential direction has reached a maximum and any additional tangential traction will result in an irrecoverable shift of the relative displacement between the insert and the matrix, and a redistribution of stress (and therefore interface tractions) is required to bring the structure back in equilibrium. Furthermore, the model does not support tensile tractions normal to the interface. Instead the tractions at this point are set to zero through the constitutive relation $k_{ij} = 0$ ($i, j = 1$ to 3) and a gap between the insert and the matrix will form. Once again a redistribution in stress is required to bring the structure back in equilibrium.

The nonlinear interface constitutive relationship for a point on the interface exhibiting the spring-coulomb friction phenomena can be derived in a manner analogous to the incremental theory of plasticity [Selvadurai, 1988]. This requires a description and use of

(1) A flow rule relating the irrecoverable nonlinear part of the displacement difference rate to the traction state at the interface.

(2) A consistency relation requiring the new traction state to lie on the newly formed yield surface (defined by the hardening rule and yield function).

(3) A yield function defining the limits of elastic behavior, and

(4) A hardening rule defining the subsequent yield surfaces.

The variable for the incremental displacement difference across the interface is assumed to be composed of an elastic part $\dot{d}_i^e$ and a plastic part $\dot{d}_i^p$

$$\dot{d}_i = \dot{d}_i^e + \dot{d}_i^p \qquad (2.4.10)$$

where the elastic component is related to the interface traction by the linear constitutive relation.

$$\dot{t}_i = k_{ij}^e \dot{d}_j^e$$

or

$$\dot{t}_i = k_{ij}^e (\dot{d}_j - \dot{d}_j^p) \qquad (2.4.11)$$

Next a flow rule is defined as

$$\dot{d}_j^p = \lambda \frac{\partial Q}{\partial t_j}$$

where λ is an unknown flow factor dependent on the current state of traction and Q is a nonlinear potential. Substituting the flow rule into equation (2.4.11) yields

$$\dot{t}_i = k_{ij}^e \dot{d}_j - k_{ij}^e \lambda \frac{\partial Q}{\partial t_j} \qquad (2.4.12)$$

The consistency relation is defined as

$$\frac{\partial F}{\partial t_i} \dot{t}_i = 0 \qquad (2.4.13)$$

Substitution of equation (2.4.12) into the consistency relationship and rearranging yields a relationship for λ.

$$\lambda = \frac{1}{G} \frac{\partial F}{\partial t_i} k_{ij}^e \dot{d}_j \qquad (2.4.14)$$

where $G = \frac{\partial F}{\partial t_m} k_{mn}^e \frac{\partial Q}{\partial t_n}$

Finally substitution of equation (2.4.14) into equation (2.4.12) yields

$$\dot{t}_i = k_{ij}^{ep} \dot{d}_j \qquad (2.4.15)$$

where

$$k_{ij}^{ep} = k_{ij}^e - k_{ij}^p \qquad (2.4.16)$$

and

$$k_{ij}^p = \frac{1}{G} k_{iq}^e k_{pj}^e \frac{\partial F}{\partial t_p} \frac{\partial Q}{\partial t_q} \qquad (2.4.17)$$

The nonlinear interface constitutive relationship is complete once F and Q are defined.

Assuming the coulomb friction criteria, the yield function F is given by

$$F = (t_2^2 + t_3^2)^{1/2} + \mu t_n = 0 \qquad (2.4.18)$$

where t_n is the total traction normal to the interface, t_2 and t_3 are the total tangential tractions and μ is the coefficient of friction between the insert and the matrix.

This relation holds only for $t_n < 0$. If $F < 0$, the interface is assumed elastic and a spring connection is used. If $F = 0$ the nonlinear interface relation (2.4.15) is used. $F > 0$ is meaningless in theory, however, since in practice we assume a finite size load step the function may become slightly positive in which case it is treated as $F = 0$. When $t_n \geq 0$, $k_{ij} = 0$ ($i, j = 1$ to 3) is assumed.

This relationship is assumed constant for the entire analysis and therefore to complete the analogy with classical plasticity, we can say we have zero hardening. The nonlinear potential Q is assumed to be the magnitude of the principal tangential traction.

$$Q = (t_2^2 + t_3^2)^{1/2} \qquad (2.4.19)$$

If we assume the special elastostatic case described in the previous sub-section for k_{ij}^e where $k_{11} = k_n, k_{22} = k_{33} = k_t$, and $k_{ij}^e = 0$ for $i \neq j$ we can write for k_{ij}^p

$$k_{ij}^p = \begin{bmatrix} 0 & 0 & 0 \\ \mu k_n \hat{t}_2 & k_t \hat{t}_2 \hat{t}_2 & k_t \hat{t}_2 \hat{t}_3 \\ \mu k_n \hat{t}_3 & k_t \hat{t}_2 \hat{t}_3 & k_t \hat{t}_3 \hat{t}_3 \end{bmatrix} \qquad (2.4.20)$$

where $\hat{t}_2$ and $\hat{t}_3$ are the normalized components of total tangential tractions, i.e.

$$\hat{t}_2 = \frac{t_2}{(t_2^2 + t_3^2)^{1/2}} \qquad \text{and} \qquad \hat{t}_3 = \frac{t_3}{(t_2^2 + t_3^2)^{1/2}}$$

In the present heat conduction and uncoupled thermoelastic implementation, only a linear (uncoupled) relation is allowed between the increment of flux and the temperature difference rate variable across the interface. This of course will be updated in the newly proposed work.

Assembly of Equations for General Matrix-Insert Interface Connections

In the assembly of the perfectly bonded inserts the number of equations in the final system was reduced by eliminating all unnecessary unknowns from the problem. In the previous case all unknown displacement/temperature nodal variables on the matrix-insert interface were eliminated leaving only the unknown tractions/fluxes. In the present case the tractions/fluxes will be eliminated retaining only the matrix-insert displacement difference variables along the interface.

The steady-state uncoupled thermoelastic equations (2.4.4) and (2.4.5) can be discretized and integrated as described in Section 2.2. Therefore, using equations (2.4.4),

(2.4.6c), and (2.4.7) a system of equations can be generated for the nodes on the discretized model of the composite matrix. The equations for the nodes on the outer boundary of the matrix and for the nodes on the hole surface can be written separately in matrix notation as:

On the Matrix Outer Surface:
$$\mathbf{G^O t^O - F^O u^O + \bar{G} t^H - F^H d - \bar{F} u^I = 0} \qquad (2.4.21a)$$

On the Matrix Hole Surface:
$$\mathbf{G^O t^O - F^O u^O + \bar{G} t^H - F^H d - \bar{F} u^I = I u^I} \qquad (2.4.21b)$$

where

t^O and u^O are traction/flux and displacement/temperature vectors on the outer surface of the composite matrix

t^H is the traction/flux vector on the hole

u^I is the displacement/temperature vector on the insert

d vector of displacement/temperature difference between the hole and insert

I is the identity matrix

G^O and F^O matrices contain coefficients from the integration over the outer boundary

$\bar{G}, \bar{F}$ and F^H matrices contain coefficients from the integration over the hole/insert

Vectors $\mathbf{u^O}$ and $\mathbf{d}$ are arranged in the same order as the equations are written so as to produce a strong main diagonal in matrices $\mathbf{F^O}$ and $\mathbf{F^H}$.

Equation (2.4.6c) is used in writing equation (2.4.21a). The $C_{ij}^O(\xi) u_j^O(\xi)$ term is lumped with its respective (diagonal block) coefficient in matrix $\mathbf{F^O}$. The sum of these two terms are accurately calculated indirectly using the rigid body technique [Banerjee and Butterfield, 1981]. Equation (2.4.7) is used in writing equation (2.4.21b). The $C_{ij}^H(\xi) d_i(\xi)$ term is lumped with its respective (diagonal block) coefficient in matrix $\mathbf{F^H}$ (also calculated by the rigid body technique). Similarly, the term $\delta_{ij} u_i^I(\xi)$ could be lumped with matrix $\bar{F}$, but instead it is put on the right-hand side (forming an identity matrix $\mathbf{I}$ for use in subsequent matrix algebraic operations. Next, equation (2.4.5) is written for every shape function node on an insert, collocating slightly outside the boundary of the insert [at a distance of (1.25)$*$(radius of insert)] where $C_{ij}^I(\xi) = 0$.

$$\mathbf{G^{I2} t^I - F^{I2} u^I = 0} \qquad (2.4.22)$$

Superscript I2 identifies the equations written at points located slightly outside the boundary of the inserts. Equation (2.4.22) can be rewritten as

$$\mathbf{F^{I2} u^I = -G^{I2} t^H} \qquad (2.4.23)$$

Post multiplying equation (2.4.21b) by matrix $\mathbf{F^{I2}}$ of equation (2.4.23) yields

$$\mathbf{F^{I2} G^O t^O - F^{I2} F^O u^O + F^{I2} \bar{G} t^H - F^{I2} F^H d - F^{I2} \bar{F} u^I = F^{I2} u^I} \qquad (2.4.24)$$

For efficiency, this operation can be carried out one insert at a time. Equation (2.4.23) can now be set equal to equation (2.4.10). This yields,

$$\mathbf{F}^{I2}\mathbf{G}^O\mathbf{t}^O - \mathbf{F}^{I2}\mathbf{F}^O\mathbf{u}^O + \mathbf{G}^M\mathbf{t}^H - \mathbf{F}^M\mathbf{d} - \mathbf{F}^{I2}\bar{\mathbf{F}}\mathbf{u}^I = 0 \tag{2.4.25}$$

where

$$\mathbf{G}^M = \mathbf{F}^{I2}\bar{\mathbf{G}} + \mathbf{G}^{I2} \quad \text{, and}$$
$$\mathbf{F}^M = \mathbf{F}^{I2}\mathbf{F}^H$$

The nonlinear interface relation $\mathbf{t}^H = -\mathbf{K}^{ep}\mathbf{d}$ is substituted in equation (2.4.21a) and (2.4.25) and noting equations (2.3.12) and (2.3.14) the final form of the system can be written as

On Outer Surface:
$$\mathbf{G}^O\mathbf{t}^O - \mathbf{F}^O\mathbf{u}^O - (\bar{\mathbf{G}}\mathbf{K}^{ep} + \mathbf{F}^H)\mathbf{d} = \bar{\mathbf{B}}\theta^I \tag{2.4.26a}$$

On Hole Surface:
$$\mathbf{F}^{I2}\mathbf{G}^O\mathbf{t}^O - \mathbf{F}^{I2}\mathbf{F}^O\mathbf{u}^O - (\mathbf{G}^M\mathbf{K}^{ep} + \mathbf{F}^M)\mathbf{d} = \bar{\mathbf{B}}^M\theta^I \tag{2.4.26b}$$

For each degree of freedom on every node on the outer surface, either the displacement/temperature or traction/flux is specified and on the matrix-insert interface the displacement/temperature difference variable and the insert temperature variable are present. Therefore the number of unknowns in the system is greater than the number of equations by exactly the number of nodes on the interface (exceeded due to the insert temperature variables). However, as was the case in uncoupled thermoelasticity with a perfectly bonded matrix-insert interface, the $\bar{\mathbf{B}}$ and $\bar{\mathbf{B}}^M$ matrices influence only the thermoelastic solution, i.e., the coefficients corresponding to the heat conduction equations are zero. Therefore, in the heat conduction equations these terms do not appear. The heat conduction equation can be uncoupled from the system in a manner similar to equation (2.3.13) rendering an equal number of heat conduction equations and unknowns. The heat conduction system is solved and the insert nodal temperatures are determined using equation (2.4.21b). With the insert nodal temperatures known, the number of unknown in the thermoelastic equation system is reduced to exactly the number of equations, and therefore the system can be solved. With the use of equations (2.4.8) and (2.4.21b) all remaining nodal boundary variables can be determined on the matrix-insert interface rendering a complete solution to the boundary value problem. Displacement, temperature, traction, flux, stress, or strain measures can now be determined at any point on the boundary or in the interior of the body as described in Section 2.2.

The elastostatic and steady-state heat conduction formulations are derived from equation (2.4.26). Noting the $\bar{F}_{ij}$ kernel vanishes in elastostatic and heat conduction analysis, equation (2.4.21) is used setting matrix $\bar{\mathbf{F}} = 0$. The final system is similar to equation (2.4.26) with $\bar{\mathbf{B}} = \bar{\mathbf{B}}^M = 0$. The number of unknowns are equal to the number of equations in a well posed problem and the system can be solved in one step.

Numerical Algorithm for Nonlinear Interface Connections

The nonlinear constitutive relationships are incorporated in the linear BEM equations and solved using the incremental (quasistatic) algorithm described below.

Nonlinear Solution Algorithm

(a) The boundary (and body force loading if present) is divided into a number of small sub-increments. Ten sub-increments have been found to be sufficient.

Initial Load Step (Iterative Start-up Loop):

(b) Assemble the BEM system equation (2.4.26) using the interface relations calculated in step (d). In the initial assembly step, linear spring connections are assumed at each interface node.

(c) Apply the sub-increment of load to the system equations (2.4.26).

(d) Determine the traction state at each interface node and derive the appropriate interface constitutive relation for that node.
- If the normal traction a the interface is positive, assume a gap opening with zero traction, i.e., $k_{ij}^{ep} = 0$
- If $F < 0$ assume a spring connection
- If $F \leq 0$ assume a nonlinear connection and evaluate the constitutive relationship using the current traction state (from current loading).

(e) Changing the interface connections from a linear spring to nonlinear relationship in the first sub-load step may cause a major redistribution in the tractions on the interface. Therefore, the constitutive relations calculated in step (c) will change. Repeat step (b), (c) and (d) until a coverage solution is achieved for the first load step.

Subsequent Load Steps:

(f) Steps (b), (c), and (d) are repeated once for each additional load step. In general, iteration within a subsequent sub-load step is not required since the constitutive relationships at interface nodes change in a gradual manner once the start-up loop is complete. The solution of any sub-increment is the accumulation of all previous sub-load steps. It is this current solution that is used in step (d) to evaluate the current interface constitutive relationships used in the next sub-incremental load step (step (c)).
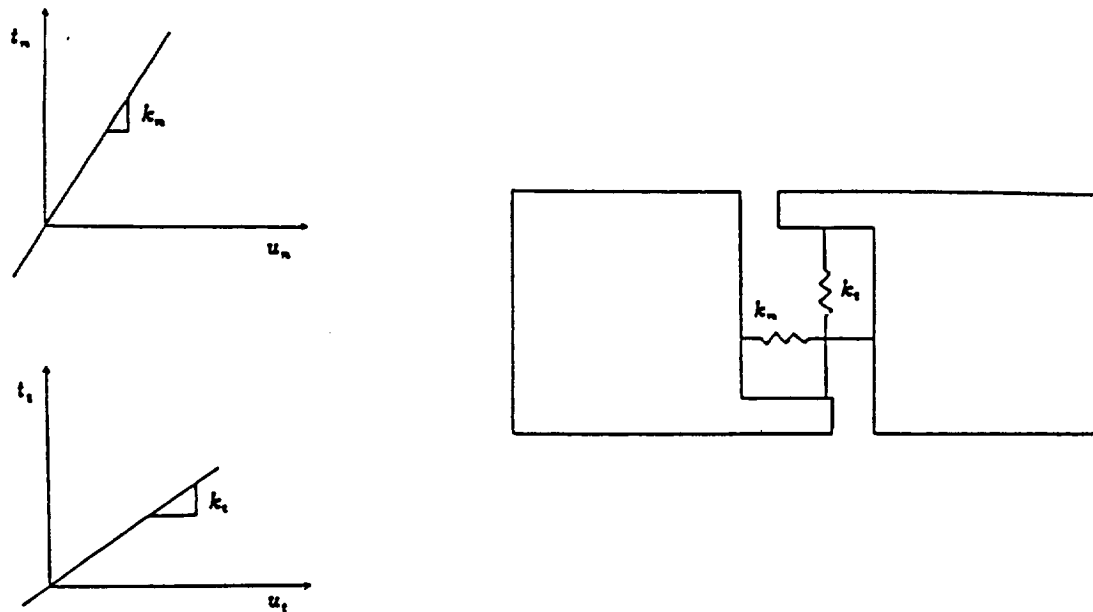
Fig. 2.4.1. Spring Interface Connection
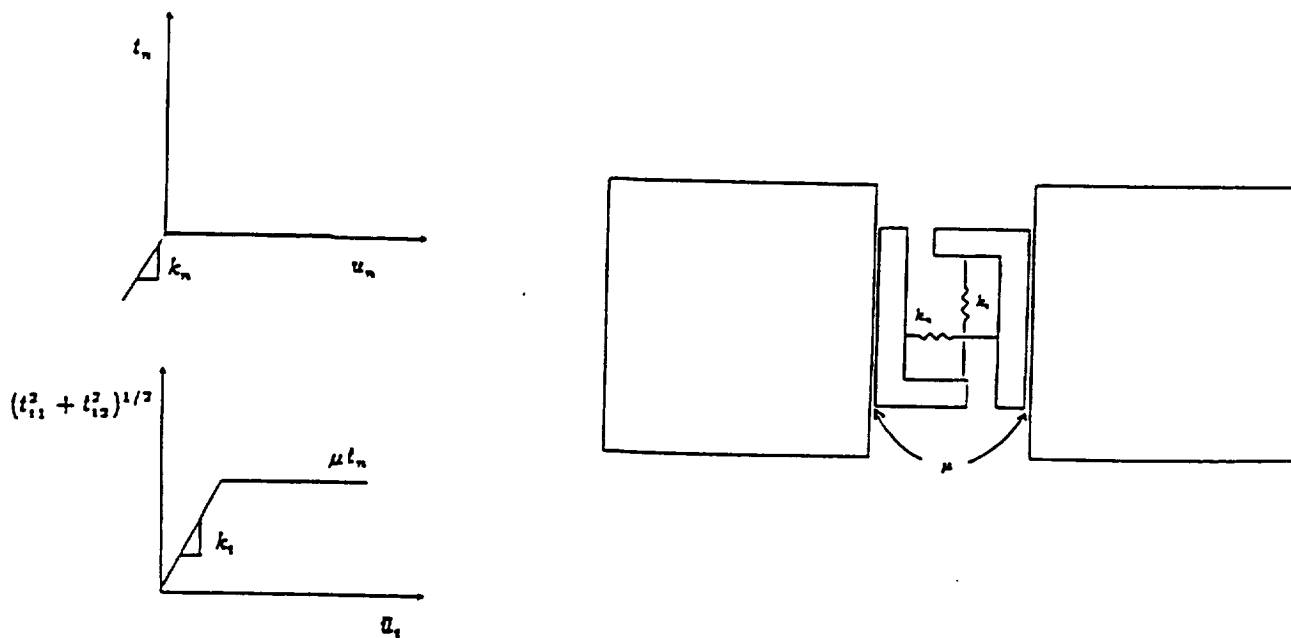
Fig. 2.4.2. Nonlinear Spring/Coulomb-frictional Interface Connection

2.5	TRANSIENT HEAT CONDUCTION AND
	TRANSIENT UNCOUPLED THERMOELASTIC
	BEM FORMULATIONS

The analysis of a body in a transient state is inherently more complicated than a steady-state analysis. The boundary element formulation for transient analysis contains convoluted integrals which must be integrated in time as well as in space. Therefore, a transient BEM analysis is expected to be more complicated and more expensive than its steady-state counterpart. To compound these difficulties, the cost saving techniques developed for the steady-state ceramic composite analyses cannot be employed for the transient case. First, the efficient modified boundary integral formulations and the resultant assembly schemes of Sections 2.2, 2.3, and 2.3 do not lend themselves to transient insert formulations for composites with general material properties. Secondly, due to the complexity of the transient kernel functions, the integration about the circumference of the insert must now be carried out using numerical integration, thus adding to the cost of the analysis.

Nevertheless, an efficient numerical integration scheme as well as an efficient assembly algorithm have been derived for the transient heat conduction and thermoelastic BEM analyses. In the assembly algorithm the equations of the inserts are backsubstituted into the equation system of the composite matrix, hence, reducing the size of the overall system. This is carried out prior to decomposition at a point when the equations for each insert can be handled individually in the most efficient manner. This assembly, as well as the decomposition, is required only for the initial time step. Later time steps require only a new calculation of the system's right-hand side.

Transient Boundary Integral Equation Formulation

The transient, uncoupled thermoelastic boundary integral equation (Dargush and Banerjee, 1991) for the displacement (and temperature) at a point ξ in a composite matrix is

$$C_{\beta\alpha}(\xi)u_\beta(\xi,t) = \int_S \left[g_{\beta\alpha} * t_\beta^Q(X,t) - f_{\beta\alpha} * u_\beta^Q(X,t) \right] dS(X)$$

$$+ \sum_{m=1}^M \int_{S^m} \left[g_{\beta\alpha} * t_\beta^H(X,t) - f_{\beta\alpha} * u_\beta^H(X,t) \right] dS^m(X) \qquad (2.5.1)$$

where

α, β indices varying from 1 to 4 for uncoupled thermoelasticity and equaled to 4 for heat conduction

$C_{\beta\alpha}$ constants determined by the geometry at ξ

u_β, t_β generalized displacement and traction

$$u_\beta = [u_1,\ u_2,\ u_3,\ \theta]^T$$

$$t_\beta = [t_1,\ t_2,\ t_3,\ q]^T$$

θ, q temperature, heat flux

$g_{\alpha\beta}, f_{\alpha\beta}$ generalized displacement and traction kernels

S, S^m are the surfaces of the outer boundary of the matrix and the mth hole (left for fiber)

M is the number of individual fibers

and, for example

$$g_{\beta\alpha} * t_\beta = \int_0^t g_{\beta\alpha}(X, t; \xi, \tau) t_\beta(X, \tau) d\tau$$

denotes a Riemann convolution integral.

Superscripts O and H are used to highlight the quantities associated with the outer surface of the matrix and the quantities associated with the surface of the hole, respectively.

The boundary integral equation for mth insert can be written as

$$C_{\beta\alpha}(\xi) u_\beta(\xi) = \int_{S^m} \left[g_{\beta\alpha}^I * t_\beta^I(X, t) - f_{\beta\alpha}^I * u_\beta^I(X, t) \right] dS^m(X) \tag{2.5.2}$$

where

$g_{\beta\alpha}^I, f_{\beta\alpha}^I$ are the fundamental solutions for a body with material properties corresponding to the mth insert.

$C_{\beta\alpha}$ are constants determined by the geometry at ξ in insert m

u_β^I, t_β^I are the generalized displacement and tractions associated with the mth insert

S^m the surface of the mth insert

In principle, at each instant of time progressing from time zero, these equations can be written at every point on the respective boundary. The collection of the resulting equations could then be solved simultaneously, producing exact values for all the unknown boundary quantities. In reality, of course, discretization is needed to limit this process to a finite number of equations and unknowns. Techniques useful for the discretization of (2.5.1) and (2.5.2) are the subject of the following section.

Numerical Implementation

The boundary integral equations (2.5.1) and (2.5.2) developed in the last section, are exact statements. No approximations have been introduced other than those used to formulate the boundary value problem. However, in order to apply (2.5.1) and (2.5.2) for the solution of practical engineering problems, approximations are required in both time and space. In this section, an overview of a general-purpose, state-of-the-art numerical implementation is presented for transient analysis.

Temporal Discretization

Consider, first, the time integrals represented in (2.5.1) and (2.5.2) as convolutions. Clearly, without any loss of precision, the time interval from zero to t can be divided into N equal increments of duration at Δt.

By assuming that the primary field variables, t_β and u_β, are constant within each Δt time increment, these quantities can be brought outside of the time integral. That is,

$$g_{\beta\alpha} * t_\beta(X, t) = \sum_{n=1}^{N} t_\beta^n(X) \int_{(n-1)\Delta t}^{n\Delta t} g_{\beta\alpha}(X - \xi, t - \tau)d\tau \qquad (2.5.3a)$$

$$f_{\beta\alpha} * u_\beta(X, t) = \sum_{n=1}^{N} u_\beta^n(X) \int_{(n-1)\Delta t}^{n\Delta t} f_{\beta\alpha}(X - \xi, t - \tau)d\tau \qquad (2.5.3b)$$

where the superscript on the generalized tractions and displacements represents the time increment number. Notice, also, that within an increment, these primary field variables are now only functions of position. Since the integrands remaining in (2.5.3) are known in explicit form from the fundamental solutions, the required temporal integration can be performed analytically, and written as

$$G_{\beta\alpha}^{N+1-n}(X - \xi) = \int_{(n-1)\Delta t}^{n\Delta t} g_{\beta\alpha}(X - \xi, t - \tau)d\tau \qquad (2.5.4a)$$

$$F_{\beta\alpha}^{N+1-n}(X - \xi) = \int_{(n-1)\Delta t}^{n\Delta t} f_{\beta\alpha}(X - \xi, t - \tau)d\tau. \qquad (2.5.4b)$$

These kernel functions, $G_{\beta\alpha}^n(X - \xi)$ and $F_{\beta\alpha}^n(X - \xi)$, are detailed in Appendix B. Combining (2.5.3) and (2.5.4) with (2.5.1) and (2.5.2) produces

$$C_{\beta\alpha}(\xi)u_\beta^N(\xi) = \sum_{n=1}^{N} \left\{ \int_S \left[G_{\beta\alpha}^{N+1-n}(X - \xi)t_\beta^n(X) - F_{\beta\alpha}^{N+1-n}(X - \xi)u_\beta^n(X) \right] dS(X) \right.$$

$$\left. + \sum_{m=1}^{M} \int_{S^m} \left[G_{\beta\alpha}^{N+1-n}(X - \xi)t_\beta^n(X) - F_{\beta\alpha}^{N+1-n}(X - \xi)u_\beta^n(X) \right] dS^m(X) \right\} \qquad (2.5.5a)$$

and for the mth insert

$$C_{\beta\alpha}(\xi)u_\beta^N(\xi) = \sum_{n=1}^{N} \left\{ \int_{S^m} \left[G_{\beta\alpha}^{N+1-n}(X - \xi)t_\beta^n(X) - F_{\beta\alpha}^{N+1-n}(X - \xi)u_\beta^n(X) \right] dS^m(X) \right\} \qquad (2.5.5b)$$

which are the boundary integral statements after the application of the temporal discretization.

Spatial Discretization

With the use of generalized primary variables and the incorporation of a piecewise constant time stepping algorithm, the boundary integral equation (2.5.5) begins to show a

strong resemblance to the equations of the steady-state analyses, particularly for the initial time step (i.e., $N = 1$). In this subsection, those similarities will be exploited to develop the spatial discretization for the uncoupled quasistatic problem. This approximate spatial representation will, subsequently, permit numerical evaluation of the surface integrals appearing in (2.5.5). The techniques described here, actually, originated in the finite element literature, but were later applied to boundary elements by Lachat and Watson (1976).

The process begins by subdividing the entire surface of the body (including the inserts) into individual elements of relatively simple shapes. The geometry of each element is, then, completely defined by the coordinates of the nodal points and associated interpolation functions, as described in Section 2.2.

In the present work, the geometry is exclusively defined by quadratic shape functions. On the other hand, the variation of the primary quantities can be described, within an element, by either quadratic or linear shape functions. (The introduction of linear variations proves computationally advantageous in some instances.)

Once this spatial discretization has been accomplished and the body has been subdivided into Q boundary elements and P insert elements, the boundary integral equation can be rewritten for the matrix as

$$C_{\beta\alpha}(\xi)u_\beta^N(\xi) = \sum_{n=1}^{N} \left\{ \sum_{q=1}^{Q} \left[\int_{S^q} G_{\beta\alpha}^{N+1-n}(X(\zeta) - \xi)L_\omega(\zeta)dS^q(X(\zeta)) \right] t_{\beta\omega}^n \right.$$

$$- \sum_{q=1}^{Q} \left[\int_{S^q} F_{\beta\alpha}^{N+1-n}(X(\zeta) - \xi)L_\omega(\zeta)dS^q(X(\zeta)) \right] u_{\beta\omega}^n$$

$$+ \sum_{p=1}^{P} \left[\int_{S^p} G_{\beta\alpha}^{N+1-n}(X(\zeta) - \xi)N_\omega(\zeta)M_\gamma(\theta)dS^p(X(\zeta), \theta) \right] t_{\beta\omega\gamma}^n$$

$$- \sum_{p=1}^{P} \left[\int_{S^p} F_{\beta\alpha}^{N+1-n}(X(\zeta) - \xi)N_\omega(\zeta)M_\gamma(\theta)dS^p(X(\zeta), \theta) \right] u_{\beta\omega\gamma}^n \right\} \qquad (2.5.6a)$$

and

$$C_{\beta\alpha}(\xi)u_\beta^N(\xi) = \sum_{n=1}^{N} \left\{ \sum_{p=1}^{P} \left[\int_{S^p} G_{\beta\alpha}^{N+1-n}(X(\zeta) - \xi)N_\omega(\zeta)M_\gamma(\theta)dS^p(X(\zeta), \theta) \right] t_{\beta\omega\gamma}^n \right.$$

$$- \sum_{p=1}^{P} \left[\int_{S^p} F_{\beta\alpha}^{N+1-n}(X,(\zeta) - \xi)N_\omega(\zeta)M_\gamma(\theta)dS^p(X(\zeta), \theta) \right] u_{\beta\omega\gamma}^n \right\} \qquad (2.5.6b)$$

where

Q is the number of boundary elements on the outer surface of the composite matrix in the region,

P is the number of insert elements in the region,

L_ω represents a two-dimensional shape function,

N_γ represents a one-dimensional shape function,

M_γ is the circular shape function defined in Section 2.2,

t, u are nodal values of generalized traction and displacement, respectively.

In the above equation, the nodal quantities $t_{\beta\omega}^n$ and $u_{\beta\omega}^n$ were brought outside the surface integrals. This positioning of the nodal primary variables outside the integrals is, of course, a key step since now the integrands contain only known functions. However, before discussing the techniques used to numerically evaluate these integrals, a brief discussion of the singularities present in the kernels $G_{\beta\alpha}^n$ and $F_{\beta\alpha}^n$ is in order.

The fundamental solutions to the uncoupled quasistatic problem contain singularities when the load point and field point coincide, that is, is when $r = 0$. The same is true of $G_{\beta\alpha}^n$ and $F_{\beta\alpha}^n$, since these kernels are derived directly from the fundamental solutions. Series expansions of terms present in the evolution functions can be used to deduce the level of singularities existing in the kernels.

A number of observations concerning the results of these expansions should be mentioned. First, as would be expected $F_{\alpha\beta}^1$ has a stronger level of singularity than does the corresponding $G_{\alpha\beta}^1$, since an additional derivative is involved in obtaining $F_{\alpha\beta}^1$ from $G_{\alpha\beta}^1$. Second, the coupling terms do not have as high degree of singularity as do the corresponding non-coupling terms. Third, all of the kernel functions for the first time step could actually be rewritten as a sum of steady-state and transient components. That is,

$$G_{\alpha\beta}^1 = {}^{ss}G_{\alpha\beta} + {}^{tr}G_{\alpha\beta}^1$$

$$F_{\alpha\beta}^1 = {}^{ss}F_{\alpha\beta} + {}^{tr}F_{\alpha\beta}^1$$

Then, the singularity is completely contained in the steady-state portion. Furthermore, the singularity in G_{ij}^1 and F_{ij}^1 is precisely equal to that for elastostatics, while $G_{\theta\theta}^1$ and $F_{\theta\theta}^1$ singularities are identical to those for potential flow. This observation is critical in the numerical integration of the $F_{\alpha\beta}$ kernel to be discussed in the next subsection. However, from a physical standpoint, this means that, at any time t, the closer one moves toward the load point, the closer the quasistatic response field corresponds with a steady-state field. Eventually, when the sampling and load points coincide, the quasistatic and steady-state responses are indistinguishable. As a final item, after careful examination of Appendix B, it is evident that the steady-state components in the kernels $G_{\alpha\beta}^n$ and $F_{\alpha\beta}^n$, with $n > 1$, vanish. In that case, all that remains is a transient portion that contains no singularities. Thus, all singularities reside in the ${}^{ss}G_{\alpha\beta}$ and ${}^{ss}F_{\alpha\beta}$ components of $G_{\alpha\beta}^1$ and $F_{\alpha\beta}^1$, respectively.

Numerical Integration

Having clarified the potential singularities present in the coupled kernels, it is now possible to consider the evaluation of the integrals in equation (2.5.6).

To assist in this endeavor, the following three distinct categories can be identified.

(1) The point ξ does not lie on the element m.

(2) The point ξ lies on the element m, but only non-singular or weakly singular integrals are involved.

(3) The point lies on the element m, and the integral is strongly singular.

In practical problems involving many elements, it is evident that most of the integration occurring in equation (2.5.6) will be of the category (1) variety. In this case, the integrand is always non-singular, and standard Gaussian quadrature formulas can be employed. Sophisticated error control routines are needed, however, to minimize the computational effort for a certain level of accuracy. This non-singular integration is the most expensive part of a boundary element analysis, and, consequently, must be optimized to achieve an efficient solution. In the present implementation, error estimates, based upon the work of Stroud and Secrest (1966), are employed to automatically select the proper order of the quadrature rule. Additionally, to improve accuracy in a cost-effective manner, a graded subdivision of the element is incorporated, especially when ξ is nearby.

The integration over the surface of the hole and insert elements must be carried out using numerical integration. The complexity of the transient kernel function prohibits an analytic integration about the circumference of the insert in a manner similar to the steady-state case. Therefore, numerical integration must be performed in both the longitudinal and the circumferential directions. An efficient integration scheme is adopted in which one-dimensional Gaussian integration formulas are applied independently in the two directions. This allows the subsegmentation and mapping, as well as the order of the Gaussian formulas to vary independently in the two directions so that the accuracy and the cost of the integration can be optimized.

Turning next to category (2), one finds that again Gaussian quadrature is applicable, however, a somewhat modified scheme must be utilized to evaluate the weakly singular integrals. This is accomplished through element subsegmentation about the singular point so that the product of shape function, Jacobian and kernel remains well behaved.

Unfortunately, the remaining strongly singular integrals of category (3) exist only in the Cauchy principal value sense and cannot, in general, be evaluated numerically, with sufficient precision. It should be noted that this apparent stumbling block is limited to the strongly singular portions, $^{ss}F_{ij}$ and $^{ss}F_{\theta\theta}$, of the $F^1_{\alpha\beta}$ kernel. The remainder of $F^1_{\alpha\beta}$, including $^{tr}F^1_{ij}$ and $^{tr}F^1_{\theta\theta}$, can be computed using the procedures outlined for category (2). However, as will be discussed in the next subsection, even category (3) $^{ss}F_{ij}$ and $^{ss}F_{\theta\theta}$ kernels can be accurately determined by employing an indirect 'rigid body' method originally developed by Cruse (1974).

Assembly of Equations

The complete discretization of the boundary integral equation, in both time and space, has been described, along with the techniques required for numerical integration of the

kernels. Now, a system of algebraic equations can be developed to permit the approximate solution of the original quasistatic problem. This is accomplished by systematically writing (2.5.6) at each node on the outer surface, on the surface of the hole, and on the surface of the insert. The ensuing nodal collocation process, then produces a global set of equations given here in matrix form, for time N.

For the nodes on matrix:

$$[G_O^1]\{t_O^N\} - [F_O^1]\{u_O^N\} + [G_H^1]\{t_H^N\} - [F_H^1]\{u_H^n\} = \{D^N\}$$

(2.5.7a)

For the nodes on the inserts:

$$[G_I^1]\{t_I^N\} - [F_I^1]\{u_I^N\} = \{D_I^N\}$$

(2.5.7b)

where $\{D^N\}$ and $\{D_I^N\}$ are the values of the convolution from the previous $N - 1$ time steps defined as

$$-\{D^N\} = \sum_{n-1}^{N-1} \{[G_O^{N+1-n}]\{t_O^n\} - [F_O^{N+1-n}]\{u_O^n\} + [G_H^{N+1-n}]\{t_H^n\} - [F_H^{N+1-n}]\{u_H^n\}\}$$

(2.5.8a)

and

$$-\{D_I^N\} = \sum_{n=1}^{N-1} \{[G_I^{N+1-n}]\{t_I^n\} - [F_I^{N+1-n}]\{u_I^n\}\}$$

(2.5.8b)

In these equations, subscripts O, H and I are used to denote the quantities associated with the outer surface of the matrix, the surface of the hole containing the insert, and surface of the insert, respectively. Furthermore, the $C_{\alpha\beta}(\xi)$ term of each equation is lumped with the respective coefficient in the F^1 matrix.

To calculate the coefficients of the singular points by the 'rigid body' technique, consider now, the first step. Thus, for $N = 1$, equation(2.5.7) becomes

$$[G_O^1]\{t_O^1\} - [F_O^1]\{u_O^1\} + [G_H^1]\{t_H^1\} - [F_H^1]\{u_H^1\} = \{0\}$$

(2.5.9a)

$$[G_I^1]\{t_I^1\} - [F_I^1]\{u_I^1\} = \{0\}.$$

(2.5.9b)

At this point, the coefficients of the F matrices corresponding to the singular node of the equations has not been completely determined due to the strongly singular nature of the kernel function. To determine the values of these coefficients, we first decompose the F_{ij} kernel into transient and steady-state parts. Following Cruse (1974) and, later Banerjee et al (1986) in elastodynamics, the steady-state part of the coefficients can be calculated indirectly by imposing a uniform 'rigid body' generalized displacement field on the same body under steady-state conditions. The steady-state part of the singular coefficients are simply the summation of the non-singular, steady-state coefficients. The remaining transient portion of the coefficients are non-singular, and hence can be evaluated to any desired precision.

For an insert that is perfectly bonded to the matrix, the displacement of the surface on the hole containing the insert is equal to the displacement of the surface of the insert, and the surface tractions are equal in magnitude but opposite in direction.

$$\{u_H^n\} = \{u_I^n\} \qquad (2.5.10a)$$

$$\{t_H^n\} = -\{t_I^n\} \qquad (2.5.10b)$$

Substitution of equation (2.5.10) into equations (2.5.7b) and (2.5.8b) and rearranging yields

$$\{t_H^N\} \doteq -[H][F_I^1]\{u_H^N\} - [H]\{D_I^N\} \qquad (2.5.11a)$$

$$\{D_I^N\} = \sum_{n=1}^{N-1} \left\{ [G_I^{N+1-n}]\{t_H^n\} + [F_I^{N+1-n}]\{u_H^n\} \right\} \qquad (2.5.11b)$$

where

$$[H] = [G_I^1]^{-1}$$

Note, since the boundary integral equation for each insert is independent of the equations for the other inserts, the inversion of $[G_I^1]$ can be reduced to a number of smaller inversions, one corresponding to each insert.

Equation (2.5.11a) can now be back substituted in equation (2.5.7a) to yield

$$[G_O^1]\{t_o^N\} - [F_O^1]\{u_O^N\} - [F^M]\{u_H^N\} = \{D^N\} + [G_H^1][H]\{D_I^N\} \qquad (2.5.12)$$

where

$$[F^M] = [G_H^1][H][F_I^1] + [F_H^1]$$

In a well-posed problem, at time Δt, the set of global generalized nodal displacements and tractions will contain exactly $(4 \times P)$ unknown components (and P unknowns for heat conduction analysis).

Then, as the final stage in the assembly process, equation (2.5.12) can be arranged to form

$$[A^1]\{x^N\} = [B^1]\{y^N\} + \{D^N\} + [G_H^1][H]\{D_I^N\} \qquad (2.5.13)$$

in which

$\{x^N\}$ unknown components of $\{u^N\}$ and $\{t^N\}$
$\{y^N\}$ known components of $\{u^N\}$ and $\{t^N\}$
$[A^1], [B^1]$ associated matrices

Note, the entire matrix $[F^M]$ becomes part of $[A^1]$ since all the (generalized) displacements on the interface are unknown quantities.

Solution

A solution of equation (2.5.13) may be achieved for any time using a time marching algorithm. For the initial time $N = 1$, $\{D^1\} = \{D^1_H\} = \{0\}$ and equation (5.13) can be rewritten as

$$[A^1]\{x^1\} = [B^1]\{y^1\} \qquad (2.5.14)$$

To obtain a solution for the unknown nodal quantities of this equation, a decomposition of matrix $[A^1]$ is required. In general, $[A^1]$ is a densely populated, unsymmetric matrix. The out-of-core solver, utilized here, was developed originally for elastostatics from the LINPACK software package (Dongarra et al, 1979) and operates on a submatrix level. Within each submatrix, Gaussian elimination with single pivoting reduces the block to upper triangular form. The final decomposed form of $[A^1]$ is stored in a direct-access file for reuse in subsequent time steps. Backsubstitution then completes the determination of $\{x^1\}$. Additional information on this solver is available in Banerjee et al (1985).

After returning from the solver routines, the entire nodal response vectors, $\{u^1\}$ and $\{t^1\}$, at time Δt are known. For solutions at later times, a simple marching algorithm is employed. Assuming that the same set of nodal components are unknown as in (2.5.14) for the first time step, equation (2.5.13) is reformulated as

$$[A^1]\{x^2\} = [B^1]\{y^2\} + \{D^2\} + [G^1_H][H]\{D^2_I\} \qquad (2.5.15)$$

Since the right-hand side contains only known quantities, (2.5.15) can be solved for $\{x^2\}$. However, the decomposed form of $[A^1]$ already exists on a direct-access file, so only the relatively inexpensive backsubstitution phase is required for the solution.

The generalization of (2.5.15) to any time step N is simply equation (2.5.13)

$$[A^1]\{x^N\} = [B^1]\{y^N\} + \{D^N\} + [G^1_H][H]\{D^N_I\} \qquad (2.5.16)$$

in which the vectors $\{D^N\}$ and $\{D^N_I\}$ contain summations which represent the effect of past events. By systematically storing all of the matrices and nodal response vectors computed during the marching process, surprisingly little computing time is required at each new time step. In fact, for any time step beyond the first, the only major computational task is the integration needed for form $[G^N]$ and $[F^N]$. Even this process is somewhat simplified, since now the kernels are non-singular. Also, as time marches on, the effect of events that occurred during the first time step diminishes. Consequently, the terms containing $[G^N]$ and $[F^N]$ will eventually become insignificant compared to those associated with recent events. Once that point is reached, further integration is unnecessary, and a significant reduction in the computing effort per time step can be achieved.

It should be emphasized that the entire boundary element method developed, in this section, has involved surface quantities exclusively. A complete solution to the well-posed linear uncoupled quasistatic problem with composite inserts can be obtained in terms of the nodal response vectors, without the need for any volume discretization. In many practical situations, however, additional information, such as, the temperature at interior locations or the stress at points on the boundary, is required. The next section discusses the calculations of these quantities.

Interior Quantities

Once equation (2.5.16) is solved, at any time step, the complete set of nodal displacements and tractions are known. Subsequently, the response at points within the body can be calculated in a straightforward manner. For any point ξ in the interior, the generalized displacement can be determined from (2.5.6a) or (2.5.6b) with $C_{\beta\alpha} = \delta_{\beta\alpha}$. Now, all the nodal variables on the right-hand side are known, and, as long as ξ is not on the boundary nor on the interface between the insert and the matrix. The kernel functions in (2.5.6) remain non-singular. However, when ξ is on the boundary or on the interface, the strong singularity in $^{ss}F_{\beta\alpha}$ prohibits accurate evaluation of the generalized displacement via (2.5.6), and an alternate approach is required. The apparent dilemma is easily resolved by recalling that the variation of surface quantities is completely defined by the elemental shape functions. Thus, for points on the outer boundary of the matrix, the desired relationship is simply

$$u_\alpha^N(\xi) = L_\omega(\zeta) u_{\alpha\omega}^N \tag{2.5.17a}$$

Where L_ω are the shape functions for the appropriate element and ζ are the intrinsic coordinates corresponding to ξ within that element. Obviously, from (2.5.17), neither integration nor the explicit contribution of past events are needed to evaluate generalized boundary displacements.

In many problems, additional quantities, such as heat flux and stress, are also important. The boundary integral equation for heat flux, can be written

$$
\begin{aligned}
q_i^N(\xi) = \sum_{n=1}^{N} \Bigg\{ &\sum_{q=1}^{Q} \left[\int_{S^q} E_{\beta\theta i}^{N+1-n}(X(\zeta) - \xi) L_\omega(\zeta) dS^q(X(\zeta)) \right] t_{\beta\omega}^n \\
&- \sum_{q=1}^{Q} \left[\int_{S^q} D_{\beta\theta i}^{N+1-n}(X(\zeta) - \xi) L_\omega(\zeta) dS^q(X(\zeta)) \right] u_{\beta\omega}^n \\
&+ \sum_{p=1}^{P} \left[\int_{S^p} E_{\beta\theta i}^{N+1-n}(X(\zeta) - \xi) N_\omega(\zeta) M_\gamma(\theta) dS^p(X(\zeta)) \right] t_{\beta\omega\gamma}^n \\
&- \sum_{n=1}^{P} \left[\int_{S^p} D_{\beta\theta i}^{N+1-n}(X(\zeta) - \xi) N_\omega(\zeta) M_\gamma(\theta) dS^p(X(\zeta)) \right] u_{\beta\omega\gamma}^n \Bigg\}.
\end{aligned}
\tag{2.5.18}
$$

where

$$E_{\beta\theta i}^n(X(\zeta) - \xi) = -k \frac{\partial G_{\beta\theta}^n(X(\zeta) - \xi)}{\partial \xi_i} \tag{2.5.19a}$$

$$D_{\beta\theta i}^n(X(\zeta) - \xi) = -k \frac{\partial F_{\beta\theta}^n(X(\zeta) - \xi)}{\partial \xi_i} \tag{2.5.19b}$$

A similar equation can be written for the heat flux inside an insert. This is valid for interior points, whereas, when ξ is on the boundary, the shape functions can again be used. In this latter case,

$$L_\omega(\zeta) q_\omega^N = n_i(\xi) q_i^N(\xi) \tag{2.5.20a}$$

$$\frac{\partial L_\omega(\zeta)}{\partial \zeta} \theta_\omega^N = -\frac{1}{k} \frac{\partial x_i}{\partial \zeta} q_i^N(\xi) \tag{2.5.20b}$$

which can be solved for boundary flux. Meanwhile, interior stresses can be evaluated from

$$
\sigma_{ij}^N(\xi) = \sum_{n=1}^{N} \left\{ \sum_{q=1}^{Q} \left[\int_{S^q} E^{N+1-n}(X(\zeta) - \xi) L_\omega(\zeta) dS^q(X(\zeta)) \right] t_{\beta\omega}^n \right.
$$

$$
- \sum_{q=1}^{Q} \left[\int_{S^q} D_{\beta ij}^{N+1-n}(x(\zeta) - \xi) L_\omega(\zeta) dS^q(X(\zeta)) \right] u_{\beta\omega}^n
$$

$$
+ \sum_{p=1}^{P} \left[\int_{S^p} E_{\beta ij}^{N+1-n}(x(\zeta) - \xi) N_\omega(\zeta) M_\gamma(\theta) dS^p(x(\zeta)) \right] t_{\beta\omega\gamma}^n
$$

$$
\left. - \sum_{p=1}^{P} \left[\int_{S^p} D_{\beta ij}^{N+1-n}(x(\zeta) - \xi) N_\omega(\zeta) M_\gamma(\theta) dS^p(x(\zeta)) \right] u_{\beta\omega\gamma}^n \right\} \qquad (2.5.21)
$$

in which

$$
E_{\beta ij}^n(X(\zeta) - \xi) = \frac{2\mu\nu}{1 - 2\nu} \delta_{ij} \frac{\partial G_{\beta\ell}^n}{\partial \xi_\ell} + \mu \left(\frac{\partial G_{\beta i}^N}{\partial \xi_j} + \frac{\partial G_{\beta j}^n}{\partial \xi_i} \right) - \beta \delta_{ij} G_{\beta\theta}^n \qquad (2.5.22a)
$$

$$
D_{\beta ij}^n(X(\zeta) - \xi) = \frac{2\mu\nu}{1 - 2\nu} \delta_{ij} \frac{\partial F_{\beta\ell}^n}{\partial \xi_\ell} + \mu \left(\frac{\partial F_{\beta i}^N}{\partial \xi_j} + \frac{\partial F_{\beta j}^n}{\partial \xi_i} \right) - \beta \delta_{ij} F_{\beta\theta}^n. \qquad (2.5.22b)
$$

A similar equation could be derived if stress measurements are required inside an insert. Equation (2.5.22) is, of course, developed from (2.5.6). Since strong kernel singularities appear when (2.5.22) is written for outer boundary points, an alternate procedure is needed to determine surface stress. This alternate scheme exploits the interrelationships between generalized displacement, traction, and stress and is the straightforward extension of the technique typically used in elastostatic implementation (Dargush and Banerjee, 1990). Specifically, the following can be obtained

$$
n_j(\xi) \sigma_{ij}^N = L_\omega(\zeta) t_{i\omega}^N \qquad (2.5.23a)
$$

$$
\sigma_{ij}^N(\xi) - \frac{D_{ijkl}^e}{2} \left(u_{k,l}^N(\xi) + u_{l,k}^N(\xi) \right) = -\beta \delta_{ij} L_\omega(\zeta) u_{\theta\omega}^N \qquad (2.5.23b)
$$

$$
\frac{\partial x_j}{\partial \zeta} u_{i,j}^N(\xi) = \frac{\partial L_\omega(\zeta)}{\partial \zeta} u_{i\omega}^N \qquad (2.5.23c)
$$

in which $u_{\theta\omega}^N$ is obviously the nodal temperatures, and,

$$
D_{ijkl}^e = \lambda \delta_{ij} \delta_{kl} + 2\mu \delta_{ik} \delta_{jl}.
$$

Equations (2.5.23) form an independent set that can be resolved numerically for $\sigma_{ij}^N(\xi)$ and $u_{i,j}^N(\xi)$ completely in terms of known nodal quantities $u_{\alpha\omega}^N$ and $t_{\alpha\omega}^N$, without the need for kernel integration nor convolution.

2.6 | NONLINEAR MATERIAL FORMULATION

Due to the material discontinuity in a composite structure, large localized stress gradients develop on the fiber-matrix interface when a specimen is loaded. These large stress gradients induce plastic yielding and creep in a composite about the fibers. It is therefore, important to include these material nonlinearities in an analysis of a composite structure loaded beyond its elastic limit. As a first step in this analysis, a formulation to account for the plastic yielding of the composite matrix is developed. The nonlinear effects are included in the boundary element formulation through a volume integral.

In this chapter, new methods are developed for the efficient evaluation of the volume integrals which appear in the integral formulations. An elastoplastic-fracture constitutive law is used to model the nonlinear behavior of ceramic composite material, and an iterative control algorithm is implemented for the solution of the nonlinear system.

Integral Equation Formulation for Elastoplasticity and Creep

The governing equation of plasticity can be written in the following form [Banerjee and Butterfield, 1981]:

$$(\lambda + \mu)\dot{u}_{j,ij} + \mu\dot{u}_{i,jj} = \dot{\sigma}^c_{ij,j} \tag{2.6.1}$$

$$i,j = 1,2,3$$

where λ and μ are Lamé constants, $\dot{u}_i$ is displacement rate and $\dot{\sigma}^c_{ii}$ is an initial stress rate (or a corrective stress rate) resulting from the nonlinearities present in the plastic domain. For the purpose of the plasticity algorithm the initial stress rate is defined as

$$\dot{\sigma}^c_{ij} = D^e_{ijkl}\dot{\epsilon}_{kl} - D^{ep}_{ijkl}\dot{\epsilon}_{kl}$$

where D^e_{ijkl} and D^{ep}_{ijkl} is the elastic and elastoplastic constitutive tensors, respectively, and $\dot{\epsilon}_{kl}$ is the strain rate.

The boundary integral equation for the displacement at a point ξ in the matrix can be expressed as

$$C_{ij}(\xi)\dot{u}_i(\xi) = \int_S [G^M_{ij}(x,\xi)\dot{t}^O_i(x) - F^M_{ij}(x,\xi)\dot{u}^O_i(x)]dS(x) + \int_V B^M_{ikj}(z,\xi)\dot{\sigma}^C_{ik}(z)dV(z)$$

$$+ \sum_{n=1}^{N} \int_{S^n} [G^M_{ij}(x,\xi)\dot{t}^H_i(x) - F^M_{ij}(x,\xi)\dot{u}^H_i(x)]dS^n(x) \tag{2.6.2}$$

where

G^M_{ij}, F^M_{ij}, and B^M_{ijk} are the fundamental solutions of a ceramic matrix of infinite extent,

C_{ij} are constants determined by the geometry at ξ,

$\dot{u}_i, \dot{t}_i$ are the displacement and traction rates,

S, S^n are the surfaces of the outer boundary and the nth hole (encompassing the fiber), respectively and

N is the number of individual fibers.

Superscript O and H identify the quantities of the outer surface of the matrix and the quantities on the surface of the hole, respectively.

A similar equation can be written for the displacement rate of a point ξ in the nth fiber as

$$C_{ij}^F \dot{u}_i(\xi) = \int_{S^n} [G_{ij}^F(x,\xi)\dot{t}_i^F(x) - F_{ij}^F(x,\xi)\dot{u}_i^F(x)]dS^n(x) \tag{2.6.3}$$

where

G_{ij}^F, F_{ij}^F are the fundamental solutions of the nth fiber,

C_{ij}^F are constants determined by the geometry at ξ,

$\dot{u}_i^F, \dot{t}_i^F$ are displacement and traction rates associated with the nth fiber

S^n the surface of the nth fiber

Note, each fiber may have different material properties.

For a perfectly bonded fiber-matrix interface the displacement rates of the matrix and the fiber are equal and the traction rates are equal and opposite:

$$\dot{u}_i^H(x) = \dot{u}_i^F(x) \tag{2.6.4a}$$

$$\dot{t}_i^H(x) = -\dot{t}_i^F(x) \tag{2.6.4b}$$

As in the derivation of the integral equation for elastostatic fiber composite analysis of Section 2.2, the Poisson's ratio of the fiber and the Poisson's ratio of the matrix is assumed to be equal. This leads to the following relation:

$$F_{ij}^F(x,\xi) = -F_{ij}^H(x,\xi) \tag{2.6.5}$$

Finally, using relations (2.6.4) and (2.6.5), equation (2.6.3) can be backsubstituted in equations (2.6.2) to yield the integral equation for the displacement rate at a point ξ in a fiber composite structure in which material nonlinearities are present in the matrix:

$$\bar{C}_{ij}(\xi) = \int_S [G_{ij}^M(x,\xi)\dot{t}_i^O(x) - F_{ij}^M(x,\xi)\dot{u}_i^O(x)]dS(x) + \int_V B_{ikj}^M(z,\xi)\dot{\sigma}_{ik}^C(z)dV(z)$$

$$+ \sum_{n=1}^{N} \int_{S^n} \bar{G}_{ij}(x,\xi)\dot{t}_i^H(x)dS^n(x) \tag{2.6.6}$$

where

$$\bar{G}_{ij}(x,\xi) = G_{ij}^H(x,\xi) - G_{ij}^F(x,\xi)$$

$\bar{C}_{ij}(\xi)$ are constants dependent on the geometry for a point ξ on the outer boundary, and

$\bar{C}_{ij}(\xi) = \delta_{ij}$ for a point ξ in the interior of the composite.

Cylinder Volume Cells

The volume integral in equation (2.6.6) is evaluated assuming a variation based on a 30 node shape function over a cylindrical volume cell (Figure 2.6.1). The initial stress rates have a variation through the volume cell which are quadratic in the longitudinal direction, linear in the radial direction, and trigometric in the circumferential direction. The variation in the circumferential direction is based on a 5-noded circular shape function shown in Figure 2.6.2. It is similar to the 3-noded circular shape function used for the displacement and traction rates on the fiber/hole, however, in addition to the constant, *sine* and *cosine* terms, the 5-noded function contains $\sin 2\theta$ and $\cos 2\theta$ terms. Hence, the shape function can model initial stress rates which have two peak values in the circumferential direction. In two-dimensional numerical experimentation, this was observed as a minimum requirement for reasonable approximation of the actual initial stress rate variation.

The integration is carried out analytically about the circumference of the volume cell to keep numerical costs down. However, due to the complexity of the resulting integrand, numerical integration was employed in the radial and longitudinal directions.

Interior Stress Rate Equations

The solution algorithm in a plasticity analysis requires the determination of stress (and strain) rates at nodes of the volume cell. The integral equation for stress (and strain) rates in the composite matrix are derived from the displacement rate integral equations (2.6.2) through the application of the strain-displacement equation and the nonlinear stress-strain relations, i.e.,

$$\sigma_{ij} = D^e_{ijkl}\epsilon_{kl} - \sigma^C_{ij} \tag{2.6.7a}$$

Nonlinear effects are incorporated in the stress rate integral equation via a volume integral. This volume integral is strongly singular, however, and special considerations must be given to this integral as described in Henry and Banerjee (1987).

The integral equation for stress rates, however, are strongly singular for a point on the fiber-matrix interface. In this instance the stress rates are best determined using a boundary stress calculation similar to equation (2.2.13). However, in a nonlinear analysis the nonlinear stress-strain rate constitutive relationship involving initial stress rates [Henry, 1987] must be used. The calculation is very efficient because the calculation does not require surface nor volume integration.

Von Mises Model

This model was developed from observations on metal behavior and assumes that yielding begins when the distortional energy reaches the distortional energy at yield in simple tension. The yield criterion is given by

$$F(J_2) = \frac{1}{2}S_{ij}S_{ij} - \frac{1}{3}\sigma_o^2 = 0 \tag{2.6.7b}$$

$$S_{ij} = \sigma_{ij} - \frac{1}{3}\delta_{ij}\sigma_{kk}$$

σ_o = uniaxial yield strength in simple tension

Plastic flow is defined through a flow rule which relates the plastic strain rate tensor to the gradient of a plastic potential through the stress point. In metals, this potential is considered identical to the yield function and the associated plastic strain rate becomes normal to the yield surface at a stress point. This normality condition is expressed as

$$\dot{\sigma}_{ij} = \dot{\lambda}\frac{\partial F}{\partial \sigma_{ij}}$$

where λ is a non-negative scalar variable which depends upon the current stress state and the past history of loading. An isotropic, strain hardening rule is utilized to describe the subsequent yield surface resulting from continuous plastic distortion.

The isotropic hardening theory assumes that the yield surface uniformly expands about the origin in stress space while maintaining its shape, center and orientation, during plastic flow.

For an isotropic hardening material, the yield function can be written as

$$F(\sigma_{ij}, \epsilon_{ij}^p, h) = 0$$

where h is the hardening parameter that represents the loading history in the plastic range.

Utilizing the above stated concepts along with a consistency condition that every subsequent stress point beyond the yield stress point must lie on an expanded yield surface, the increment of stress for a given mechanical strain increment is obtained as

$$\dot{\sigma}_{ij} = D_{ijkl}^{ep}\dot{\epsilon}_{kl}^m$$

where

$$\dot{\epsilon}_{kl}^m = \dot{\epsilon}_{kl}^{elastic} + \dot{\epsilon}_{kl}^{plastic}$$

and D_{ijkl}^{ep} is the plastic modulus tensor which can be explicitly written as

$$D_{ijkl}^{ep} = 2\mu\left[\delta_{ik}\delta_{jl} + \frac{\nu}{1-2\nu}\delta_{ij}\delta_{kl} - \frac{3S_{ij}S_{kl}}{2\sigma_o^2(1+h/3\mu)}\right]$$

Creep Model

Any generic viscoplastic-creep model can be described by

$$\dot{\varepsilon}^{vp} = < rF^N > \frac{\partial Q}{\partial \sigma_{ij}}$$

where Q is the plastic potential.

Multiaxial creep behavior in BEST-CMS is defined using a power law in the form

$$\dot{\varepsilon}^c = \gamma \sigma_{eq}^n$$

where the equivalent stress is defined as

$$\sigma_{eq} = \left(\frac{3}{2} S_{ij} S_{ij}\right)^{\frac{1}{2}}$$

and γ and n are material parameters. The creep strain rates are then expressed as

$$\dot{\varepsilon}_{ij}^C = \dot{\varepsilon}^c \frac{\partial Q}{\partial \sigma_{ij}} = \gamma \sigma_{eq}^n \frac{\partial Q}{\partial \sigma_{ij}}$$

where the dot indicates a derivative with respect to time.

More general viscoplastic models can easily be inserted in BEST-CMS, for example

$$\dot{\varepsilon}_{ij}^C = [(m+1)t^m] \left[(k+1)\dot{\varepsilon}_r^C (\sigma/\sigma_R)^{n/(k+1)} (\varepsilon^c)^{k/(k+1)}\right] \frac{\partial Q}{\partial \sigma_{ij}}$$

where σ_R and $\dot{\varepsilon}_R^C$ are reference values.

Plasticity-Fracture Constitutive Model for Ceramic Composites

An elastic-plastic strain hardening-fracture model (Chen, 1975) is used to model the inelastic behavior of the ceramic material. The yield function describing the inelastic behavior is bounded by two surfaces: the elastic limit or initial discontinuous surface; and a failure surface. When the stress level violates the elastic limit, plastic yielding occurs with strain hardening. At this point unloading would behave linear-elastically, however, permanent deformation will have occurred. A new elastic limit surface, uniquely defined by the invariants p and J_2 (described below), corresponds to the highest stress state achieved in prior loading history. Loading can continue until the failure surface is reached. At this time the material will fracture and the stress will be reduced to zero at this point.

An important feature that this model exhibits to accurately represent ceramic material behavior is its ability to sustain stress in compression many times greater than in tension. Further, this phenomenon is magnified during plastic hardening since both the isotropic

and kinematic hardening rules are employed. The strain-hardening model is illustrated in Figures 2.6.3 and 2.6.4.

The loading function shown in Figure 2.6.4 has the following form:

in the compression domain:

$$f(\sigma_{ij}) = \frac{J_2 + \frac{1}{3}Ap}{1 - \frac{1}{3}Bp} = \tau^2 \tag{2.6.8a}$$

in the tension and tension-compression domains:

$$f(\sigma_{ij}) = \frac{J_2 - \frac{1}{6}p^2 + \frac{1}{3}Ap}{1 - \frac{1}{3}Bp} = \tau^2 \tag{2.6.8b}$$

where

$p = (\sigma_x + \sigma_y + \sigma_z)/3,$

$J_2 = \frac{1}{6}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2\right] + \sigma_{xy}^2 + \sigma_{xz}^2 + \sigma_{yz}^2,$

$A = \frac{A_0\tau_u^2 - A_u\tau_0^2}{\tau_u^2 - \tau_0^2},$

$B = \frac{A_u - A_0}{\tau_u^2 - \tau_0^2},$

τ is a function of hardening, and

A_0, τ_0, A_u, τ_u are material constants which can be obtained from uniaxial and biaxial tests.

By employing the normality condition the final form of the constitutive equation relating the stress increment to the strain increment can be derived.

$$\dot{\sigma}_{kl} = \left[D^e_{klij} - PD^e_{klmn}\frac{\partial f}{\partial \sigma_{mn}}\frac{\partial f}{\partial \sigma_{qr}}D^e_{qrij}\right]\dot{\varepsilon}_{ij} \tag{2.6.9}$$

where

$\frac{1}{P} = 2\sigma_e H'\left(\frac{\partial f}{\partial \sigma_{pt}}\frac{\partial f}{\partial \sigma_{pt}}\right)^{\frac{1}{2}} + \frac{\partial f}{\partial \sigma_{rs}}D^e_{rsvw}\frac{\partial f}{\partial \sigma_{vw}}$

D^e_{klij} is the elastic constitutive matrix relation

H' is the current slope of the equivalent stress-equivalent plastic strain hardening curve, and

$\sigma_e = \sqrt{f}$ is the definition of equivalent stress

Assembly of Equations

The discretization and integration of the integral equations for displacement and stress rates follows the procedure outlined in Section 2.2. After the integration of the kernel functions, the next critical step in the formulation is the assembly of the system equations. Here, efficiency is of utmost importance. The approach to writing an efficient algorithm is to keep the number of system equations to a minimum by eliminating all unnecessary

unknowns from the system. The strategy is to retain in the system only traction variables on the fiber-matrix interface. This is in contrast to a general multi-region problem where both displacement and tractions are retained on an interface. The elimination of the displacement on the interface is achieved through a backsubstitution of the fiber equations in the system equations which are made up exclusively from equations written for the composite matrix (on the outer surface and on the surface of the holes). The procedure is described below.

Equation (2.6.6) is used to generate a system of equations for nodes on the outer surface of the composite matrix and for nodes on the surface of the holes containing the fibers. Written in matrix form we have

On the Matrix Outer Surface:
$$\mathbf{G}^M \dot{t}^O - \mathbf{F}^M \dot{u}^O + \bar{\mathbf{G}} \dot{t}^H + \mathbf{B}^M \dot{\sigma}^C = 0 \qquad (2.6.10a)$$

On the Matrix Hole Surface:
$$\mathbf{G}^M \dot{t}^O - \mathbf{F}^M \dot{u}^O + \bar{\mathbf{G}} \dot{t}^H + \mathbf{B}^M \dot{\sigma}^C = \mathbf{I} \dot{u}^H \qquad (2.6.10b)$$

where

t^O and u^O are traction and displacement rates

$\dot{t}^H$ and $\dot{u}^H$ are traction and displacement rates

σ^C is the initial stress rate vector

$\mathbf{I}$ is the identity matrix

$\mathbf{G}^M$ and $\mathbf{F}^M$ matrices contain coefficients from the integration over the outer boundary. $\bar{\mathbf{G}}$ matrix contains coefficients integrated about the fiber/hole

$\mathbf{B}^M$ matrix contains coefficients from the integration over the volume cells

Our goal is to eliminate $\dot{u}^H$ from the system. To this end, equation (2.6.3) is written for every node on a fiber, collocating slightly outside the boundary of the fiber [at a distance of (1.25)*(fiber radius)] where $C_{ij}^F(\xi) = 0$.

$$\mathbf{F}^{F2} u^F = \mathbf{G}^{F2} t^F$$

Superscript $\mathbf{F2}$ identifies the equations written at points located slightly outside the boundary of the fibers.

Noting $\dot{u}^H = \dot{u}^F$ and $\dot{t}^H = -\dot{t}^F$ we have

$$\mathbf{F}^{F2} \dot{u}^H = -\mathbf{G}^{F2} \dot{t}^H \qquad (2.6.11)$$

Post multiplying equation (2.6.10b) by the $\mathbf{F}^{F2}$ matrix in equation (2.6.12) yields

$$\mathbf{F}^{F2}\mathbf{G}^M \dot{t}^O - \mathbf{F}^{F2}\mathbf{F}^M \dot{u}^O + \mathbf{F}^{F2}\bar{\mathbf{G}} \dot{t}^H + \mathbf{F}^{F2}\mathbf{B}^M \dot{\sigma}^C = \mathbf{F}^{F2} \dot{u}^H \qquad (2.6.12)$$

Equation (2.6.11) can now be set equal to equation (2.6.12) and the final form of the system is derived.

On Outer Surface: $\quad \mathbf{G}^M \dot{t}^O - \mathbf{F}^M \dot{u}^O + \bar{\mathbf{G}} \dot{t}^H + \mathbf{B}^M \dot{\sigma}^C = \mathbf{O}$

On Hole: $\quad \mathbf{F}^{F2} \mathbf{G}^M \dot{t}^O - \mathbf{F}^{F2} \mathbf{F}^M \dot{u}^O + (\mathbf{F}^{F2} \bar{\mathbf{G}} + \mathbf{G}^{F2}) \dot{t}^H + \mathbf{F}^{F2} \mathbf{B}^M \dot{\sigma}^C = \mathbf{O}$ $\qquad$ (2.6.13)

At every point on the outer surface, either the traction or the displacement rate is specified and on the surface of the hole only the traction rates are retained. Therefore, the number of displacement rate equations in the system are equal to the final number of boundary unknowns. The initial stress rate vector, however, is unknown and must be determined by a quasistatic, iterative solution algorithm. This requires the use of stress rate equations and a constitutive law. The stress rate equation can be written in matrix form as

$$\dot{\sigma} = \mathbf{G}^\sigma \dot{t}^O - \mathbf{F}^\sigma \dot{u}^O + \mathbf{G}^\sigma \dot{t}^H - \mathbf{F}^\sigma \dot{u}^H + \mathbf{B}^\sigma \dot{\sigma}^C \qquad (2.6.14)$$

Note, the displacement rate vector u^H for the hole is known from equation (2.6.10b) and could be eliminated from equation (2.6.14). For numerical efficiency, however, a two step procedure is used in which u^H is first determined using equation (2.6.10b) and then substituted in equation (2.6.14). Nevertheless, for illustration purposes, a backsubstitution of equation (2.6.10b) into equation (2.6.14) will be implied. Hence, the displacement rate and stress rate equations (2.6.13) and (2.6.14) are assembled by collecting the known and unknown values of traction and displacement rates and their coefficients together. The final system equations can be cast as:

$$\mathbf{A}^b \dot{x} = \mathbf{B}^b \dot{y} + \mathbf{C}^b \dot{\sigma}^C \qquad (2.6.15a)$$

$$\dot{\sigma} = \mathbf{A}^\sigma \dot{x} + \mathbf{B}^\sigma \dot{y} + \mathbf{C}^\sigma \dot{\sigma}^C \qquad (2.6.15b)$$

where $\dot{x}$ is the vector of unknown variables at boundary and interface nodes; $\dot{y}$ is the vector of known variables; $\dot{\sigma}^C$ is the vector of initial stress rates; $\mathbf{A}^b, \mathbf{B}^b, \mathbf{C}^b$ are the coefficient matrices of the boundary (displacement rate) system equations (2.6.13); and $\mathbf{A}^\sigma, \mathbf{B}^\sigma, \mathbf{c}^\sigma$ are the coefficient matrices of the stress rate equation (2.6.14). It should be noted that $\mathbf{A}^b$ is a square matrix. Furthermore, in a substructured system the matrices $\mathbf{A}^b$ and $\mathbf{B}^b$ are block banded while matrices $\mathbf{C}^b, \mathbf{A}^\sigma, \mathbf{B}^\sigma$, and $\mathbf{C}^\sigma$ are block diagonal.

Nonlinear Iterative Solution Algorithm

The algorithm described below provides the solution of the system equations given by (2.6.15). This requires complete knowledge of the initial stress rate distribution $\dot{\sigma}^C$ within the yielded region that is induced by the imposition of the boundary loading. Unfortunately the nonlinear initial stress rates are not known a priori for a particular load increment and therefore must be calculated within each loading stage.

An important feature incorporated in the iterative algorithm used in elastoplastic analysis is an acceleration scheme based on the initial stress rates generated by the past history. In this procedure, the path followed by the previous load increment is used to extrapolate the initial stress rates at the beginning of the current increment before the start of the iterative operations. This results in substantial reduction in computer time. This procedure is graphically illustrated in Figure 2.6.5 for a simple tension problem with a variable hardening parameter. The initial loading from point A generates initial stress rates which are distributed during the iterations to arrive at the solution at B. Upon loading from state B the extrapolated path BC is followed. At C the resulting initial stress rates are distributed to reach state D.

The steps of the incremental plasticity (and creep) algorithm is described below:

1. Obtain the elastic solution $\dot{x}$ for an arbitrary increment of boundary loading $\dot{y}$ from

$$\mathbf{A}^b\dot{x} = \mathbf{B}^b\dot{y} \qquad (2.6.16a)$$

and determine the stress rates at the nodes of the volume cells from

$$\dot{\sigma} = \mathbf{A}^\sigma\dot{x} + \mathbf{B}^\sigma\dot{y} \qquad (2.6.16b)$$

2. For creep analysis skip to step 3, for elastoplstic analysis scale the elastic solution such that the node with the highest stress rate is at yield. In the case of nonproportional loading, this onset of yielding cannot be reached by scaling. Instead, incremental loads must be applied until yielding is reached.

3. Impose a small load increment $\dot{y}$ for elastoplastic analysis and add the (usually less than 5% of the yield load) value of the initial stress rates accumulated from the previous load step and evaluate $\dot{x}$ and σ using

$$\mathbf{A}^b\dot{x} = [\mathbf{B}^b\dot{y} + \mathbf{C}^b\dot{\sigma}^C] \qquad (2.6.17a)$$

$$\dot{\sigma} = \mathbf{A}^\sigma\dot{x} + \mathbf{B}^\sigma\dot{y} + \mathbf{C}^\sigma\dot{\sigma}^C \qquad (2.6.17b)$$

where $\dot{\sigma}^c$ is the estimated initial stress rates. If no prior plastic history exists the value of the estimated initial stress rates are zero.

4. Accumulate all incremental quantities of stress, traction, and displacement rates and use Eq. (2.6.7a) to calculate the strain for this increment.

5. Evaluate the current constitutive matrix using the new stress rate history and calculate the current initial stress rates via

$$\dot{\sigma}^C_{ij} = (D^e_{ijkl} - D^{ep}_{ijkl})\dot{\varepsilon}_{kl} \quad \text{for elastoplastic analysis} \qquad (2.6.18a)$$

or

$$\dot{\sigma}^c_{ij} = D^e_{ijkl} \left[\int \dot{\epsilon}^c_{kl} dt \right] \quad \text{for creep analysis} \tag{2.6.18b}$$

Accumulate the initial stress rate history of this load increment for use as the estimated initial stress rate of the next load increment in step 3.

6. For creep analysis go to step 7, for elastoplastic analysis if the current increment of initial stress rates computed in Eq. (2.6.18) is greater than a prescribed tolerance (normally 0.005 times the yield stress) then calculate the incremental quantities ($\dot{x}$ and $\dot{\sigma}$) due to these rates using

$$\mathbf{A}^b \dot{\mathbf{x}} = \mathbf{C}^b \dot{\sigma}^c \quad \text{and} \quad \dot{\sigma} = \mathbf{A}^\sigma \dot{\mathbf{x}} + \mathbf{C}^\sigma \dot{\sigma}^c$$

and return to step 4 for the next iteration. If the value is less than the prescribed tolerance, go to step 7. Note the boundary loading is zero for this calculation. If the number of iterations is greater than a specified limit (usually 50) the system is assumed to have reached the state of failure.

7. Return to step 3 and apply the next load increment and the accumulated initial stress rates from this load step. (If the size of the load increment changes the estimated initial stress rates should be scaled proportionally.)

Any residual initial stress at the end of the iteration is carried forward and applied to the system with the next load increment.
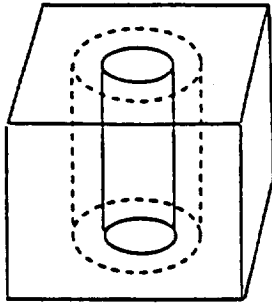
$$C_{ij}(\xi)\dot{u}_i(\xi) = \int_S [G_{ij}^M(x,\xi)\dot{t}_i^O(x) - F_{ij}^M(x,\xi)\dot{u}_i^O(x)]dS(x) + \int_V B_{ikj}^M(x,\xi)\dot{\sigma}_{ik}^C(x)dV(x)$$

$$+ \sum_{n=1}^{N} \int_{S^n} [G_{ij}^M(x,\xi)\dot{t}_i^H(x) - F_{ij}^M(x,\xi)\dot{u}_i^H(x)]dS^n(x)$$

$$C_{ij}^F \dot{u}_i(\xi) = \int_{S^n} [G_{ij}^F(x,\xi)\dot{t}_i^F(x) - F_{ij}^F(x,\xi)\dot{u}_i^F(x)]dS^n(x)$$

$$\hat{C}_{ij}(\xi) = \int_S [G_{ij}^M(x,\xi)\dot{t}_i^O(x) - F_{ij}^M(x,\xi)\dot{u}_i^O(x)]dS(x) + \int_V B_{ikj}^M(x,\xi)\dot{\sigma}_{ik}^C(x)dV(x)$$

$$+ \sum_{n=1}^{N} \int_{S^n} \hat{G}_{ij}(x,\xi)\dot{t}_i^H(x)dS^n(x)$$

where

$$\hat{G}_{ij}(x,\xi) = G_{ij}^H(x,\xi) - G_{ij}^F(x,\xi)$$
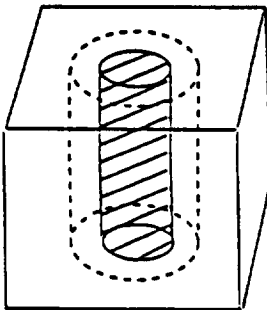
$$i,j = 1,2,3$$

Fig. 2.6.1 Integral Equation Formulation for the Nonlinear Analysis of Fiber Composite Materials. A 30-Node Cylindrical Volume Cell is Generated Automatically.
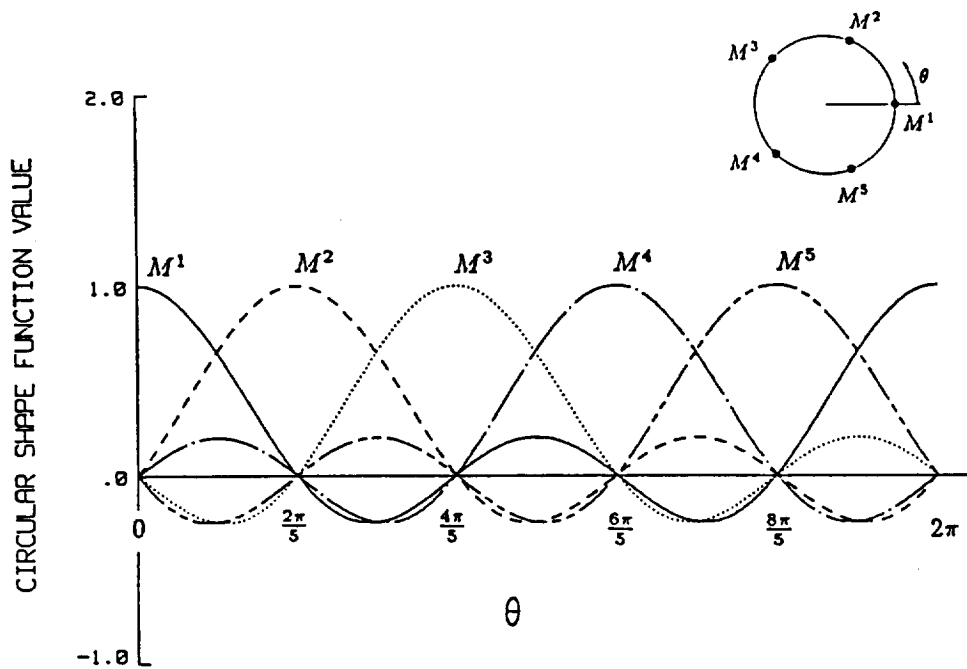
Fig. 2.6.2 Value of the 5-Noded Circular Shape Function about the Circumference of the Cylindrical Volume Cell
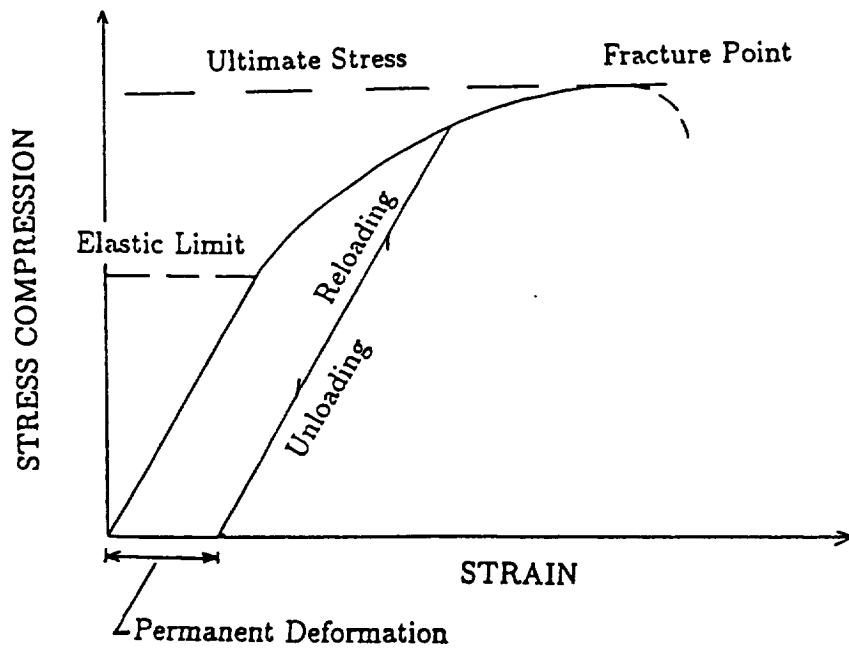
Fig. 2.6.3 Typical Stress-Strain Curve in Equivalent Stress-Strain Space
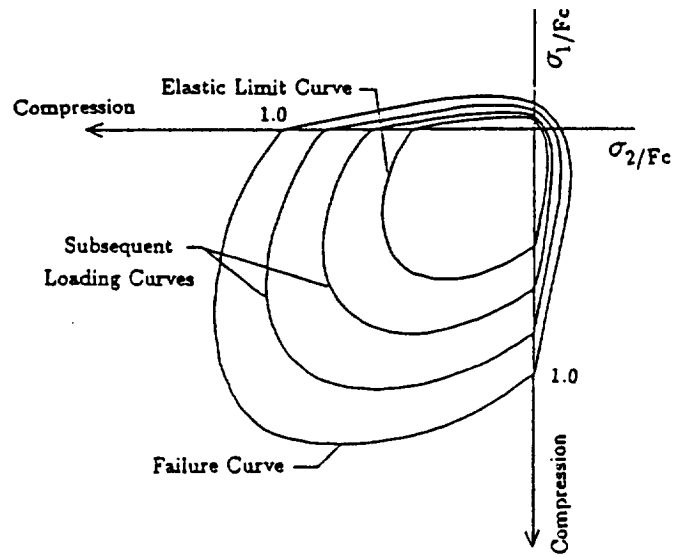
Fig. 2.6.4 Loading Function in Biaxial Principal Stress Space
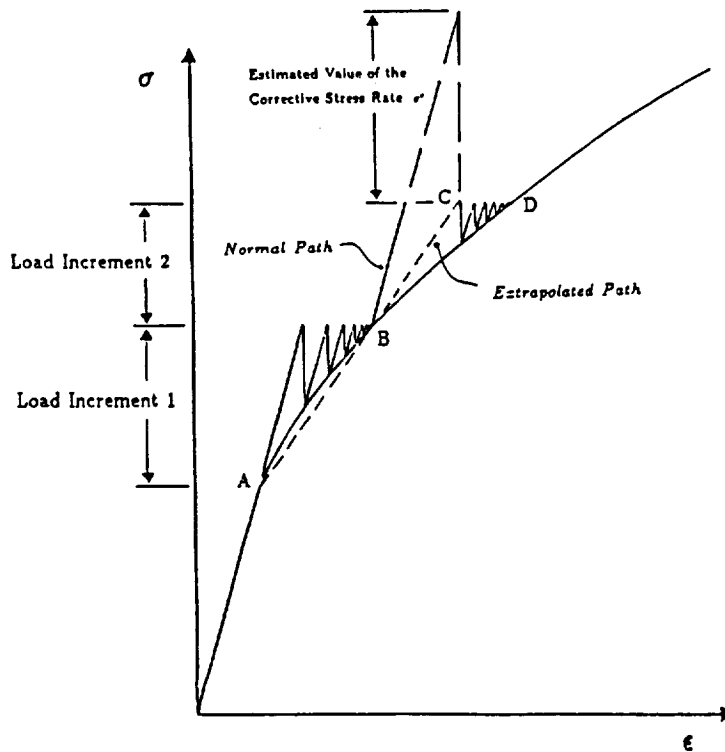
Fig. 2.6.5 Accelerated Iterative Plasticity Algorithm

3.0 | COMPUTER PROGRAM DEVELOPMENT

3.1 | INTRODUCTION

The goal of the computer program developed for ceramic composites is the accurate and efficient implementation of the formulation described in Section 2. Of equal importance is the degree of generality required in the definition of component geometry, loading and material properties.

For this reason the ceramic composite formulation has been implemented in the three-dimensional boundary element computer code 'BEST3D' (Boundary Element Stress Technology - Three-dimensional) which was developed for NASA by Pratt and Whitney and SUNY/Buffalo under contract NAS3-23697. Since its development, BEST3D has been proven to be a highly accurate and numerically efficient boundary element program.

The development of the computer program 'BEST-CMS' is discussed in the following sections.

3.2	PROGRAM STRUCTURE

BEST-CMS is designed to be a fully general ceramic composite analysis system employing the boundary element method. The program is written using standard FORTRAN 77. Development has been carried out at SUNY/Buffalo on an HP 9000 minicomputer, a SUN-4 workstation, and a SUN Sparcstation-1. The nature of the method is such that, for any realistic problem, not all required data can reside simultaneously in core. For this reason extensive use is made of both sequential and direct access scratch files.

The program first executes an input segment. After the input has been processed, the surface integrals are calculated and assembled into the set of system equations using specified boundary conditions, followed by the insert assembly and the inclusion of the insert equations in the general system. The system matrix is then decomposed and saved on disk, followed by the calculation of the solution vector. The full displacement and traction (and/or temperature and flux) solution on each boundary element and insert element is then reconstructed from the solution vector. In a time dependent problem the process of constructing the load vector for the system equations is repeated at each time step, but the integration, formation and decomposition of the system matrix are done only once. However, in an analysis with nonlinear matrix-insert interface connections, the assembly and decomposition must be carried out several times.

Various aspects of the computer program are discussed below.

3.3 PROGRAM INPUT

The input for **BEST-CMS** is free field. Meaningful keywords are used to identify data types and to name particular data sets. The input is divided into five types:

1. Case Control Cards

The case control cards define global characteristics of the problem. In addition to the problem title, the times for multiple time steps are defined. The reading or writing of restart data is also defined at this point.

2. Material Property Definition

The material property input allows the definition of material properties for a variety of materials. The Young's modulus can be prescribed in tabular form for a user-defined set of temperatures. Temperature independent values of Poisson's ratio are also defined.

3. Geometry Input

Geometry input is defined one GMR (generic modeling region, or subregion) at a time. To initiate the input, a tag is provided to identify the GMR, a material name and reference temperature are defined to allow initialization of material properties.

The next block of geometry input consists of the Cartesian coordinates of the user input points for the outer surface geometry definition of the composite matrix, together with identifiers (normally positive integers) for these geometric nodes.

Following the definition of an initial set of nodal points, the surface connectivity of the outer surface of the composite matrix is defined through the input of one or more named surfaces. Each surface is made up of a number of elements, with each element defined in terms of several geometric nodes. Three sided elements, defined using six rather than eight geometric nodes, are used for mesh transition purposes. The terms quadrilateral and triangle are normally used to refer to the eight and six noded elements, although the real geometry represented is, in general, a nonplanar surface patch. Seven and nine noded elements are made available by adding a central node to the six or eight noded elements.

Over each element the variation of displacement and traction (and/or temperature and flux) can be defined using either the linear, quadratic or quartic shape functions. Linear and quadratic (or quadratic and quartic) elements can share a common side, which is then constrained to have linear (or quadratic) displacement and traction (and/or temperature and flux) variation.

Finally an option is available to allow quadratic functional variation (8 or 6 nodes) to be used in conjunction with linear geometry (4 or 3 nodes). In this case the program generates the additional nodes automatically at mid-point of the sides. The characteristics of the various element types are summarized below.

Surface Element Type	Geometry Nodes	Displacement/Traction Nodes (and/or Temperature/Flux Nodes)
Linear Quadrilateral	4, 8 or 9	4
Linear Triangle	3, 6 or 7	3
Quadratic Quadrilateral	4, 8 or 9	8 or 9
Quadratic Triangle	3 or 6, or 7	6
Quartic Quadrilateral	4, 8 or 9	12 or 15
Quartic Triangle	3, 6 or 7	17 or 25

Following the definition for the composite matrix outer surface, the embedded inserts are then defined. These are defined as curvilinear line elements with a prescribed radius of the cross-section. The inserts are generally straight, however as noted, curved inserts are also allowed. The user first defines the nodal coordinates of the centerline of the insert. Thereafter, the radius and the insert connectivity is defined. Linear and quadratic elements are available for both geometry and functional variation, however, quadratic functional variation over linear geometry is not presently available. The various options for the insert elements are summarized below.

Insert Element Type	Geometry Nodes	Displacement/Traction Nodes (and/or Temperature/Flux Nodes)
Linear-Linear	2	2
Quadratic-Linear	3	2
Quadratic-Quadratic	3	3

Note only the surface of the insert needs to be defined, i.e., the hole in the composite matrix which encompasses the insert does not have to be explicitly defined.

4. Interface Conditions

The interface input describes the connection of surfaces or elements of one region to another and between the matrix and the inserts. Special types of matrix-insert interface conditions which are available presently include fully-bonded and sliding contact (including coulomb friction) and springs.

5. Boundary Condition Input

The final input section provides for the definition of boundary conditions, as functions of both position and time. Data can be input for an entire surface, or for a subset (elements or nodes) of a surface. Input can be in global coordinates, or can define rollers or pressure (or flux) in the local coordinate system. Input simplifications are available for the frequently occurring cases of boundary data which is constant with respect to space and/or time variation. Each boundary condition set can be defined at a different set of times.

Following the processing of the input data, the surface integrals occurring in equations (2.2.6), (2.2.7), (2.5.6), (2.5.18) and (2.5.21) are evaluated numerically. This is the most time consuming portion of the analyses. In **BEST-CMS** the results of these integrations are stored as they are calculated, rather than being assembled into the final equation system immediately. Although this is somewhat more costly in terms of storage and CPU (central processing unit) time, it has led to much greater clarity in the writing of **BEST-CMS**. In addition, it provides much greater flexibility in the implementation of various restart and boundary condition options.

The calculations proceed first by GMR (generic modeling region), then by source point (the equation being constructed) and finally by surface element and insert element. The results for each source point element pair are written to disk. All of the calculations are carried out and stored in the global (Cartesian) coordinate system.

The integration of the BEM equations is the most complex part of the code. In this process either singular or nonsingular integrals can be encountered. The integrals are singular if the source point for the equations being constructed lies on the element being integrated. Otherwise, the integrals are nonsingular, although numerical evaluation is still difficult if the source point and the element being integrated are close together.

In both the singular and nonsingular cases Gaussian integration is used. The basic technique is developed in Banerjee and Butterfield, 1981. In the nonsingular case an approximate error estimate for the integral was developed based on the work of Stroud and Secrest (1966). This allows the determination of element subdivisions and orders of Gaussian integration which will retain a consistent level of error throughout the structure. Numerical tests have shown that the use of 3, 4 and 5 point Gauss rules provide the best combination of accuracy and efficiency. In the present code the 4 point rule is used for nonsingular integration, and error is controlled through element subdivision. The origin of the element subdivision is taken to be the closest point to the source point on the element being integrated.

If the source point is very close to the element being integrated, the use of a uniform subdivision of the element can lead to excessive computing time. This frequently happens in the case of aerospace structures, due either to mesh transitions or to the analysis of thin walled structures. In order to improve efficiency, while retaining accuracy, a graded element subdivision was employed. Based on one-dimensional tests, it was found that the subelement divisions could be allowed to grow geometrically away from the origin of the element subdivision. Numerical tests on a complex three-dimensional problem have shown that a mesh expansion factor as high as 4.0 can be employed without significant degradation of accuracy.

In each case of singular integration (source point on the elements being integrated) the element is first divided into subelements. The integration over each subelement is carried out using a Jacobian transformation in mapping. This coordinate transformation produces nonsingular behavior in all except one of the required integrals. Normal Gauss rules can then be employed. The remaining integral (that of the traction kernel F_{ij} times the isoparametric shape function which is 1.0 at the source point) is still singular, and cannot be numerically evaluated with reasonable efficiency and accuracy. Its calculation is carried out indirectly, using the fact that the stresses due to a rigid body translation are zero (Lachat and Watson, 1976). It has been found that subdivision in the circumferential direction of a two-dimensional surface element is required to preserve accuracy in the singular integration of the outer surface. A maximum included angle of 15 degrees is used. Subdivision in the radial direction has not been required.

The integrals required for calculation of displacement, stress, temperature, and flux at interior points are of the same type as those involved in the generation of the system equations, except that only nonsingular integrals are involved. If the source point involved is located on the surface of the body, then numerical integration is not required. Instead, the required quantities are calculated using the displacements and tractions (or temperature and flux) on the element (or elements) containing the source point, as discussed in Section 2.2.

3.5 | SYSTEM MATRIX ASSEMBLY

The first step in the assembly process is the reduction of the rectangular matrix of **F** integrals to a square matrix. This matrix is the prototype of the system matrix. The columns of the matrix are transformed or replaced, as required by the boundary conditions, as the assembly process proceeds.

The next step in the process is the incorporation of the insert equations in the system. As was described in detail for each specific analysis in Section 2, the insert assembly consists of an insert by insert matrix multiplication and backsubstitution. The backsubstitution minimizes the number of equations required in the system by eliminating some of the unknown quantities on the inserts.

A key problem in the entire process is the proper definition of appropriate coordinate systems, on a nodal basis. This is a problem common in any direct boundary element method which treats structures with nonsmooth surfaces. It arises because the tractions at a point are not uniquely determined unless the normal direction to the surface varies continuously at the point in question.

The original surface integral calculations are all done in global coordinates. If the displacement (or temperature) boundary condition is specified at a given node, in global coordinates, then no new coordinate system definition is required. It is only necessary to keep track of the subset of elements, containing the given node, on which the fixed displacement (or temperature) is to be reacted. However, if a displacement (or temperature) is specified in a nonglobal direction at a given node, then a new nodal coordinate system must be defined and, potentially, updated as further boundary conditions are processed. The associated nonzero reactions must then be expressed in the new coordinate system.

Following this preparatory work, the final assembly of the system equations is carried out. It is performed in three major steps:

1. Transformation of the columns of the matrices to appropriate local coordinate systems and incorporation of any boundary conditions involving springs.

2. Incorporation of compatibility and equilibrium conditions on interfaces.

3. Application of specified displacements and tractions (and/or temperatures and fluxes).

Two particular features of the equation assembly deserve special comment. First, in multi-GMR problems the system matrix is not full. Rather it can be thought of as consisting of an NxN array of submatrices, each of which is either fully populated or completely zero. Only the nonzero portions of the system equations are preserved during system matrix assembly. In order to improve the numerical conditioning of the system matrix for the solution process, the columns are reordered so that the variables from the two regions, lying on the same interface, are as close together as possible.

Second, rather than simply assembling an explicit load vector at each time point in the solution process, load vector coefficient matrices are assembled and stored. These allow the updating of the load vector at any required time point simply by interpolating the time dependent boundary conditions and performing a matrix multiplication. A Similar process is used in the calculation of interior and boundary stresses.

3.6 | SYSTEM EQUATION SOLUTION

The solver employed in **BEST-CMS** operates at the submatrix level, using software from the LINPACK package (Dongarra, 1979) to carry out all operations on submatrices. The system matrix is stored, by submatrices, on a direct access file. The decomposition process is a Gaussian reduction to upper triangular (submatrix) form. The row operations required during the decomposition are stored in the space originally occupied by the lower triangle of the system matrix. Pivoting of rows within diagonal submatrices is permitted.

The calculation of the solution vector is carried out by a separate subroutine, using the decomposed form of the system matrix from the direct access file. The process of repeated solution, required for problems with multi-time steps, is highly efficient.

3.7 │ NONLINEAR PROCESS

The nonlinear interface solution algorithm starts with an elastic analysis (with linear elastic spring interface connection between the matrix and the insert) of the problem for the first loading increment (complete with the specified boundary and body force loading). At the end of the elastic increment the state variables are calculated and the nonlinear constitutive interface relations are established. The difference between the actual tractions and the elastic (or previous) tractions is estimated. A new equation system is assembled with the calculated nonlinear constitutive interface relations. The process is essentially repeated until the constitutive equations yield tractions that are negligibly different from the previous step. A new loading increment is taken and the process is repeated for each subsequent increment.

The nonlinear material algorithm uses the displacement rate equations written at the nodes of the boundary elements and the stress rate equations written at the nodes of the cylindrical volume cells together with the constitutive models to solve the system in an incremental fashion. The σ^o vector is initially assumed zero and then is determined in an iterative process during each load increment. The implementation includes a time saving feature that reduces the number of iterations needed for convergence, by utilizing the past history of initial stresses to estimate the value of initial stress rates for the following load increment. The iterative procedure has been implemented for the Von Mises model (with strain hardening), a creep model, and a plastic-fracture model.

3.8 | OUTPUT DESCRIPTION

The output from **BEST-CMS** is relatively straightforward. It consists of ten sections, as follows:

1. Complete echo of the input data set.

2. Summary of case control and material property input.

3. Complete definition for each GMR, including all surface insert nodes, surface and insert elements.

4. Complete summary for each interface and boundary condition set, including the elements and nodes affected.

5. Boundary solution (on an element basis), including displacements and tractions (and/or temperature and fluxes) at each node of each element.

6. The resultant load on each element and on the entire GMR is calculated and printed.

7. Solution for the displacements and tractions (and/or temperatures and fluxes) at the Insert-Matrix composite interface (on an element basis).

8. Displacement, stress and strain (and/or temperature and fluxes) on a nodal basis, at all surface nodes, for each GMR, and at nodes of the cylindrical volume cell in a nonlinear material analysis.

4.0 | GUIDE TO USING BEST-CMS

Since **BEST-CMS** has been designed to cover a very wide range of problems in engineering it may appear to be a little difficult for a beginner to get started. This section is therefore written to provide some guidance to such an user. It is hoped to expand this section of the manual fully with wider user participation.

4.1 | GETTING STARTED

Usually the user is motivated by a specific problem in a given technical area (heat transfer or elasticity for example). It is suggested that he should first read the analysis section of the manual to get some flavor of the BEM in that area. He should then briefly examine the structure and organization of the input data in Section 5 in conjection with a sample problem data set given in Section 6. Section 7 may contain a brief description of a specific engineering example in the technical area of interest to the user.

It may also be helpful to use a specific test data given in Section 6 and modify it to create a new test problem. In order to do this the user must of course study the relevant parts of Section 5.

4.2	AVAILABILTY OF BEST-CMS

BEST-CMS is written in FORTRAN 77 and is therefore adaptable to any computer which has such a compiler.

| 4.3 | FILESYSTEM IN BEST-CMS |

BEST-CMS makes use of unit 5 as its input data file and unit 6 as its output file. In addition to these an extensive set of disk files are used during the execution of the code. For the complete range of analysis used in **BEST-CMS** it is necessary to have the 60 simultaneous open files in the system. Not all of these files are necessary for the simpler analyses such as quasistatic elastic or steady state heat transfer analyses, where usually only 1/3 of the total are used. For more complex time dependant analyses extensive use of disc files is made. The files are either of sequential or direct access type and are defined as FT** based on IBM terminology.

For the efficient execution of **BEST-CMS**, it is desirable to have at least 8 megabytes of system memory. Additionally, **BEST-CMS** makes extensive use of disk files during execution of the code. While most of these files are of temporary nature, some are required for restart analyses. In any case, it is recommended that workstation-based users have at least 300 MB of disk space free in order to run practical three-dimensional problems.

4.4 | SPECIAL FEATURES OF BEST-CMS

4.4.1 | DEFINITIONS

The following definitions are used throughout the manual.

Points, Nodes or Nodal Points - are generic names for all points in a data set for which coordinates are defined. These points may be source points and/or geometric points which are used in the boundary, hole or insert discretizations. Points are usually user defined, however, **BEST-CMS** may create additional points through automated surface generation (see the argument REFNAME on the SURF card in **GMR input in chapter 5). Additional Nodes may also be generated by **BEST-CMS** for hole and/or inserts. All points defined in a data set by the user should have unique node numbering.

Geometric Points - are points used in the geometrical definition of the body of interest. Specifically, geometric points are used in the description of the geometry of a boundary element, hole element, insert element, or volume cell. Geometric points also may or may not be source points.

Functional Nodes - same as source points.

(Boundary) Source Points - are points in a discretization of the boundary surface (or interface) which are used in the functional representation of the field variables across the boundary elements. At every boundary source point (and only at boundary source points) unknowns in the boundary system equation are retained corresponding to the unknown boundary conditions at these points. Likewise, known boundary conditions (implicitly or explicitly defined) are required at these points. Boundary conditions specifications for points other than boundary source points will result in a fatal error. Boundary source points are selected by **BEST-CMS** based on the type of functional variation of the primary variables across the boundary element which is defined in the data set by the user. (see SURF and TYPE cards under **GMR input in Chapter 5).

Insert Elements - are used to model a circular composite fiber in a 3-D body where the elastic modulus of the fiber is different from that of the matrix.

Generic Modeling Region (GMR) - in a boundary element analysis the body under investigation may be fictitously divided in a number of smaller parts for convienence in mesh modelling and efficiency in computation. Each part is called a generic modelling region and is modelled as an individual boundary element model. The nodes and elements of each region must match up at common interfaces and are connected by relations defined by the user. The term generic refers to the fact that only geometry is defined in this section and the part defined can then be used for any type of analysis (e.g. elasticity, heat transfer etc.).

4.4.2	MESH SIZE

Most of the currently available experience of developing mesh for a given problem is based on nearly two decades of the finite element or finite difference analyses. It is possible to take only the boundary part of a given finite element mesh system to generate the boundary element mesh system. Unfortunately this often leads to an inefficient BEM analysis because of use of too many unnecessary elements. For three-dimensional problems it is often necessary to reduce this residual surface elements so generated by almost a factor of two to four. If the geometry of the surface is too complex so that this reduction could not be achieved without sacrificing geometrical details that are important in the analysis then one must use the mesh without modification.

| 4.4.3 | SPECIFICATION OF BOUNDARY RESTRAINTS |

BEM analyses require the same type of restraint on the boundaries so that unwanted movements of the solution region is prevented. It must be noted however, restraints often introduces local singularities, particularly at the edges of the restraints. Since finite element method does not capture any of these local singularities, engineers do not have to pay any special attention to them.

Since the specification of the boundary element condition needs to be very precise in a boundary based solution scheme it is often possible to capture high stress and strain gradients purely by specifying unnecessary restraints. For example a cantilever completely fixed at the supported end by specifying both displacement components zero will not provide the correct distribution of bending and shear stresses of a beam solution at the supported end. A suitable roller support, on the other hand, will provide the beam bending solution.

It is therefore strongly recommended that a user specifies roller support to simulate the necessary restraints.

In the case of a heat conduction problem the temperature is uniquely defined but its normal derivatives are multivalued at a corner node. Similarly, for an elasticity problem the displacements are uniquely defined but the surface tractions are multivalued at a corner node. Thus if we wish to write the equation (for an elasticity problem)

$$\beta u^P = \sum_{q=1}^{N} \left[\left(\int_{\Delta S} G^{pq} N^q ds \right) t_n - \left(\int_{\Delta S} F^{pq} N^q ds \right) u_n \right] \qquad 4.1$$

for m boundary nodes including one true corner node, the resulting final system of equations will be

$$Gt - Fu = 0 \qquad 4.2$$

where

$F =$ $2m \times 2m$ or $3m \times 3m$ matrix for two- and three-dimensional problems respectively

and

$G =$ $2m \times (2m + 2)$ or $3m \times (3m + 4)$ matrix for two- and three-dimensional problems respectively

The additional columns in G arise from the multivalued tractions defined at the corner node. If these tractions are all specifically prescribed then the solution of Eq. (4.2) presents no difficulty. A suitable mixture of tractions and displacement boundary conditions (Fig. 4.1a-b) at the corner also presents no difficulty if the final system matrix involving all the unknowns is square and of order $2m \times 2m$ for two-dimensional problems and $3m \times 3m$ for three-dimensional ones. If the displacements alone are specified at the corner (Fig. 4.1c) it is not possible to solve Eq. (4.2) exactly and an approximation will be utilized at this node.
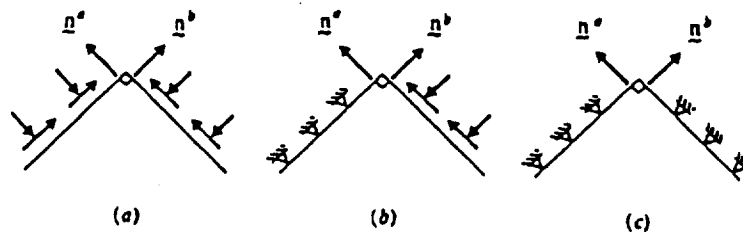
Fig. 4.1 The problems at a corner of the boundary

It should be noted that the preceding remarks only apply to a sharp corner that actually exists in a specific problem. The discretization of a smooth surface using flat boundary elements also results in boundary discontinuities, but these must be treated as if the boundary were continous if correct results are to be obtained.

5.0	BEST-CMS INPUT

The basic input required by **BEST-CMS** is the definition of **Geometry, Material Properties and Boundary conditions.** While this is the same definition required by a finite element structural analysis program, a somewhat different set of information is required to accomplish the definition for a boundary element program.

The input to **BEST-CMS** is intended to be as simple as possible. Meaningful keywords are used for the identification of data types. Free field input of both keywords and numerical data is permitted, however there are a number of general rules that must be followed.

1. Upper Case

All alphanumeric input must be provided in upper case.

Proper Usage:

```
**CASE
   TITLE  TRANSIENT HEAT CONDUCTION - TEST CASE
   TRANSIENT
   SYMMETRY QUARTER
```

Improper Usage:

```
   transient
   SYMMETRY quarter
```

2. Parameter Positioning

Parameters may appear anywhere on an input line, as long as they appear in the proper order and are separated by at least one blank space.

Proper Usage:

```
   CTHERMAL PLASTICITY STEADY
   ELEMENT 1 6 8
```

Improper Usage:

```
   CTHERMALPLASTICITY STEADY
   ELEMENT, 1, 6, 8
```

3. Length of Input Line

An input line cannot exceed a maximum of 80 characters including blank spaces.

4. Keyword Truncation

Any keywords that are longer than four letters may be truncated to the first four letters.

Proper Usage:

```
SYMMETRY QUAR
SYMM QUAR
ELEM 1 6 8
```

Improper Usage:

```
SYMMETRY QUA
```

5. Floating Point Numbers

Any real parameters may be input in either FORTRAN E or F format, however, the representation used must contain no more than 16 total characters. Additionally, there is a limit of 8 characters to the left of the decimal point.

Proper Usage:

```
EMOD 30.E+7
ALPHA 1.E-06
POINTS
   0.004 1.110 0.0
```

Improper Usage:

```
EMOD 300000000.0
ALPHA 1.-6
POINTS
   4.0-3 1.110 0
```

General Rules for Input Data

---

6. Comments

Comments can be inserted in the data file by placing a dollar sign ($) anywhere on an input line. The remainder of that input line is then ignored by the **BEST-CMS** input processor.

Proper Usage:

```
ELEMENT 1 6 8 $ ELEMENTS ON THE OUTER RIM
$
$ MODIFIED 03/08/88 GFD
   POINTS 25 26 27
```

---

7. Blank Lines

Blank Lines can be inserted anywhere in the data file and are useful for aesthetic purposes.

---

8. Units

A consistent system of units must be used for input of all types (material properties, geometry, boundary conditions, time steps). Output will be in the same consistent system of units. The selection of appropriate units is the user's responsibility.

---

9. ** Keywords

Certain keywords are prefixed by the symbol **. These identify the beginning of a block of data of a particular type, and serve to direct the program to the appropriate data processing routine. There should be no blank spaces between the ** symbol and the pertinent keyword. Additionally, the ** data blocks must appear in the following specific order:

```
**CASE
**MATERIAL
**GMR
**INTERFACE
**BCSET
```

There may be multiple data blocks of each type, except for the **CASE block.

General Recommendations for Input Data

1. Ordering of Input Items

While there is some flexibility in the ordering of lines within a **BEST-CMS** data set, it is strongly recommended that the user follow the order provided in the manual. Examples of proper ordering are provided throughout this chapter.

2. Documenting Data Sets

The $ keyword is provided to permit comments anywhere in the input data set. This should be used generously to fully document the analysis. Blank spaces can also be used to improve readability. The format, displayed in the examples of this chapter and in Section 6.0, is recommended.

General Limits of **BEST-CMS**

It should be noted that there are certain limits which must be observed in the preparation of input for **BEST-CMS** . These limits are of two main types:

1 - Limits on the maximum number of entities of various types within a single analysis.

2 - Limits on the user specified numbering of certain entities.

The program limits of BEST-CMS are set in a file called 'PARAM', and can be customized to suit the user's size requirements. To determine the limits of a compile code, insert in a data file "PRINT LIMITS" under "**CASE INPUT," and execute the program with this data file. The limits will be output in the BEST-CMS result file (see Section 5.1.8).

The following table (without values) is provided for the convenience of the user. After customizing the 'PARAM' file the user can complete this table for future reference.

ENTITY*   LIMIT

GLOBAL PARAMETERS
total geometric modeling regions
total points (including non-source points)
total boundary source points
total boundary elements
total fiber insert elements
cyclic symmetry interfaces

REGION (GMR) PARAMETERS
surfaces in any region
points in any region
boundary source points in any region
boundary elements in any region**
fiber insert elements in any region

ENTITY*

OTHER PARAMETERS
table points
temperature points for material properties

USER SPECIFIED NUMBERING
points   99999
elements   99999

---

* Definition of the terminology used in this table can be found in Section 4.4.1.

** Total boundary elements in a region include user specified boundary elements plus elements artificially created in symmetric regions when the symmetry option is invoked.

Individual Data Items

The remainder of this chapter provides detailed information on each of the data items available within **BEST-CMS**. The individual items are grouped in sections, under the associated ** keyword, as follows:

5.1 CASE CONTROL INFORMATION (**CASE)

5.2 MATERIAL PROPERTY DEFINITION (**MATE)

5.3 GEOMETRY DEFINITION (**GMR)

5.4 INTERFACE DEFINITION BETWEEN SUBREGIONS (**INTE)

5.5 BOUNDARY CONDITION DEFINITION (**BCSE)

5.1 | CASE CONTROL

This input section provides **BEST-CMS** with information controlling the overall execution. It provides the title and determines which of the major program branches will be executed. It also defines the times at which solutions of the given problem are to be evaluated. This section must be input exactly once for each analysis and must be input before any other data.

A list of keywords recognized in the case control input are given below, and a detailed description follows. It is recommended that the user supply the relevant keywords in the order provided by this list.

SECTION	KEYWORD	PURPOSE
5.1.1	Case Control Input Card	
	**CASE	Start of case control input
5.1.2	Title	
	TITL	Title of job
5.1.3	Times for Output	
	TIME	Times of solution output (static and steady-state analysis)
	TIME STEP	Time step for transient solution algorithms

SECTION	KEYWORD	PURPOSE
5.1.4	Type of Analysis	
	CTHE	Concurrent thermal analysis
	ELAS	Elastic analysis
	HEAT	Heat Transfer analysis
	PLAS	Elastoplastic analysis
	CREEP	Creep analysis
5.1.5	Geometric and Loading Symmetry Control	
	SYMM HALF	Symmetry about Y-Z plane
	SYMM QUAR	Symmetry about X-Z and Y-Z planes
	SYMM OCTA	Symmetry about X-Z, Y-Z, and X-Y planes
5.1.6	Restart Facility	
	REST WRIT	Save integration files for future runs
	REST READ	Use integration files from previous run
5.1.7	Integration Precision and Efficiency Optimization	
	PREC	Numerical integration precision
5.1.8	Output Options	
	ECHO	Produce echo of input data
	PRIN BOUN	Printout displacement and traction results
	PRIN NODA	Print boundary displacement, stress, strain at nodal points
	PRIN LOAD	Print load calculation
	PRIN ALL	Print maximum printed output file
	PRIN NONE	Print only a minimal printed output file
	PRIN LIMI	Print current BEST-CMS limits
	PRIN FEAT	Print current implentation status of BEST-CMS special features
	CALC BOUN	Calculate displacement and traction results
	CALC NODA	Calculate boundary displacement, stress, strain at nodal points
	CALC LOAD	Calculate element loads
	NEUT	Produce formatted ASCII BEST-CMS neutral file

5.1.9   Miscellaneous Control Options

CHEC	Check input data only
FILE	Specify directory for creation and storage of scratch files
CVCP	Cylindrical volume cell parameter
EXAC	Exact insert approach
NUME	Numerical integration of inserts

5.1.1	CASE CONTROL INPUT CARD

****CASE**

Status - REQUIRED

Full Keyword - ****CASE** control

Function - Identifies the beginning of the case control input section.

Input Variables - NONE

Additional Information - NONE

Examples of Use -

1. Request a three-dimensional steady-state heat transfer analysis.

```
**CASE
  TITLE  HEAT CONDUCTION IN A MOLD
  HEAT
```

5.1.2	TITLE

TITL   CASETITLE

Status - REQUIRED

Full Keyword - TITLE

Function - Defines title for analysis.

Input Variables -

     CASETITLE   (Alphanumeric)   - REQUIRED - 72 chars. max. length

Additional Information - NONE

Examples of Use -

    1. Describe the analysis.

```
**CASE
  TITLE  TURBINE BLADE A7311 - THERMOELASTIC ANALYSIS
  ELASTIC
```

5.1.3	TIMES FOR OUTPUT

TIME   T1 T2 T3 ...   TN

Status - OPTIONAL

Full Keyword - TIMES

Function - Identifies times at which output is required (only for static analysis).

Input Variables -

       T1   (Real) - REQUIRED

       T2 ... TN   (Real) - OPTIONAL

Additional Information -

This input may be continued on more than one card, if required. Each card must begin with the keyword TIME. A maximum of twenty output times may be selected. A minimum of one output time must be chosen.

This card is only functional for static analysis. The 'TIME STEP' card (see next page) is used for transient analysis.

Examples of Use -

1. Conduct an elastic analysis at times 1.0, 2.5 and 6.0 and output the results.

```
**CASE
  TITLE  ROTOR - ELASTIC ANALYSIS
  TIMES 1.0 2.5 6.0
  ELASTIC
```

TIME STEP NSTEP DELTA

Status - OPTIONAL (required for transient analysis algorithms)

Full Keyword - TIME STEP

Function - Identifies the number of time steps in a transient solution algorithm and the size of the time steps.

Input Variables -

NSTEP (Integer) - REQUIRED

Sets the number of time steps for which the transient analysis is to be carried out.

DELTA (Real) - REQUIRED

Defines the size of the time step.

Additional Information -

For heat conduction or uncoupled thermoelasticity the time step size should be $0.0025 \leq DELTA *c/L^2 \leq 0.05$ where $c = k/\rho c_\epsilon$ is diffusivity, k is the conductivity, ρ is the density, c_ϵ is the specific heat and L is the width of the smallest element.

Examples of Use -

1. Conduct a transient heat transfer analysis of a spherical tank.

```
**CASE
  TITLE   SPHERICAL TANK - SUDDEN PRESSURIZATION
  HEAT
  TIME STEP   10  0.01
```

5.1.4	TYPE OF ANALYSIS

CTHE   ITYPE

Status - OPTIONAL

Full Keyword - **CTHERMAL**

Function - Identifies a concurrent thermoelastic analysis.

Input Variables -

ITYPE   (Alphanumeric)   - OPTIONAL

Allowable values are STEA and TRAN.

Default is linear, steady-state analysis.

STEAdy - Identifies a steady-state analysis.

TRANsient - Identifies a transient analysis.

Additional Information -

Displacements, tractions, temperatures and heat flux are determined at each time step.

Examples of Use -

1. Conduct a transient concurrent thermoelastic analysis of an axisymmetric pressure vessel. Use a time increment of 0.05 and include 14 steps.

```
**CASE
  TITLE  PRESSURE VESSEL AG-9 THERMAL LOAD
  AXISYMMETRIC
  CTHERMAL  TRANS
  TIME STEP  14  0.05
```

2. Determine the steady-state thermoelastic response of a turbine blade.

```
**CASE
  TITLE  TURBINE BLADE A7311 - THERMOELASTIC
  CTHERMAL STEADY
```

ELAS

Status - OPTIONAL

Full Keyword - ELASTIC

Function - Identifies an elastic analysis.

Input Variables - NONE

Additional Information -

Elastic analysis is the default analysis type. Therefore, an elastic analysis will be performed.

Examples of Use -

1. Request an axisymmetric elastic stress analysis of a concrete reactor pressure vessel.

```
**CASE
  TITLE  REACTOR PRESSURE VESSEL - LOAD CASE 2B
  AXISYMMETRY
  ELASTIC
```

HEAT ITYPE

Status - OPTIONAL

Full Keyword - **HEAT**

Function - Identifies a heat conduction analysis.

Input Variables -

ITYPE Alphanumeric - OPTIONAL

Allowable value is STEA or TRAN. (Default is steady-state analysis)

STEAdy - Identifies a steady-state analysis.

TRANsient - Identifies transient analysis.

Additional Information -

None

Examples of Use -

1. Find the steady-state temperature distribution in a heat exchanger.

```
**CASE
  TITLE   TUBE-AND-FIN HEX - STEADY-STATE
  HEAT
```

PLAS

Status - OPTIONAL

Full Keyword - PLASTICITY

Function - Identifies an analysis involving quasi-static elastoplastic material response.

Input Variables - NONE

Additional Information -

   If this card is input, the CVCP card should be present in **Case.

Examples of Use -

   1. Perform elastoplastic analysis.

```
**CASE
  TITLE PRESSURE VESSEL - PLASTICITY
  TIMES   1.0   2.0   3.0   4.0
  PLASTICITY
  CVCP  1.63
```

CREEP NSTEP SIZE NPRINT

Status - OPTIONAL

Full Keyword - CREEP

Function - Identifies an analysis involving creep material response.

Input Variables -
> NSTEP (Integer) - REQUIRED
> Number of time steps
> SIZE (real) - REQUIRED
> The size of each time step
> NPRINT (Integer) - OPTIONAL
> Results will be output every "NPRINT" steps. Default is 1

Additional Information -

> In order to guarantee stability of the explicit time integration scheme it is important to choose a time step ("SIZE") in accordance with the following criteria

$$\Delta t \leq \Delta t_{crit} = \frac{2}{\gamma E n \sigma^{m-1}}$$

> where E is the modulus of elasticity of the composite matrix, σ is the current equivalent stress, and γ and n are material constants defined on the CREEP card in **MATERIAL.

> The TIME or TIME STEP card should NOT appear in **CASE for CREEP analysis.

> If this card is input, the CVCP card should be present in **CASE

Examples of Use -

> 1. Perform elastoplastic analysis.

```
**CASE
  TITLE PRESSURE VESSEL - PLASTICITY
  TIMES   1.0    2.0    3.0    4.0
  PLASTICITY
  CVCP  1.63
```

5.1.5	GEOMETRIC AND LOADING SYMMETRY CONTROL

SYMM   STYPE

Status - OPTIONAL

Full Keyword - **SYMMETRY**

Function - Identifies a problem with geometric and loading symmetry.

Input Variables -

> STYPE   (Alphanumeric)   - REQUIRED
>
> Allowable values are HALF, QUAR, and OCTA.
>
> > HALF - Half symmetry, about the Y-Z plane.
> >
> > QUAR - Quarter symmetry, about the X-Z and Y-Z planes.
> >
> > OCTA - Octal symmetry, about the X-Z, Y-Z, and X-Y planes.

Additional Information -

> To model the problem geometry, in all cases, use the part of the geometry which is on the positive side of the axis (axes) of symmetry.
>
> If the SYMM card is used the plane of symmetry does not have to be modelled, and therefore, boundary elements should not appear on the plane of symmetry (see the figure on the following page).
>
> The use of the SYMM card automatically invokes the condition of zero displacement (and zero flux) on and perpendicular to the plane of symmetry. Therefore displacement (and/or flux) in the perpendicular direction does not have to be set to zero at the plane or at any other point for the purpose of preventing (arbitrary) rigid-body motion (in this direction) as is usually required.

Examples of Use -

> 1. Perform an elastic analysis on a hollow sphere utilizing a model of only the first (positve) octant.

```
**CASE
  TITLE  HOLLOW SPHERE WITH INTERNAL PRESSURE
  ELASTIC
  SYMMETRY OCTAL
```
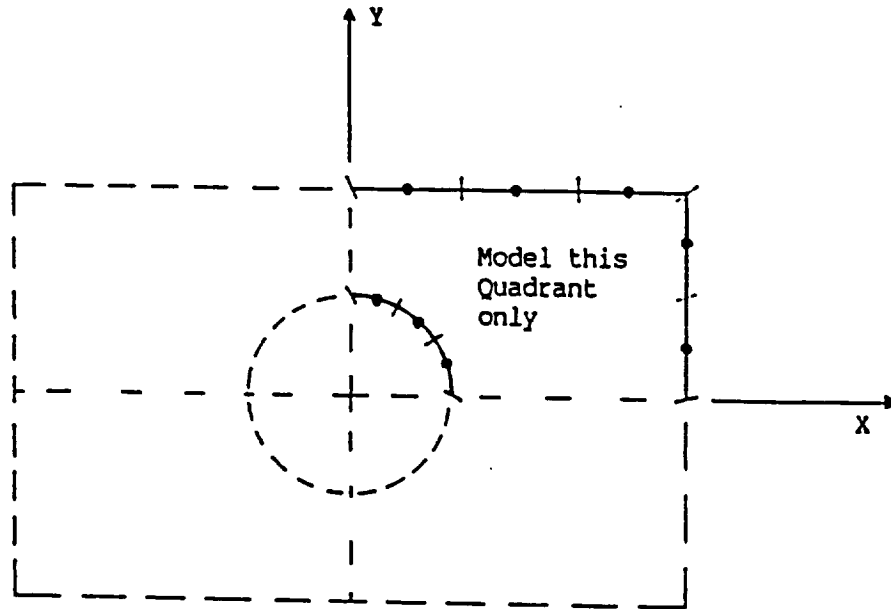
Figure for **CASE: SYMM card
Two-dimensional Quarter symmetry model

5.1.6	RESTART FACILITY

REST   RTYPE

Status - OPTIONAL

Full Keyword - **RESTART**

Function - Enables the restart facility.

Input Variables -

     RTYPE   (Alphanumeric) - REQUIRED
     Allowable values are WRIT, READ

          WRITe - Saves all of the integration files generated during the current run for later reuse.

          READ - Bypasses the integration phase for the current run. Instead, the integration files from a previous run are utilized.

Additional Information -

Integration is generally the most expensive part of any boundary element analysis. Consequently, when the same model is to be run with several sets of boundary conditions, the restart facility should be used.

A complete analysis must first be run with RESTart WRITe specified. The files FT31, FT32, FT33, FT34, FT35, FT36, FT37, FT38 and FT39 are then retained after completion of the run. These files contain all the integration coefficients that were computed. Subsequent runs can then be made, with different sets of boundary conditions, by using RESTart READ. In this case, the integration phase will be skipped. Instead; the integration coefficients will be read from the files FT31, FT32, FT33, FT34, FT35, FT36, FT37, FT38 and FT39.

Geometry and material properties must be the same for both the RESTart WRITe and RESTart READ data sets. However, no checking is done by **BEST-CMS**. This is the user's responsibility.

The restart facility is not available for transient analyses.

Examples of Use -

1. Save the integration files generated during an elastic analysis of an axle.

```
**CASE
   TITLE  AXLE - LOAD CASE 1A
   TIMES  1.0
   ELASTIC
```

RESTART WRITE

2. Rerun an elastic analysis of the same axle with a different set of boundary conditions by using existing integration files.

```
**CASE
  TITLE  AXLE - LOAD CASE 1B
  TIMES  1.0
  ELASTIC
  RESTART READ
```

5.1.7	INTEGRATION PRECISION

PREC   PTYPE

Status - OPTIONAL

Full Keyword - PRECISION

Function - Provides numerical efficiency through the use of lower order surface integration.

Input Variables -

> PTYPE   (Alphanumeric)   - OPTIONAL

> Allowable values are HIGH, MEDIUM, and LOW.

>> HIGH - Use 4 X 4 integration on each element subsegment.

>> MEDI - Use 3 X 3 integration on each element subsegment.

>> LOW - Use 2 X 2 integration on each element subsegment.

Additional Information -

> In many cases, MEDIum or LOW order integration produces results that are acceptable for engineering analysis, and significant computational savings can be realized. However, for other problems, particularly those involving thin-walled bodies, PRECision HIGH is recommmended.

> High precision (4 X 4 integration) is used if a PREC card is not present.

> This card does not affect the integration of hole elements.

Examples of Use -

> 1. Perform a steady-state heat conduction analysis of a casting mold.

```
**CASE
  TITLE  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTART WRITE
  PRECISION LOW
```

5.1.8	OUTPUT OPTIONS

ECHO

Status - OPTIONAL

Full Keyword - ECHO

Function - Requests a complete echo print of all card images in the input data set.

Input Variables - NONE

Additional Information - Default is no echo print.

Examples of Use -

1. Request an elastic analysis with an echo of the input data set.

```
**CASE
  TITLE  TURBINE BLADE
  ELASTIC
  RESTART WRITE
  ECHO
```

PRIN PTYPE

Status - OPTIONAL

Full Keyword - PRINTOUT-CONTROL

Function - Requests specific printed output.

Input Variables -

PTYPE (Alphanumeric) - REQUIRED

Allowable values are BOUN, NODA, LOAD, ALL, NONE and LIMI.

BOUN - For printing the displacements and tractions, or corresponding quantities such as temperature, pressure, and flux at all boundary source points

NODA - For printing the displacements, stresses, and strains at all geometry nodes on the boundary. (available only for linear elasticity).

LOAD - For printing the resultant load value on each boundary element and the total load equilibrium of each region, excluding resultant body force.

ALL - For printing BOUN, NODA, and LOAD information with a single request.

NONE - To produce only a minimal printed output.

LIMI - To printout the current limits of **BEST-CMS.**

FEAT - To printout a table reporting the current implementation status of **BEST-CMS** special features in file "BEST-CMS.FEATURES". An integer value may be included after the keyword FEAT (e.g. PRINT FEATURES 80) to indicate the number of lines per page used in the table. The default is 66 which corresponds to the number of lines printed per page by a standard line printer.

Additional Information -

For printing two or more types of output, a separate PRIN request must be included for each type.

If a PRIN, BOUN, NODA or LOAD request does not appear in the case control input then all three types of output (BOUN, NODA, and LOAD) will be printed by default.

Results for sampling points and volume source points are automatically printed, unless a PRINT NONE request is included.

Examples of Use -

1. In the elastic analysis of a rotor, print out the resultant boundary element loads.

```
**CASE
  TITLE  ROTOR - ELASTIC ANALYSIS
  TIMES  1.0   2.5   6.0
  ELASTIC
  PRINT LOAD
```

CALC   CTYPE

Status - OPTIONAL

Full Keyword - CALCULATE

Function - Requests calculation of specific quantities.

Input Variables - CTYPE   (Alphanumeric) - REQUIRED

Allowable values are BOUN, NODA, and LOAD.

BOUN - For calculating the displacements and tractions, or corresponding quantities such as temperature, pressure, and flux at all boundary source points.

NODA - For calculating the displacements, stresses, and strains at all geometry nodes on the boundary. (Available only for concurrent thermoelasticity and consolidation.)

LOAD - For calculating the resultant load value on each boundary element and the total load equilibrium of each region, exclusing resultant body force.

Additional Information -

For calculating two or more types of output, a separate CALC request must be included for each type.

The CALC card should only be used for quantities that are not requested for printout, but which may be needed for post-processing. Thus, if CALC LOAD is used along with a NEUTRAL RESULTS card, then the **BEST-CMS** neutral file will include the resultant loads for elements.

Examples of Use -

1. In the thermal analysis of a rotor, calculate the boundary temperature and flux, but print only resultant heat loads. The neutral file result file will include the boundary temperatures and flux, as well as the heat loads.

```
**CASE
  TITLE  ROTOR - THERMAL ANALYSIS
  HEAT
  CALC BOUN
  PRIN LOAD
  NEUTRAL RESULTS
  PATRAN
```

NEUT   NTYPE

Status - OPTIONAL

Full Keyword - TIMES

Function - Requests the generation of a **BEST-CMS** result neutral file written in ASCI format. This neutral file is a **formatted, direct accessed file** called NEU-TRAL.OUT. The file contains material and geometric data as well as result information. More information on this can be found in Chapter 8.

Input Variables -

NTYPE   (Alphanumeric)   -   REQUIRED

Allowable value is PRINT.

PRINT - Create a second file called NEUTRAL.READ which is a **formatted sequential file** which may be accessed by an editor or output on a printer.

Additional Information -

Upon completion of the solver phase of an analysis, the file NEUTRAL.OUT is produced containing the requested information. This file can function as the interface to user supplied postprocessing programs.

Examples of Use -

1. Generate an ASCI neutral result file for the steady-state heat conduction analysis of a casting mold.

```
**CASE
  TITLE  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTART HOLE
  PRECISION LOW
  NEUTRAL RESULTS
```

5.1.9	MISCELLANEOUS CONTROL OPTIONS

CHEC

Status - OPTIONAL

Full Keyword - CHECK

Function - Perform only input data checking, and printout an error summary. No analysis is performed.

Input Variables - NONE

Additional Information -

This option is often useful for checking the input data for a new model. In addition to the error summary, all of the relevant material, geometry and boundary condition information is processed and printed in tabular form. Of particular interest is the identification of the boundary and volume source points, since these are determined by the program based upon the element and cell functional variation.

Examples of Use -

1. Check the input data for a thermoelastic turbine blade model.


```
**CASE
  TITLE  TURBINE BLACE A7311 - THERMOELASTIC
  CTHERMAL STEADY
  RESTART WRITE
  CHECK  $  DATA CHECKING ONLY
```

FILE   DNAME

Status - OPTIONAL

Full Keyword - FILE-CONTROL

Function - Specifies the directory in which the scratch files (FTNNN) reside during execution of **BEST-CMS**

Input Variables -

DNAME   (Alphanumeric)   -   REQUIRED

Allowable values are DIRNAMEALL or DIRNAMEFTNNN

DIRNAMEALL - All scratch files (FTNNN) all be created, accessed and stored in directory specified by DIRNAME.

DIRNAMEFTNNN - Scratch file (FTNNN) will be created, accessed and stored in directory specified by DIRNAME.

Additional Information -

Default storage location for scratch files (FTNNN) is the current directory boldtbl BEST-CMS is being run from

FT file numbers have to be specified in a 3 digit format. For example FT009 is correct while FT9 or FT09 are incorrect.

When specifying an individual FT file, only that file will be created in the specified directory, the rest will be created in the current directory.

FILE directive can be used multiple times, e.g.
        FILE /home/scr1/ALL
        FILE /home/temp/FT035
will cause all FT files except FT035 to be created in /home/scr1 directory. But if a combination such as
        FILE /home/temp/FT035
        FILE /home/scr1/ALL
is used, this will cause all FT files to be created in /home/scr1 directory.

Examples of Use -

1. Specify the directory /home/scr1 to receive FT037 and /home/scr2 to receive FT038. The rest of the FT files will be created in the current directory (UNIX systems).

```
**CASE
  TITLE  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  PRECISION LOW
```

```
NEUTRAL RESULTS
FILE   /home/scr1/FT037
FILE   /home/scr2/FT038
```

2. Specify the directory /home/scr1 to receive all FT files except FT037, which will be created in /home/scr2 directory (UNIX systems).

```
**CASE
TITLE  MOLD COMPONENT 6 - STEADY CONDITIONS
HEAT
PRECISION LOW
NEUTRAL RESULTS
FILE   /home/scr1/ALL
FILE   /home/scr2/FT037
```

CVCP RATIO

Status - OPTIONAL

Full Keyword - CYLINDRICAL VOLUME CELL PARAMETER

Function -

Specifies the radius of a cylindrical volume cell as a function of the radius of the inserts in a plastic or creep analysis involving inserts.

Input Variables -

RATIO (Alphanumeric) - REQUIRED

(radius of cylindrical volume cell) = (RATIO) * (prescribed radius of insert)

Additional Information -

Required in plasticity and creep analysis with inserts.

If the keywords PLAS or CREEP are present in **CASE, and CVCP is included, the cylindrical volume cells around the fibers will automatically be generated.

CVCP may only be input once. Therefore, inserts of different sizes will have cylindrical volume cells of different sizes.

The radius of the cylindrical volume cell must not intersect the surface of an insert or the outer surface of the composite.

Cylindrical volume cells may intersect each other. However, this is not recommended.

Examples of Use -

1. Specify the radius of a cylindrical volume cell as a function of the radius of the insert by providing a ratio of 1.56 for a plasticity analysis of a fiber composite with inserts.

```
**CASE
  TITLE    PLASTICITY IN A FIBER COMPOSITE
  TIMES   1.0   2.0   3.0   4.0   5.0   6.0
  PLASTICITY
  CVCP  1.56
```

EXAC

Status - OPTIONAL

Full Keyword - **EXACT**

Function - Specifies that the exact insert approach is to be employed.

Input Variables - NONE

Additional Information -

> This is an alternate formulation for fiber composite analysis. In this algorithm the fibers are integrated and the resulting equations are backsubstituted into the equations for the composite matrix (see Section 2.5). If this card is NOT input a formulation is used in which the equations for the fiber have been analytically substituted in the composite matrix equation (see Section 2.2). The latter formulation assumes the Poisson ratio of the fiber is the same as the composite matrix. The latter does not make this assumption, hence the name EXACT.

> The transient heat conduction and the transient uncoupled thermoelastic analysis are based on the EXACT formulation and assume EXACT for the default.

> Analysis with sampling points assume the EXACT formulation.

> The use of this card increases computation time of the analysis.

Examples of Use -

```
**CASE
  TITLE   TURBINE BLADE
  ELASTIC
  TIME    1.0   2.0
  EXACT
```

NUME

Status - OPTIONAL

Full Keyword - **NUMERICAL**

Function -

> Specifies integration of the fiber insert will be numerical integration over the entire fiber.

Input Variables - NONE

Additional Information -

> If this card is not input semi-analytical integration (analytical in the circumferential direction, numerical in the axial direction) is assumed.

> The use of this card will increase the computational time of the analysis.

> This card is intended to be used with fiber inserts that have curvature in the axial direction.

Examples of Use -

```
**CASE
  TITLE   TURBINE BLADE NO. 2
  ELASTIC
  NUMERICAL
```

5.2 | MATERIAL PROPERTY DEFINITION

This input section defines the properties of the various materials used in an analysis. A complete set of material property input must be provided for each material used. At least one set must be input for every analysis. A consistent set of units must be used for all properties.

A list of keywords recognized in the Material input are given below and a detailed description follows.

SECTION	**KEYWORD**	**PURPOSE**

5.2.1 Material Property Input Card
          MATE   Beginning of a material property input set

5.2.2 Material Identification
          ID   Identifier of a material type

5.2.3 Mass Parameter
          DENS   material mass density

5.2.4 Isotropic Elastic Parameters
          EMOD   Young's modulus
          POIS   Poisson's ratio

5.2.5 Isotropic Thermal Parameters
          COND   conductivity of material
          SPEC   specific heat

5.2.6 Isotropic Temperature-dependent Thermoelastic Parameters
          TEMP   temperature values at which elastic material properties will be defined
          EMOD   Young's modulus
          ALPH   coefficients of thermal expansion

SECTION	KEYWORD	PURPOSE

5.2.7 Elastoplastic and Creep Parameters

| | INEL | signals the beginning of an inelastic material model input |
| | TIME | time for reversal of loading |

5.2.7.1   Von Mises Model

	VON	Von Mises material model
	YIEL	proportional limits of linear elastic behavior
	CURV	stress-strain curve for Von Mises model with isotropic hardening

5.2.7.2   Creep Model

| | CREEP | creep model |

5.2.7.3   Fracture Model (With Hardening)

	FRAC	fracture model
	CZON	compression zone parameters
	TZON	tension zone parameters

Note: Refer to the following table for a list of required material properties

A list of material properties required for different types of analysis are defined below:

$$\boxed{\text{REQUIRED MATERIAL PROPERTIES}}$$

TYPE OF ANALYSIS	MATERIAL PROPERTIES
1. Isotropic Elastic Stress Analysis	EMOD, POIS
2. Heat Transfer Analysis	
2a. Steady-state (Potential Flow)	COND
2b. Transient (Diffusion)	COND, DENS, SPEC
3. Concurrent Thermoelastic Analysis	
3a. Steady-state	TEMP, EMOD, POIS, ALPH, COND
3b. Transient (Quasistatic)	TEMP, EMOD, POIS, ALPH, COND, DENS, SPEC
4. Plasticity and Creep Analysis:	
4a. Von Mises model	EMOD, POIS, YIEL, CURV
4b. Creep Model	EMOD, POIS, CREEP
4c. Fracture model	EMOD, POIS, CZON, TZON

**MATE

Status - REQUIRED

Full Keyword - MATERIAL PROPERTY

Function - Signals the beginning of a material property definition.

Input Variables - NONE

Additional Information -

A complete set of material property input must be provided for each material used.

All materials for a problem must be defined before any geometry is specified.

Examples of Use -

1. Define the elastic material properties for a carbon steel.

```
**MATE
  ID STEEL
     EMOD    30.3+6
     POIS    0.30
```

ID   NAME

Status - REQUIRED

Full Keyword - ID

Function - Provides an identifier for a set of material properties related to a given material, thereby allowing later reference to the material property definition.

Input Variables -

NAME   (Alphanumeric)   - REQUIRED

Additional Information -

The specified name must be unique compared to all other material names included in the problem.

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

Examples of Use -

1. Define the thermal properties for an aluminum alloy 3003.

```
**MATERIAL
  ID ALUM3003
      COND   25.0
      DENS   0.1
      SPEC   2000.
```

DENS   DEN1

Status - (see required material property table)

Full Keyword - DENSITY

Function - Defines the material mass density.

Input Variables -

DEN1   (Real)   - REQUIRED

Additional Information - NONE

Examples of Use -

1. Define material properties for a free vibration analysis.

```
**MATE
  ID  STEEL
     EMOD    30.E+6        $       PSI
     POIS    0.30
     DENS    7.324E-4
```

EMOD   EM1

Status - (see required material property table)

Full Keyword - EMODULUS

Function - Defines values of Young's modulus

Input Variables -

EM1   (Real)   - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify a elastic material.

```
**MATERIAL
  ID  MAT1
     EMOD      1.E6
     POIS      0.36
     DENS      0.15
```

POIS   POI

Status - (see required material property table)

Full Keyword - POISSON

Function - Defines the (temperature independent) value of Poisson's ratio.

Input Variables -

POI   (Real)   - REQUIRED

Allowable values - $-1.0 < POI \leq 0.5$

Additional Information - NONE

Examples of Use -

1. Specify room temperature elastic properties of carbon steel.

```
**MATE
  ID  STEEL
     EMOD  30.E6
     POIS  0.30
```

COND   CD1

Status - REQUIRED (for concurrent thermoelastic or heat conduction analysis)

Full Keyword - CONDUCTIVITY

Function - Defines the isotropic conductivity.

Input Variables -

CD1   (Real)   - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify thermal properties of aluminum for steady-state heat conduction.

```
**MATE
  ID  ALUM
     CONDUCTIVITY  25.0
```

1. Specify thermoelastic properties for a quasistatic analysis.

```
**MATE
  ID  M200
     TEMP  500.0
     EMOD  1.0E+6
     POIS  0.24
     ALPH  1.E-5
     COND  5.86
     DENS  0.05
     SPEC  215.
```

SPEC   SP1

Status - REQUIRED (for transient concurrent thermoelasticity or heat conduction)

Full Keyword - **SPECIFIC**

Function - Defines the specific heat.

Input Variables -

SP1   (Real)   - REQUIRED

Additional Information -

The user must be careful in selecting appropriate units for specific heat. The CONDuctivity divided by the product of DENSity times SPECific equals the diffusivity. The diffusivity must have units of (length**2)/time.

Examples of Use -

1. Material model for transient heat conduction.

```
**MATE
  ID  STEEL
      COND   5.8      $      IN.-LB./(SEC.IN.F)
      DENS   0.283    $      LB/(IN3)
      SPEC   1000.    $      IN.-LB./(LB.F)
```

TEMP   TEM1 TEM2 ...   TEMN

Status - (see required material property table)

Full Keyword - **TEMPERATURES**

Function - Provides the temperature values at which elastic material properties will be defined.

Input Variables -

      TEM1   (Real)   - REQUIRED

      TEM2 ... TEMN   (Real)   - OPTIONAL

Additional Information -

At least one temperature value must be input. If only one value is input then the properties will be treated as temperature independent.

The temperature values must be specified in ascending order.

This input may be continued on more than one card. Each card must begin with the keyword TEMP.

A maximum of 21 temperature values may be specified for the material.

Examples of Use -

1. Define a material with temperature varying Young's modulus.

```
**MATE
  ID  MAT0IT
     TEMP  50.0   300.0   500.0
     EMOD  30.3+6  29E+6   27.5E+6
     POIS  0.3
```

Top figure

**MATERIAL

TEMP	1000.0	2000.0	3000.0	5000.0
EMOD	1.0E+05	2.0E+05	2.0E+05	3.0E+05
ALPHA	1.0E-05	2.0E-05	2.0E-05	2.0E-05

EMOD ($\times 10^5$)

3.0

2.0

implied

1.0

explicitly defined

implied

0   1   2   3   4   5   6   7

TEMPERATURE ($\times 10^3$)

Bottom figure

**MATERIAL

TEMP	1000.0	2000.0	3000.0	5000.0
EMOD	1.0E+05	2.0E+05	2.0E+05	3.0E+05
ALPHA	1.0E-05	2.0E-05	2.0E-05	2.0E-05

ALPHA ($\times 10^{-5}$)

3.0

2.0

implied

1.0

explicitly defined

implied

0   1   2   3   4   5   6   7

TEMPERATURE ($\times 10^3$)

Figures for **MATE: TEMP, EMOD and ALPHA cards

Definition for temperature dependent elastic modulus and thermal coefficient of expansion

EMOD   EM1 EM2 ...   EMN

Status - (see required material property table)

Full Keyword - EMODULUS

Function - Defines values of Young's modulus at the temperature values specified on the TEMP card(s).

Input Variables -

EM1   (Real)   - REQUIRED

EM2 ... EMN   (Real)   - OPTIONAL

Additional Information -

At least one value must be input.

This input may be continued on more than one card. Each card must begin with the keyword EMOD.

Examples of Use -

1. Specify a thermally-dependent elastic material.

```
**MATERIAL
  ID  MAT1
     TEMP      0.0      200.0     300.0     350.0     400.0
     TEMP      450.0    500.0
     EMOD      1.E6     0.95E6    0.9E6     0.83E6    0.7E6
     EMOD      0.6E6    0.35E6
     POIS      0.36
     DENS      0.15
```

ALPH   AL1 AL2 ...   ALN

Status - REQUIRED (for thermal stress analysis)

Full Keyword - ALPHA

Function - Defines the values of the coefficient of thermal expansion at the temperature values specified on TEMP cards.

Input Variables -

     AL1   (Real)   - REQUIRED

     AL2 ... ALN   (Real)   - OPTIONAL

Additional Information -

     At least one value must be input.

     This input may be continued on more than one card. Each card must begin with the keyword ALPH.

Examples of Use -

     1. Define a thermoelastic material model.

```
**MATE
  ID ALUM
     TEMP     0.0      200.0     300.0
     EMOD     10.E3    9.8E3     8.E3
     ALPH     13.E-6  12.E-6    10.5E-6
     POIS     0.33
```

INEL

Status - OPTIONAL (for plastic and creep analysis)

Full Keyword - INELASTIC

Function - Identifies the fact that an inelastic material model will be defined.

Input Variables - NONE

Additional Information -

If the INEL card is used during a material property definition, the program will then expect one of three models to be chosen. These models are specified using the VON, CREEP or FRAC keywords. Only one of these three cards may be used in a single material property input data set. Input for the three nonlinear models is described below.

NOTE: In present version of the program, all non-linear materials used in an analysis must be based on the same model.

Examples of Use -

1. Request an inelastic analysis using the Von Mises Material Model.

```
**MATERIAL
   ID MAT1
   TEMP 70.0
   EMOD 7000.
   POIS 0.2
   INELASTIC
      VON MISES
```

TIME   I1 I2 ... IN

Status - OPTIONAL

Full Keyword - TIME

Function - Defines time steps at which the accelerated nonlinear solution algorithm should not be used.

Input Variables -

     I1   (Integer)   - REQUIRED

     I2 ... IN   (Integer)   - OPTIONAL

Additional Information -

In general it is recommended that the TIME card be used to turn off the accelerated algorithm whenever the loading is reversed. The accelerated algorithm attempts to extrapolate initial stress based on recent history and will extrapolate in the wrong direction at times when loading reverses.

Not available for creep analysis.

A maximum of 20 time steps may be input.

Examples of Use -

1. Specify that a reversal of loading takes place at two time step numbers during cyclic loading of a material.


```
**MATERIAL
    ID MAT1
    TEMP 70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    VON MISES
    TIME  3.0   9.0
```

VON

Status - OPTIONAL

Full Keyword - VON MISES MODEL

Function - Selects the use of the inelastic Von Mises Model in which the stress-strain curve is defined as a set of equivalent stress-equivalent plastic strain pairs, with linear interpolation between points.

Input Variables - NONE

Additional Information -

The input for this model is extremely easy to generate if a monotonic stress-strain curve is available for the material. The model is intended primarily for the analysis of monotonic loading situations.

Examples of Use -

1. Request the use of the Von Mises Model for an inelastic analysis.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
```

Von Mises model with variable strain hardening
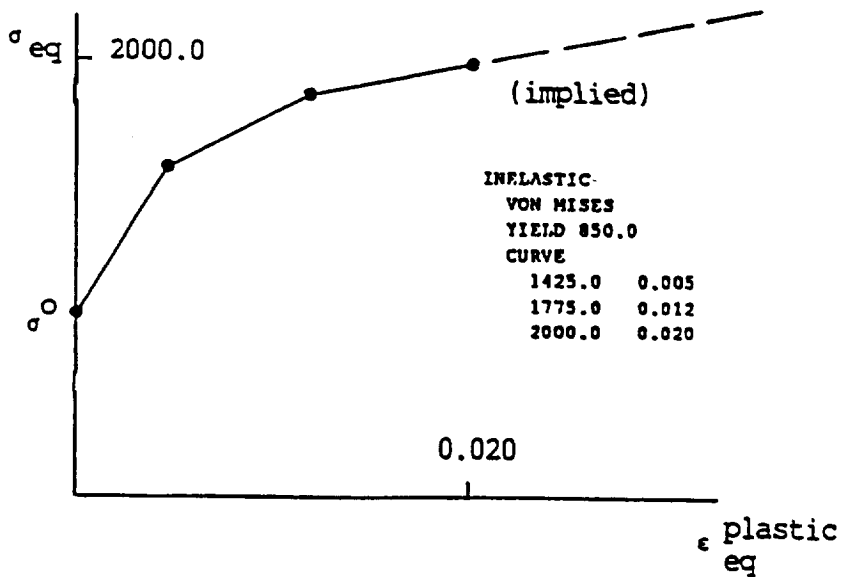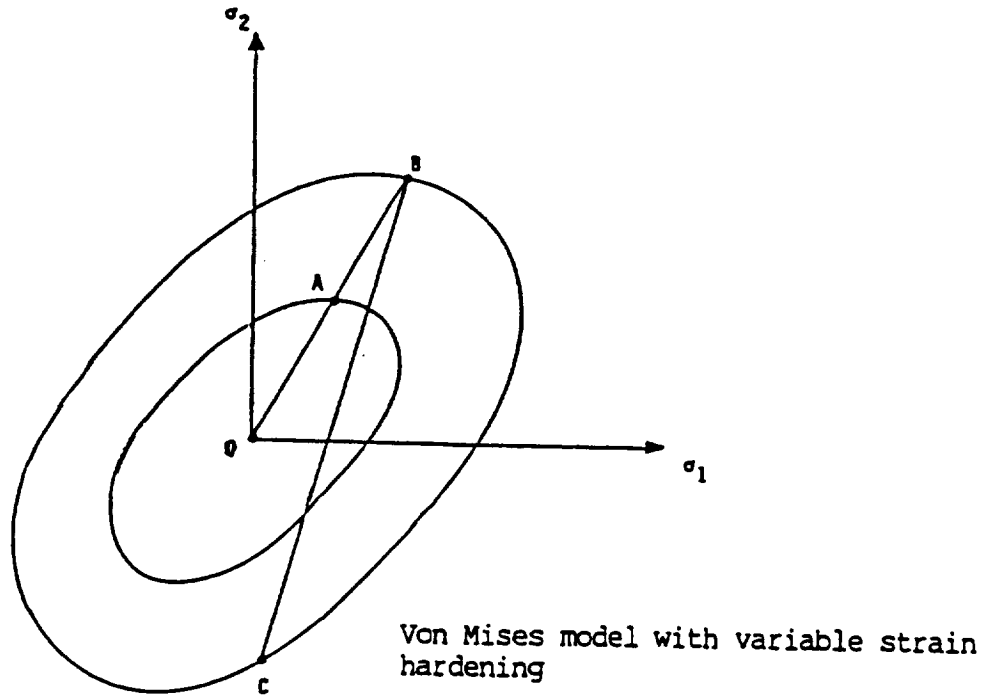
Figure for **MATE: VON (YIELD and CURVE) cards

VON MISES equivalent stress/equivalent plastic strain hardening curve

YIEL   Y1

Status - REQUIRED (if VON is input)

Full Keyword - YIELD

Function - Defines the proportional limit for the isotropic inelastic material models.

Input Variables -

Y1   (Real)   - REQUIRED
Inner proportional limit.

Additional Information -

Y1 is the normal proportional limit, at which the transition away from purely elastic behavior occurs.

Examples of Use -

1. Specify the yield stress value at which the material ceases to behave in an elastic manner.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
    YIELD 24.3
```

CURV

Status - REQUIRED (with VON only)

Full Keyword - CURVE

Function - Identifies the beginning of the input defining the stress-strain curve for the isotropic model.

Input Variables - NONE

Additional Information -

> This card is immediately followed by input cards defining the points on the equivalent stress - equivalent plastic strain curve. These data cards are described immediately below.

(NONE)   SIG1   EP1

Status - REQUIRED (if CURV is input)

Full Keyword - NO KEYWORD REQUIRED

Function - To define a single point on the material stress strain curve.

Input Variables -

> SIG1   (Real)   - REQUIRED
> Stress value
>
> EP1   (Real)   - REQUIRED
> Equivalent plastic strain value

Additional Information -

> This card type is input once for each point used to describe the stress-strain curve. The stress values must be monotonically increasing. The point (yield stress,0) is inserted automatically by the program and must not be input by the user.
>
> In order to define a perfectly plastic material the user should input a single point at which the stress is the yield stress and the equivalent plastic strain has any positive value.
>
> A maximum of nineteen points (excluding the initial point) can be input by the user.
>
> Linear interpolation is used between points. Perfectly plastic material behavior is assumed after the last point.

Examples of Use -

1. Define the equivalent stress-equivalent plastic strain curve for a Von Mises
   material.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
    YIELD 100.
    CURVE
       150. 0.25
       180. 0.5
       190. 0.75
       200. 1.0
```

CREEP GAMMA POWER

Status - OPTIONAL

Full Keyword - CREEP MODEL

Function - Selects the use of the creep model based on the power law.

Input Variables -
GAMMA - (Real)
The value of the multiplier (γ) in the power law.

POWER - (Real) - REQUIRED
The value of the exponent(n) in the power law.

Additional Information -

- The multiaxial creep behavior is defined using the power law

$$\dot{\epsilon}_{ij}^{c} = \gamma \sigma^{n} \frac{\partial Q}{\partial \sigma_{ij}}$$

where the dot indicates a derivative with respect to time, σ is the current equivalent Von Mises stress, Q is the plastic potential assumed to be the Von Mises yield function (zero hardening), and γ and n are material parameters input by the user on this card.

- The time step increment is limited by the values of POWER and GAMMA input on this card. (See CREEP card in **CASE.)

- This model is primarily intended for the creep analysis under constant or monotonic loading situations.

Examples of Use -

1. Request the use of the Creep Model for an inelastic analysis.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    CREEP  3.0  1.8E-18
```

FRAC

Status - OPTIONAL

Full Keyword - **FRACTURE MODEL**

Function - Selects the use of the plastic facture model.

Input Variables - NONE

Additional Information -

Reference - Chen, W.F., Limit Analysis and Soil Plasticity, Elsevier, New York, 1975.

Only available with the iterative plasticity algorithm.

Examples of Use -

1. Request the use of the fracture model for an inelatic analysis.


```
**MATERIAL
  ID MAT1
  TEMP  70.0
  EMOD 3.7E+06
  POIS 0.2
  INELASTIC
    FRACTURE
      CZONE  1134.0  2207.0   699.0  1165.0  0.15E+06
      TZONE   327.0   545.0  1215.0  2025.0  0.16E+06
```
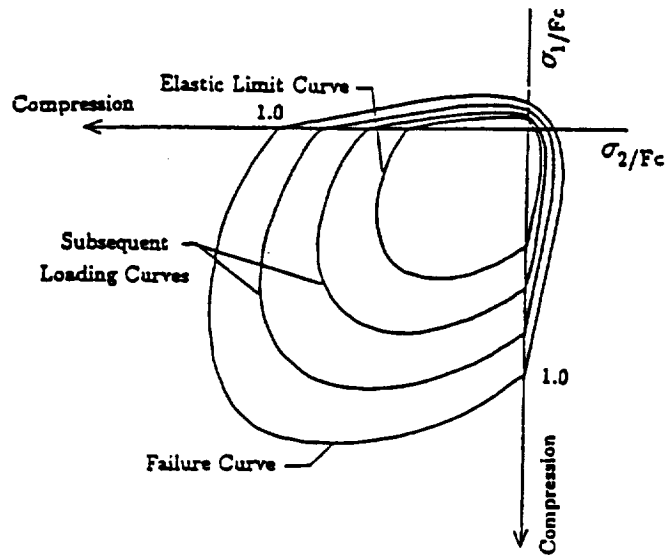
Figure for FRAC card

Loading Function in Biaxial Principal Stress Space

CZON   **TL   TU   AL   AU   HARD**

Status - REQUIRED (if FRAC is input)

Full Keyword - CZONE (compression zone)

Function -

Defines the material properties for the compression zone of the fracture model.

Input Variables -

TL   (Real)   - REQUIRED (Elastic Limit; TL = $\sqrt{F_{el}}$)
TU   (Real)   - REQUIRED (Ultimate Limit; TL = $\sqrt{F_{ult}}$)
AL   (Real)   - REQUIRED (Shape parameter at elastic limit)
AU   (Real)   - REQUIRED (Shape parameter at ultimate stress)
HARD   (Real)   - REQUIRED (Constant hardening parameter)

Additional Information -

Reference - Chen, W.F., Limit Analysis and Soil Plasticity, Elsevier, New York, 1975.

Examples of Use -

1. Specify the compression zone material properties for an inelastic analysis using the fracture model.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 3.7E+06
  POIS 0.2
  INELASTIC
    FRACTURE
      CZONE   1134.0   2207.0    699.0   1165.0   0.05E+06
      TZONE    327.0    545.0   1215.0   2025.0   0.16E+06
```

TZON   TL   TU   AL   AU   HARD

Status - REQUIRED (if FRAC is input)

Full Keyword - TZONE (tension zone)

Function -

Defines the material properties for the tension zone of the fracture model.

Input Variables -

TL   (Real)   - REQUIRED (Elastic Limit; $TL = \sqrt{F_{el}}$)
TU   (Real)   - REQUIRED (Ultimate Limit; $TL = \sqrt{F_{ult}}$)
AL   (Real)   - REQUIRED (Shape parameter at elastic limit)
AU   (Real)   - REQUIRED (Shape parameter at ultimate stress)
HARD   (Real)   - REQUIRED (Constant hardening parameter)

Additional Information -

Reference - Chen, W.F., Limit Analysis and Soil Plasticity, Elsevier, New York, 1975.

Examples of Use -

1. Specify the compression zone material properties for an inelastic analysis using the fracture model.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 3.7E+06
  POIS 0.2
  INELASTIC
    FRACTURE
      CZONE   1134.0   2207.0    699.0   1165.0   0.05E+06
      TZONE    327.0    545.0   1215.0   2025.0   0.16E+06
```

5.3 | DEFINITION OF GEOMETRY

In the current version of **BEST-CMS**, surface geometry is defined using six, eight, and nine noded parametric surface patches for 3-D problems. These patches can be defined to have either linear, quadratic or quartic variation of the primary field variables. Additionally, fiber inserts on the interior of a 3-D body can be defined via two and three noded line elements. The insert elements can have either linear or quadratic functional variation in the longitudinal direction. An entire model may be assembled from several generic modelling regions (GMR). Each generic modelling region is defined in a single block of input introduced with a **GMR card.

The information provided in a single GMR input block consists of six main types:

    1 - Region identification

    2 - Nodal point definition

    3 - Surface connectivity definition

    4 - Fiber or Insert definition

Keywords recognized in the GMR input are listed and a detailed description follows.

SECTION	**KEYWORD**	**PURPOSE**
5.3.1 Geometry Input Card		
	**GMR	start of generic modelling region input
5.3.2 Region Identification		
	ID	region ID
	MATE	material property(set) for region
	TREF	reference temperature of region
	TINI	initial temperature of the region
5.3.3 Nodal Point Definition		
	POIN	nodal points for boundary discretization (Coordinates)
5.3.4 Surface Element Definition		
	SURF	beginnining of surface discretization
	TYPE LINE	linear surface variation of field quantities
	TYPE QUAD	quadratic surface variation of field quantities
	TYPE QUAR	quartic surface variation of field quantities
	ELEM	element list (element connectivity)
	TRAN	surface generation by translation
	REFE	axis of rotation for surface generation
	DIRE	direction of axis of rotation
	ROTA	angle of rotation for surface generation
	NORM	defines outer normal of surface

SECTION	**KEYWORD**	**PURPOSE**

5.3.5 Insert Element Definition

	INSE	beginning of Insert definition
	INTE	insert/matrix interface definition
	SPRI	spring coefficients
	RESI	thermal resistance across interface
	FRIC	coefficient of friction
	POIN	nodal points for insert discretization (coordinates)
	TYPE LINE	linear variation of field quantities of Insert
	TYPE QUAD	quadratic variation of field quantities of Insert
	ELEM	start of connectivity for each Insert (element connectivity and radius of insert)

5.3.6 Sampling Points

	SAMP POINT	start of definition of sampling points (coordinates)

5.3.7 Sampling Surfaces

	SAMP SURF	beginning of sampling surface discretization
	ELEM	sampling surface element list (sampling surface element connectivity)

5.3.1	GEOMETRY INPUT CARD

**GMR   TYPE

Status - REQUIRED

Full Keyword - GMREGION

Function - This card signals the beginning of the definition of a generic modelling region.

Input Variables -

TYPE (alphanumeric) - OPTIONAL
Allowable Value is MICRo-surface

Specifies that 'microsurface' elements will be automatically generated using basic user defined geometric primitives of the standard insert elements.

Additional Information -

At least one GMR must be defined for an analysis. If more than one GMR is defined, then the input for each is initiated with a **GMR card.

GMR definitions must all precede all Interface and Boundary Condition set. Each GMR must be a closed region of two-dimensional space.

A GMR may have multiple internal boundaries in addition to a single external boundary.

If 'MICRO' appears on the **GMR card, then the insert element input in this GMR is used to create a boundary element discretization of the insert using standard three-dimensional boundary elements. The discretization of the insert elements from a single GMR are grouped together and placed in a program defined GMR which is named 'FIBER-XX' where XX is the two digit number defined by BEST-CMS. An identical discretization of the holes (or voids) containing the inserts is also created and placed in the original GMR. An interface set defining the interface definition between the insert and the holes is automatically created by BEST-CMS, as well as a material set defining the properties of the insert. The resulting data is equivalent to a data set for which a user has discretized each insert (and hole), defined material properties for the fiber insert, and the interface between the hole and the insert. The advantage of the micro-surface definition is that the input format is the same as the standard insert element input. Only the centerline and insert radius is required to define the geometry of the insert (and hole).

Currently, it is required that in a single data file, all insert definitions be either micro-surface inserts or standard inserts.

The micro-surface insert definition is very effective in analyses with curved fibers or fiber weaves. It is also effective in analyses with nonlinear interfaces between the fiber and matrix. Micro-surface insert definition is not available for analysis with nonlinear material behavior (plasticity or viscoplasticity).

Use of the micro-surface insert definition allows for detail analysis of the fiber. Displacement, tractions, stresses, strains, temperatures and fluxes are all calculated on the surface of the insert fiber (and hole) and results are output in the result file and in the Neutral file where they can be used in post-processing to view contour plots of these quantities on the fibers' surfaces.

Examples of Use -
1. A region in which the fiber insert definitions will be modeled with 'standard insert elements':

```
**GMR
  ID   REG1
  MATE   STEEL
  TREF   70.0
  TINI    0.0
  POINT
     1   10.0   0.0   2.0
     2   10.0   1.0   2.0
              .
              .
```

2. A region in which the fibers insert definition wil be modeled with micro-surface inserts:

```
**GMR   MICRO-SURFACE
  ID   REG1
  MATE   STEEL
  TREF
  TINI
  POINT
     1   10.0   0.0   2.0
     2   10.0   1.0   2.0
```

5.3.2	REGION IDENTIFICATION

ID   NAME

Status - REQUIRED

Full Keyword - ID

Function - This card provides the identifier for the GMR.

Input Variables -

NAME   (Alphanumeric)   - REQUIRED

Additional Information -

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

The name provided on this card is used to reference the GMR in other portions of the input as well as in the **BEST-CMS** output file.

The NAME must be unique compared to all the other GMR names defined in the problem.

Examples of Use -

```
**GMR
   ID REG1
   MATE STEEL
```

MATE   NAME

Status - REQUIRED

Full Keyword - **MATE**

Function - This card identifies the material property set for the GMR.

Input Variables -

NAME   (Alphanumeric)   - REQUIRED

Additional Information -

The material name reference must have been previously defined in the material property input (identified as NAME on the ID card in **MATE input).

Examples of Use -

```
**GMR
  ID GMR1
  MATE STEEL
```

TREF   TEMP

Status - OPTIONAL

Full Keyword - TREFERENCE

Function - Defines the reference temperature at which the material properties will be evaluated for use in integration of this GMR.

Input Variables -

TEMP   (Real)   - REQUIRED

Additional Information -

If temperature dependent material properties were input in **MATE, the properties for the GMR will be calculated based on the temperature specified on this card using linear interpolation.

For problems in which the temperature changes in time and/or space, it is recommended that the reference temperature be chosen as the (time/volume weighted) average temperature over the GMR.

If this card is not input then a reference temperature of zero is assumed.

Examples of Use -

1. Specify the reference temperature at which the material properties are evaluated.

```
**GMR
  ID REG1
  MATE MAT1
  TREF 70.0
```

`TINI    TEMP`

Status - OPTIONAL (used in temperature dependent problems)

Full Keyword - `TINITIAL`

Function - This card defines the initial temperature (i.e. the datum temperature or the zero stress-strain state) of the region at the beginning of a temperature dependent problem.

Input Variables -

> TEMP   (Real)   - REQUIRED

Additional Information -

> If this card is not input, the initial temperature is assumed as zero.

Examples of Use -

> 1. Specify the initial temperature of the region REG1.

```
**GMR
  ID REG1
  MATE MAT1
  TREF 70.0
  TINI  0.0
```

5.3.3	NODAL POINT DEFINITION

POIN

Status - REQUIRED (for defining the GMR)

Full Keyword - POINTS

Function - This card initiates the definition of nodal points for the boundary elements of the GMR.

Input Variables - NONE

Additional Information -

Under certain conditions **BEST-CMS** can automatically generate nodal points. This is done by translation and/or rotation of previously defined surfaces. At the beginning of the definition of the first GMR some points must be defined in order to begin the process, although explicit definition of all nodal points in the first GMR is not required. It is possible that GMRs defined subsequent to the first may require no explicit nodal point input. For example, the case when a GMR is obtained by translation and/or rotation of previously defined surfaces.

Nodal points used for insert elements discretization **CANNOT** be defined here. Instead, the nodal points for inserts must be defined under their respective section.

(NONE)   NNODE X Y Z

Status - REQUIRED

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for a single nodal point.

Input Variables -

NNODE   (Integer)   - REQUIRED
User node number for the node.

X,Y,Z   (Real)   - REQUIRED
Cartesian coordinates of the node.

Additional Information -

This card is input once for each point.

User node numbering must be unique.

All node numbers must be less than or equal to 99999.

Nodal points used for insert elements **CANNOT** be defined here. Instead, the nodal points for inserts must be defined under their respective section.

Examples of Use -

1. Define a set of nodal coordinates in GMR1 for a 3-D analysis.

```
**GMR
  ID   GMR1
  MAT MAT1
  TREF 70.0
  HALF
  POINTS
        1     0.000     0.000     0.000
        2     0.700     0.700     0.000
        3     0.700     0.000     0.000
        4     0.000     0.700     0.000
        5     1.000     0.000     0.000
        .       .         .   .
        .       .         .   .
        .       .         .   .
```

5.3.4	SURFACE ELEMENT DEFINITION

SURF   NAME REFNAME

Status - REQUIRED (minimum of one per GMR)

Full Keyword - SURFACE

Function - This card initiates the definition of a surface of the current GMR.

Input Variables -

NAME   (Alphanumeric)   - REQUIRED

The name of the surface being defined.

REFNAME   (Alphanumeric)   - OPTIONAL

The name of a previously defined surface which will be used to create the current surface (defined as NAME on SURF card in prior GMR input).

Additional Information -

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

The names assigned to the various surfaces in the problem must be unique.

Two techniques are available to define a surface.

1 - A set of surface elements may be defined using explicitly defined nodes. In this case REFNAME is not input, and the card following the SURF card must be a TYPE card. (The first surface in the first GMR must be defined in this way).

2 - If REFNAME is input, then that (previously defined) surface will be used to define the surface NAME through translation and rotation. A TRAN card and/or a set of cards consisting of a REF, DIR and ROT cards must be included. Translation takes priority over rotation. Any new nodal points required will be automatically generated, and any duplicate nodes will be eliminated. The TYPE variation of the new surface is automatically assumed to be the same as the reference surface (this cannot be changed).

REFNAME can only be used in three dimensional analysis. Boundary conditions and/or interface conditions may be applied to the automatically generated surfaces, however, it is the user responsibility to insure that the elements and nodes generated by the program are referenced by the correct numbers. The RESTART WRITE option may be useful in this respect. In any case, boundary conditions that are applied to the entire surface can be input without complications.

The current verions of **BEST-CMS** allows RENAME to refer to a surface previously defined in the current GMR.

Examples of Use -

1. Define a 3-D quadratic surface named SIDE

```
SURFACE  SIDE
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
 1003  5  6  7  107  207  206  205  105
 1004  7  8  1  105  201  208  207  107
NORMAL 101 +
```

2. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE   SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

TYPE   ATYPE   BTYPE

Status - REQUIRED (if REFNAME not input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities over the elements of the current surface.

Input Variables -

ATYPE   (Alphanumeric)   - REQUIRED

Allowable values are LINE, QUAD and QUAR
    LINEar - linear shape function
    QUADratic - quadratic shape function
    QUARtic - quartic shape function

BTYPE   (Alphanumeric)   - OPTIONAL (for QUAD and QUAR)

Allowable values are LAGR and SERE. The default is SERE
    SEREndipity - Serendipity type shape function
    LAGRangian - Lagrangian type shape function

Additional Information -

See figure on subsequent pages.

A TYPE card must be defined for each surface.

All of the elements of a single surface must have the same type of variation. Different surfaces of the same GMR may have different variation. When three-dimensional surfaces of different variation types meet, the element sides along the junction are constrained to have the variation of the element with the lower order. (see figure on subsequent pages).

A surface may consist of a single element. By contrast a single surface may define the entire boundary of a GMR.

Examples of Use -

1. Specify that the field quantities vary quadratically over the elements of the surface SURF1.

```
SURFACE   SURF1
TYPE   QUAD SERE
ELEMENT
        101   1   2   3   103   203   202   201   101
        102   3   4   5   105   205   204   203   103
         .    .   .   .    .     .     .     .     .
         .    .   .   .    .     .     .     .     .
         .    .   .   .    .     .     .     .     .
         .    .   .   .    .     .     .     .     .
```

3D BOUNDARY ELEMENT FUNCTIONAL VARIATION

LINE

3 Source Points

4 Source Points

QUAD SERE

6 Source Points

8 Source Points

QUAD LAGR

6 Source Points

9 Source Points

QUAR SERE

15 Source Points

17 Source Points

QUAR LAGR

15 Source Points

25 Source Points
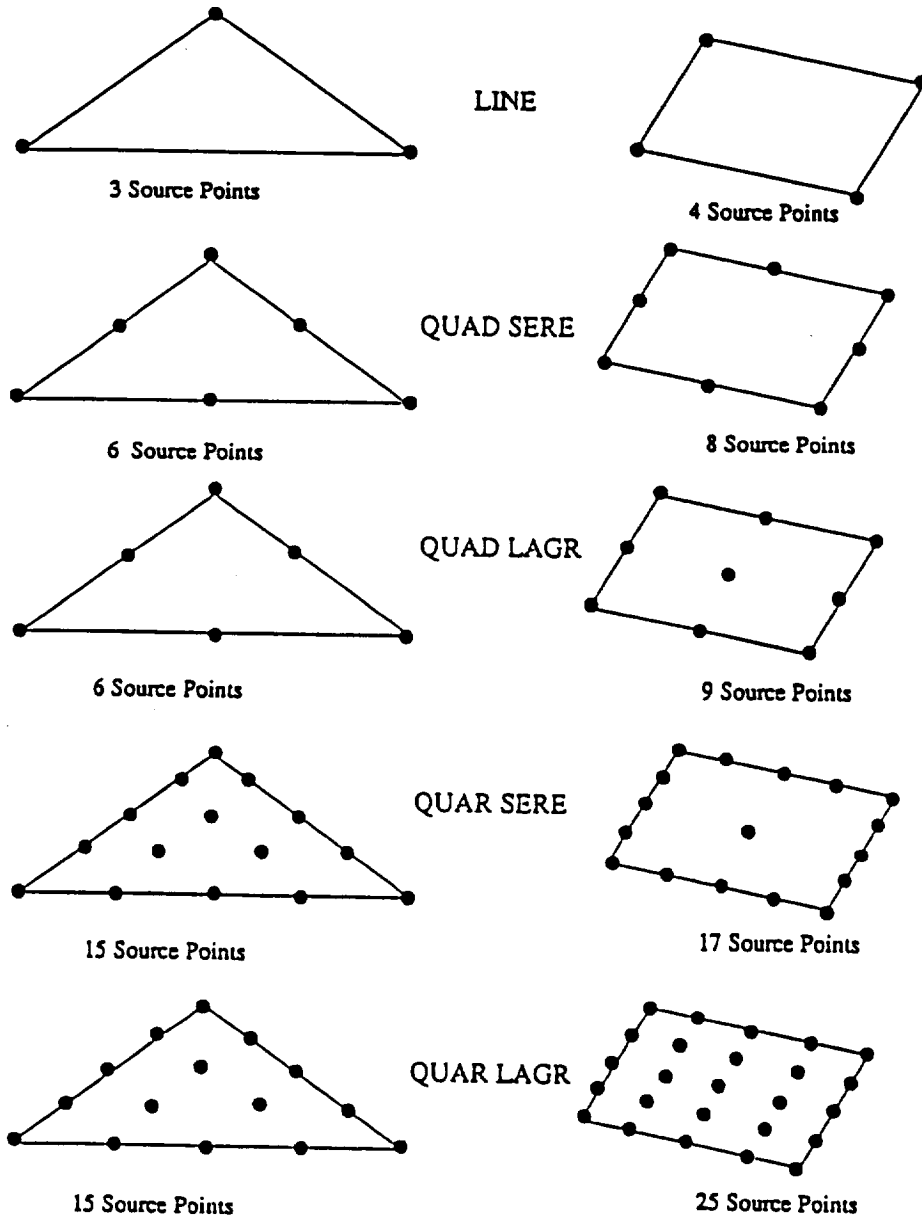
Figure for **GMR: TYPE card

(See element connectivity card for geometrical input)

L — LINEAR

Q — QUADRATIC

O — SOURCE POINTS RETAINED
IN SYSTEM EQUATIONS
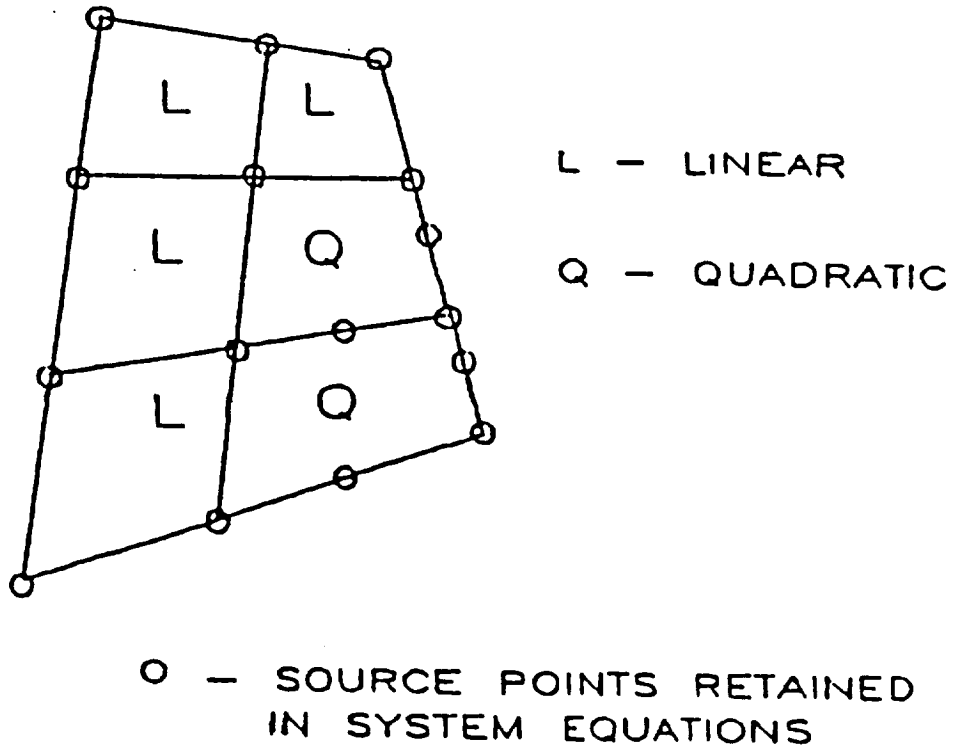
Figure for **GMR: TYPE card
Effect of mixed variation

ELEM

Status - REQUIRED (if REFNAME not input)

Full Keyword - **ELEMENTS**

Function - Signals the beginning of the connectivity definition for surface elements of the current surface.

Input Variables - NONE

Additional Information - NONE

(NONE)   NEL NODE1 ... NODEN

Status - REQUIRED (minimum of one card if TYPE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the connectivity for a single surface element.

Input Variables -

NEL   (Integer)   - REQUIRED

User element number.

NODE1 ... NODEN   (Integer)   - REQUIRED

User node numbers of the six, eight, or nine nodes for defining the geometry of the element. Every surface patch must have three, six, eight, or nine nodes, regardless of whether TYPE = LINE, QUAD or QUAR. (The shape functions for geometry is always quadratic)

Additional Information -

This card is input once for each element.

The input card need not specify whether a three, six, eight, or nine node element is being defined. The starting point in 3-D may be any of the corner nodes, however the direction of input around the element is arbitrary, as long as, input is consecutive.

User element numbers must be unique and less than or equal to 99999.

All of the nodes referenced in the surface element connectivity must have been defined previously in POINts.

Examples of Use -

1. Specify the connectivity definition for elements of the surface SIDE using four 8-noded quadratic elements.

```
SURFACE   SIDE
TYPE   QUAD
ELEMENT
   101   1   2   3   103   203   202   201   101
   102   3   4   5   105   205   204   203   103
   103   5   6   7   107   207   206   205   105
   104   7   8   1   105   201   208   207   107
NORMAL 101 +
```

3-D BOUNDARY ELEMENT FAMILY FOR GEOMETRICAL INPUT
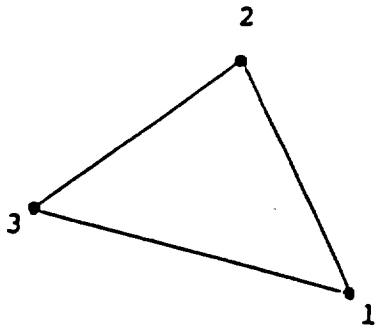
Linear 3-noded Element
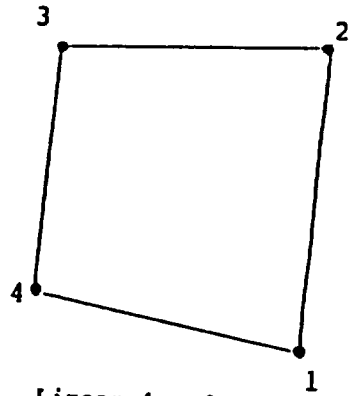
Linear 4-noded Element
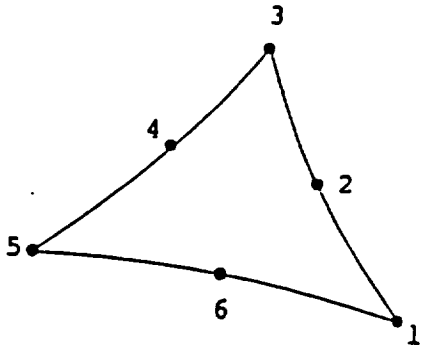
Quadratic 6-noded Element
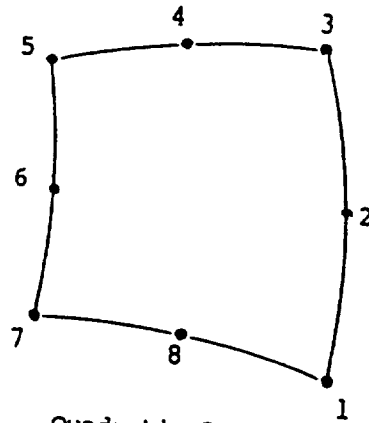
Quadratic 8-noded Element
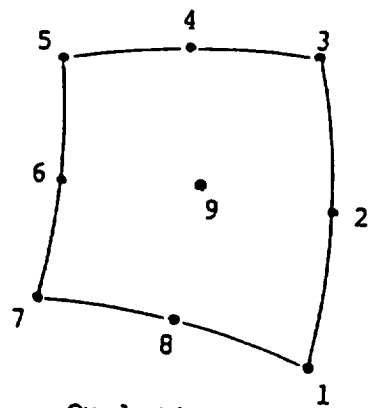
Quadratic 9-noded Element

Figure for **GMR: Element connectivity card
(see TYPE card for functional variation)

TRAN   DELX DELY DELZ

Status - OPTIONAL (if REFNAME was input)

Full Keyword - TRANSLATE

Function - Defines a translation to be applied to the reference surface in the creation of the current surface.

Input Variables -

DELX, DELY, DELZ   (Real)   - REQUIRED

Cartesian components of the translation.

Additional Information - Available for 3-D only.

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1   2   3   103   203   202   201   101
 1002  3   4   5   105   205   204   203   103
SURFACE SURNEW SURF1
 TRAN   1.0   -1.0   0.0
 REFE   1.0    0.0   0.0
 DIRE   0.0    0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

REFE   X Y Z

Status - OPTIONAL (if REFNAME was input)

Full Keyword - REFERENCE

Function - Defines a point on the axis defining the rotation.

Input Variables -

X,Y,Z   (Real)   - REQUIRED

Cartesian coordinates of a point on the axis of rotation.

Additional Information - None

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE   SURF1
TYPE   QUAD
ELEMENT
 1001  1   2   3   103   203   202   201   101
 1002  3   4   5   105   205   204   203   103
SURFACE  SURNEW  SURF1
 TRAN    1.0   -1.0    0.0
 REFE    1.0    0.0    0.0
 DIRE    0.0    0.0    1.0
 ROTA    180.0
NORMAL 1001 +
```

DIRE   D1 D2 D3

Status - OPTIONAL (if REFNAME was input)

Full Keyword - DIRECTION

Function - Defines the positive direction of the axis of rotation.

Input Variables -

D1,D2,D3   (Real)   - REQUIRED

Components of a vector along the positive direction of the axis of rotation.

Additional Information - None

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE   SURF1
TYPE   QUAD
ELEMENT
 1001  1   2   3   103   203   202   201   101
 1002  3   4   5   105   205   204   203   103
SURFACE  SURNEW  SURF1
 TRAN    1.0   -1.0    0.0
 REFE    1.0    0.0    0.0
 DIRE    0.0    0.0    1.0
 ROTA    180.0
NORMAL 1001 +
```

ROTA   THETA

Status - OPTIONAL (if REFNAME was input)

Full Keyword - **ROTATION**

Function - Defines the angle of rotation (in degrees) of the reference surface about the axis of rotation.

Input Variables -

THETA   (Real)   - REQUIRED
Angle of rotation (degrees).

Additional Information -

The angle of rotation is taken to be positive if it is counterclockwise when looking in the positive direction along the axis of rotation (right hand rule).

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE   SURF1
TYPE   QUAD
ELEMENT
 1001  1   2   3   103   203   202   201   101
 1002  3   4   5   105   205   204   203   103
SURFACE SURNEW SURF1
 TRAN   1.0   -1.0    0.0
 REFE   1.0    0.0    0.0
 DIRE   0.0    0.0    1.0
 ROTA   180.0
NORMAL 1001 +
```

```
NORM        NEL1 F1 NEL2 F2 ...  NELN FN
```

Status - REQUIRED

Full Keyword - NORMAL

Function - Defines the outer normal direction on each disjoint boundary of the current GMR.

Input Variables -

NEL1   (Integer)   - REQUIRED

User element number for an element in the current GMR.

F1   (Alphanumeric)   - REQUIRED

Flag relating the outer normal direction on the element NEL1 to the input node ordering. Allowable values for F1 are + and − .

NEL2 ...NELN   (Integer)   - OPTIONAL

F2 ...FN -AL - OPTIONAL

Additional (element number, flag) pairs used to define the outer normal direction on disjoint boundaries of the current GMR, if any exist.

Additional Information -

A 'disjoint' boundary should not be confused with a 'surface'. A surface is merely a user convenience, permitting the collection of any number of elements without regard to connectivity. On the other hand, each disjoint boundary is the collection of contiguous elements. For example, the user could employ a single surface to model a thick-walled hollow sphere, but there will always be two disjoint boundaries. (see Figures on subsequent pages)

The number of (element number, flag) pairs to be input for a given GMR is exactly equal to the number of disjoint boundaries which make up that GMR.

With the outer normal defined for one element on each disjoint boundary, the outer normals of all other elements in the (3-D) GMR are automatically determined by the program.

If the input ordering of the nodes for element NEL1 (or NEL2 or NELN) is counterclockwise when viewed from outside the element, then F1 = +. Otherwise F1 = − (right hand rule).

Examples of Use -

1. Define the direction of outward normal to the element 101 as being positive in relation to input node ordering.

```
**SURFACE   SIDE
  TYPE   QUAD
  ELEMENT
        101    1    2    3    103    203    202    201    101
        102    3    4    5    105    205    204    203    103
        103    5    6    7    107    207    206    205    105
        104    7    8    1    105    201    208    207    107
  NORMAL  101  +
```
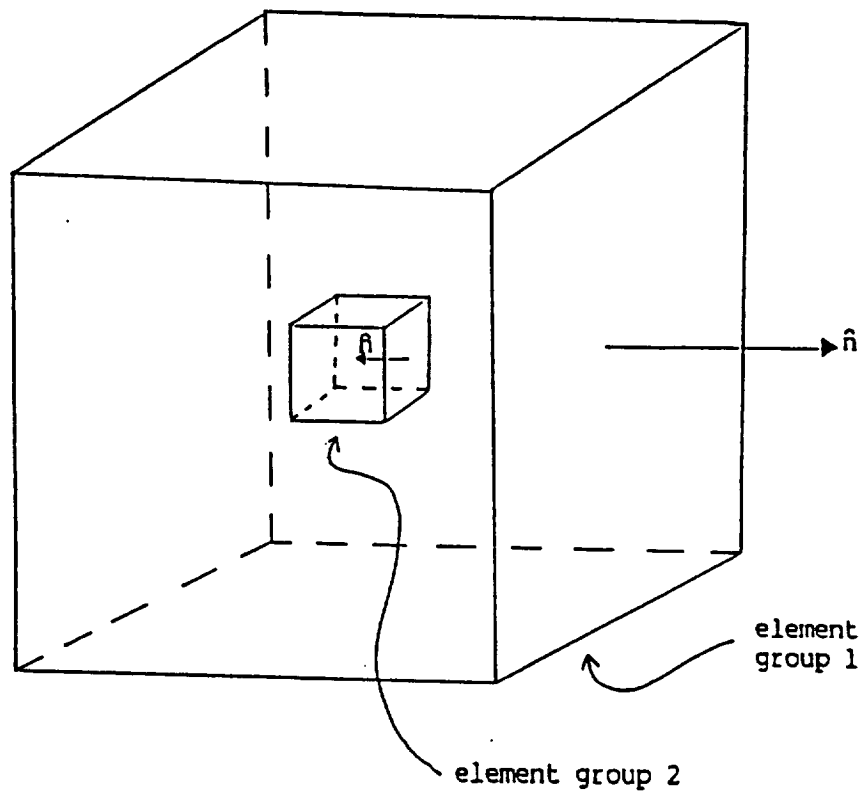
Element groups 1 and 2 are disjoint sections and
therefore normals for one element in each section
must be defined.

Figure for **GMR: NORM card
Three-dimensional disjoint (unconnected) elements
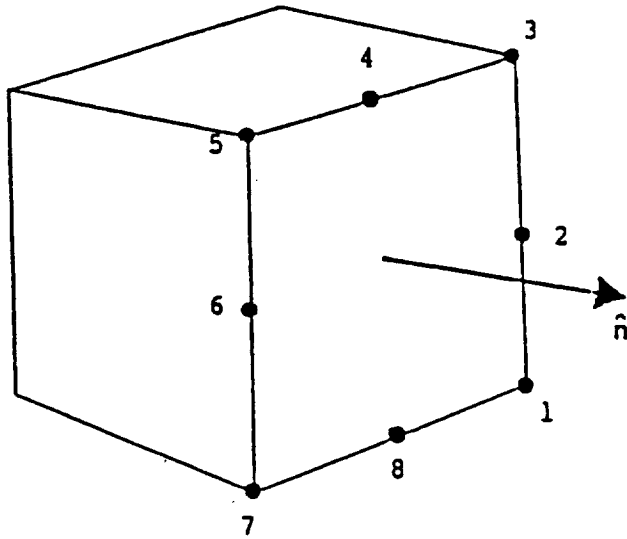
input shown is for
FLAG = +
on NORM card for 3-D
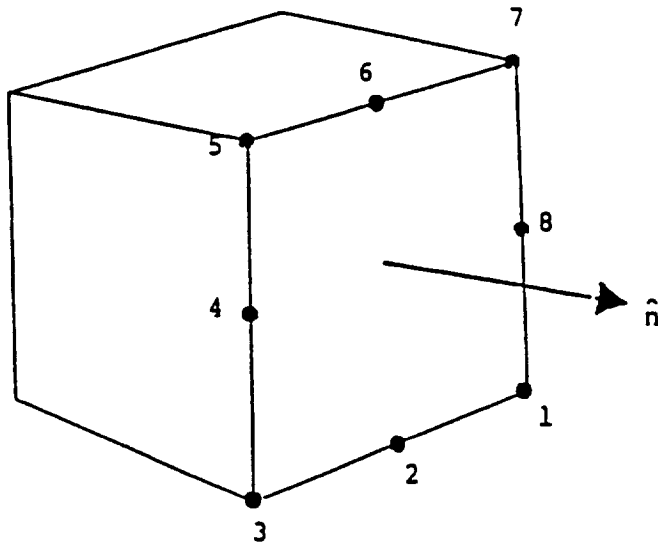
input shown is for
FLAG = -
on NORM card for 3-D

Figure for **GMR: NORM card

Three-dimensional outer normal convention
(In 3-D a mixed convention is allowed, as long as one
and only one normal per disjoint section is defined)

5.3.5	INSERT ELEMENT DEFINITION

INSE   EVAL   COND   ALPH   POIS

Status - OPTIONAL

Full Keyword - **INSERT**

Function - This card initiates the definition of insert (fiber) inclusions within the current GMR and defines the Modulus, the conductivity, and the thermal coefficient of the Insert.

Input Variables -

  EVAL   (Real)   - REQUIRED

  Elastic Modulus of the Insert

  COND   (Real)   - Optional (required if ALPH or POIS is input)

  Conductivity of the insert

  ALPH   (Real)   - Optional (required if POIS is input)

  Coefficient of thermal expansion of the insert

  POIS (Real) - Optional

  Poisson's ratio of the insert

Additional Information -

  If 'POIS' is not input, the Poisson's ratio of the insert is assumed to be the same as that of the GMR.

  If 'POIS' is input, then the exact insert approach is automatically selected (see EXACT card in **CASE input).

  In the present implementation, an insert is defined by a centerline and a radius. The centerline is represented by a two or three-noded line element. The three-noded element may be curved in space. The radius is constant along the length of the element.

  In a heat conduction analysis the value for EVAL is not used, however, a dummy value must be input.

Examples of Use -

  1. Define two inserts, one containing a single quadratic insert element, the other containing two quadratic insert elements.

```
INSERT   35.0E+06   10.0E+04   1.5E-06   0.3
POINTS
 1001   0.25   0.25   0.0
 1002   0.25   0.25   0.25
 1003   0.25   0.25   0.5
 1004   0.25   0.25   0.75
 1005   0.25   0.25   1.0
 2001   0.75   0.75   0.0
 2002   0.75   0.75   0.5
 2003   0.75   0.75   1.0
TYPE   QUAD
 ELEMENT
 101    0.1   1001   1002   1003
 102    0.1   1003   1004   1005
 ELEMENT
 201    0.1   2001   2002   2003
$(end of insert input)
```

INTE   ATYPE

Status - OPTIONAL

Full Keyword - INTERFACE

Function - Defines the type of interface between the insert and the GMR; defines the constitutive relationship used to model the interface.

Input Variables -

ATYPE   (Alphanumeric) - REQUIRED

Allowable values for ATYPE are:

BOND - bonded interface (linear condition)
LINE - linear initerface model (spring and/or thermal resistance)
SLID - sliding interface connection
NONL - specifies a nonlinear Spring-Coulumb friction model (see figure). This model cannot withstand tension. If the interface is subjected to tension, a gap appears and the tractions on the interface are zero.

Additional Information -

If the INTE card is not included, then BOND is the default.

if ATYPE = BOND then no constitutive properties should be input.

If ATYPE = SLID then no constitutive properties should be input.

If ATYPE = LINE then SPRING and/or RESI card, should be included.

if ATYPE = NONL then SPRING and FRIC cards should be included. RESI may be included for CTHERM analysis.

ATYPE = NONL is available for ELAS and CTHERM analyses only.

if ATYPE = NONL, an iterative nonlinear algorithm is used to solve the nonlinear system.

Examples of Use -

1. Define a frictional interface between insert and the GMR.

```
INSERT 10.0E+06   5.0E+04   2.0E-06
   INTERFACE NONL
      SPRI    10.0E+06    10.0E+05
      RESI    1.0E-03
      FRIC    0.3
```
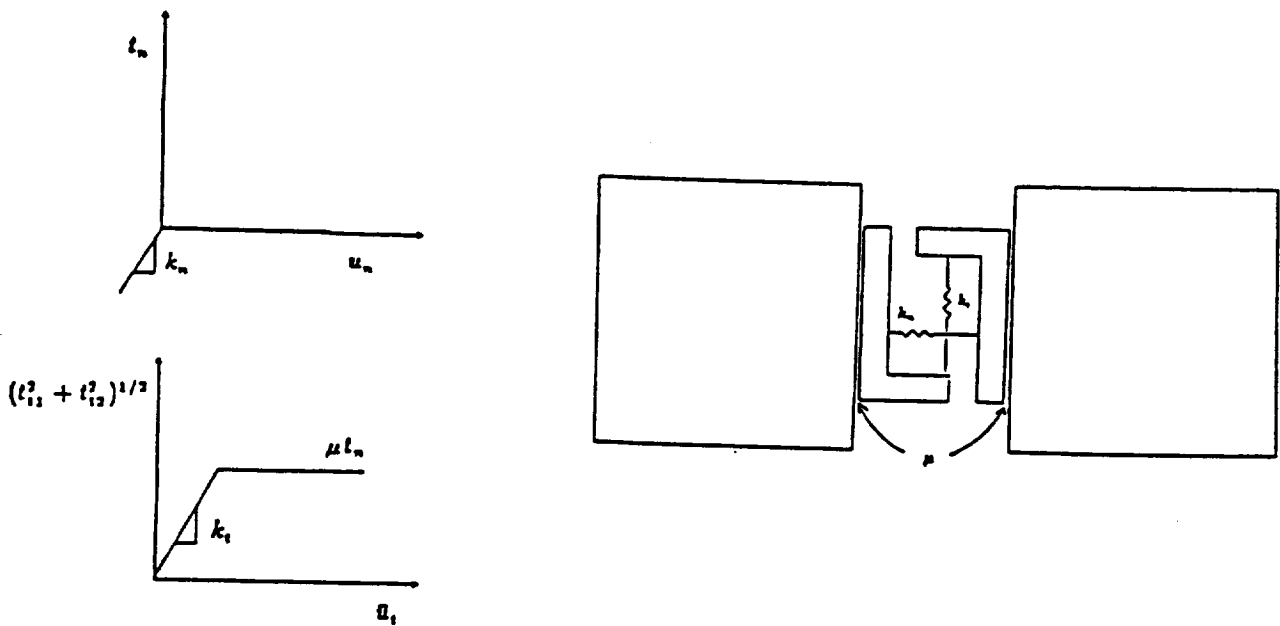
Nonlinear Spring/Coulomb-frictional Interface Connection

Figure for **GMR: INTE card

SPRI KN KT

Status - OPTIONAL

Full Keyword - SPRING

Function - Defines the spring coefficients between the Insert and GMR.

Input Variables -

> KN   (Real)   - REQUIRED
>
>> Spring coefficient normal to the insert
>
> KT   (Real)   - REQUIRED
>
>> Spring coefficient tangent to the insert

Additional Information -

> The Spring option utilizes the relationship

$$t_i = K_{ij}(d_j^I - d_j)$$

where

$$K_{ij} = \begin{bmatrix} K_n & 0 & 0 \\ 0 & K_t & 0 \\ 0 & 0 & K_t \end{bmatrix}$$

> d_j^I is the local displacement of the insert
>
> d_j is the local displacement of the GMR
>
> t_i is the local traction of the GMR

The user is responsible for providing k_n and k_t in proper units, consistent with the specifications of material properties, geometry, and boundary conditions.

If k_t is not specified then $k_t = k_n$ is assumed. The spring coefficient k_n and k_t should be a positive real number ($k > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -

> 1. Defines a linear spring interface relation between the insert and the GMR.

```
INSERT   10.0E+06
  INTERFACE   LINEAR
  SPRING  10.0E+06  10.0E+05
       .
       .
       .
       .
       .
       .
```

RESI   R1

Status - OPTIONAL

Full Keyword - RESISTANCE

Function - Identifies an interface with thermal resistance between the insert and the GMR. The flux across this interface is linearly related to the temperature difference between these two surfaces.

Input Variables -

R1   (Real)   - REQUIRED
Thermal resistance coefficient (R)

Additional Information -

The RESIstance option utilizes the relationship:

$$q_1 = \frac{1}{R1}(\theta_1 - \theta_2)$$

where

θ_1	local temperature of insert.
θ_2	local temperature of GMR.
q_1	local heat flux from insert.

The user is responsible for providing R in the proper units, consistent with the specification of material properties, geometry and boundary conditions.

The resistance R should be a positive real number $(R > 0)$. If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -

1. Defines thermal resistance at the insert/GMR interface.

```
INSERT 10.0E+06  5.0E+04  2.0E-06
  INTERFACE  LINEAR
  RESI   1.0E-03
  POINTS
    1    0.5   0.5 0.0
    .     .     .    .
    .     .     .    .
    .     .     .    .
    .     .     .    .
```

FRIC   FVAL

Status - OPTIONAL

Full Keyword - FRICTION

Function - Defines the coefficient of Coulumb's friction between the insert and the GMR.

Input Variables -

FVAL   (Real)   - REQUIRED
Coefficient of Coulumb's friction

Additional Information -

The value of FVAL should be greater than zero.

Examples of Use -

1. Define a frictional interface between the insert and the GMR.

```
INSERT 10.0E+06 5.0E+04  2.0E-06
 INTERFACE NONL
  SPRING  10.0E+06  10.0E+05
  RESI    1.0E-03
  FRIC    0.2
```

POIN

Status - REQUIRED (if INSE is input)

Full Keyword - POINTS

Function - This card initiates the definition of insert nodal points for the current GMR.

Input Variables - NONE

Additional Information -

Insert nodal points must be defined separately from the surface nodal points.

(NONE) NNODE X Y Z

Status - REQUIRED (if INSE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for insert nodes. (This format is used for inserts with constant circular cross-sections.)

Input Variables -

NNODE (Integer) - REQUIRED
User number for the insert nodes.

X,Y,Z (Real) - REQUIRED Cartesian coordinates of the of the insert nodes.

Additional Information -

This is one of two permissible formats that can be used for this card (see next page). This format is used for inserts with constant circular cross-sections.

DO NOT USE this card for inserts with elliptical cross-sections or inserts with cross-sections which vary in length. Instead use the format described on the next page. Also do not mix (in the same data set) the format of this page with the format described on the next page.

This card is input once for each point.

User node numbering must be unique. This includes surface, insert and interior points.

All point numbers must be less than or equal to 99999.

All insert nodes should be located on the interior of the part. Additionally, insert nodes should not be coincident with surface or interior points.

Examples of Use -

1. Define a set of node numbers and the cartesian coordinates for nodes on insert elements.

```
INSERT   30.0E+06
POINTS
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   1.0   1.0
    201   2.0   1.5   0.5
    202   2.0   1.5   0.5
    203   2.0   1.5   1.0
```

(NONE)   **NNODE X Y Z**   **RAD1 RAD2 VECX VECY VECZ**

Status - REQUIRED (if INSE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for insert nodes, and the cross-sectional properties of the insert at this node.

Input Variables -

    NNODE   (Integer) - REQUIRED

    User number for the insert nodes.

    RAD1 (Real) - REQUIRED

      - If RAD2 is not input, then RAD1 represents the radius of a circular cross-section (which may vary in length) at this node.

      - if RAD2 is input, then RAD1 represents the first radius of an elliptical cross-section at this node.

    RAD2 (Real) - OPTIONAL

      RAD2 is the second radius of an elliptic cross-section.

    VECX,VECY, VECZ (Real) OPTIONAL (required if RAD2 is input)

      Vector defining the orientation of the elliptic cross-section at this node. The vector obtained from the cross-product of this vector and the axis of the insert fiber at this node defines the direction of the second radius (RAD2). Therefore, this vector is the approximate direction of the first radius (RAD1).

Additional Information -

    This is one of two permissible formats that can be used for this card (see previous page). This format is intended for use with insert fibers with elliptical cross-sections or circular cross-sections that vary along the length of the fiber; however, inserts with constant circular cross-sections can also be modeled using this format.

    When this format is used, numerical integration of the insert fibers is automatically employed (see the NUMERICAL card under **CASE input). Numerical integration is more computationally expensive than semi-analytical integration which is normally used, therefore, this format should be reserved for fibers with elliptical and variable cross sections.

    The format of the coordinate card must be consistent throughout the data set. DO NOT MIX the format of this page and the format of the previous page in the same input data file.

    This format is not allowed with micro-surface insert fibers (see **GMR card).

    The radii (RADI, RAD2) and the vector (VECX, VECY, VECZ) defined on this card may vary from point to point; however, the variation should not be abrupt.

When the format of this page is used, the radius input on the connectivity card (see the insert connectivity card following the 'ELEM' card) is ignored. A 'dummy' input value; however, must be input on the connectivity card.

This card is input once for each point.

User node numbering must be unique. This includes surface, insert and interior points.

All point numbers must be less than or equal to 99999.

All insert nodes should be located on the interior of the part. Additionally, insert nodes should not be coincident with surface or interior points.

Examples of Use -

1. Define a set of node numbers and the cartesian coordinates for nodes on an insert fiber with a circular cross-section that varies along the length of the fiber. (Note the radius value (0.1) in the connectivity card is a dummy value which is ignored by the program when using this coordinate input format for fibers with variable cross-sections.

```
INSERT 30.0E+06
POINTS
      101   1.0   0.0    1.0   0.2
      102   1.0   0.25   1.0   0.25
      103   1.0   0.50   1.0   0.30
      104   1.0   0.75   1.0   0.35
      105   1.0   1.00   1.0   0.40
TYPE QUAD
   ELEMENT
      1000  0.1   101   102   103
      1001  0.1   103   104   105
```

2. Define a set of node numbers, cartesian coordinates, and cross-sectional properties for an elliptical fiber insert. The radii are 0.15 and 0.2 with the 0.15 radius oriented with the global y-direction (0.0, 1.0, 0.0). Note the radius value (0.1) on the connectivity card is a dummy value which is ignored by the program when using this coordinate input format for fibers with elliptic cross-sections.

```
INSERT 30.0E+06
POINTS
      101      0.0  1.0  1.0    0.15  0.2    0.0  1.0  0.0
      102      0.5  1.0  1.0    0.15  0.2    0.0  1.0  0.0
      103      1.1  1.0  1.0    0.15  0.2    0.0  1.0  0.0
      104      1.5  1.0  1.0    0.15  0.2    0.0  1.0  0.0
      105      2.0  1.0  1.0    0.15  0.2    0.0  1.0  0.0
TYPE QUAD
   ELEMENT
      1000  0.1  101   102   103
      1001  0.1  103   104   105
```

TYPE   ETYPE   FTYPE

Status - REQUIRED (if INSE is input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities in the longitudinal direction for three-noded insert elements.

Input Variables -

ETYPE   (Alphanumeric)   - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

FTYPE (Alphanumeric) - OPTIONAL (only for use with micro-surface elements with ETYPE = QUAD)

Allowable values for FTYPE are LAGR and SERE. The default is SERE.

SEREndipity - Serendipity generates (6 and 8-noded) microsurface elements.

LAGRangian - Lagrangian generates (6 and 9-noded) micorsurface elements.

Additional Information -

Only one TYPE card for inserts is allowed per GMR.

All of the insert elements in a single GMR will have the same type of variation defined on the TYPE card.

This card has no effect on the circumferential variation of the field variables about the standard (i.e., not a microsurface) insert.

Examples of Use -

1. Specify that all three-noded insert elements appearing in the element list for insert elements have a quadratic variation of field quantities in the longitudinal direction.

```
INSERT   30.03E+06
POINTS
     101   1.0   0.0   10
     102   1.0   0.25  1.0
     103   1.0   0.5   1.0
     104   1.0   1.0   1.0
     201   2.0   1.5   0.0
     202   2.0   1.5   0.5
     203   2.0   1.5   1.0
TYPE QUAD
```

ELEM

Status - REQUIRED (if INSE is input)

Full Keyword - **ELEMENTS**

Function - Signals the beginning of the connectivity definition for single insert with one or more insert elements.

Input Variables - NONE

Additional Information -

The ELEM card is repeated for each individual fiber insert. An individual insert may contain more than one insert element which are connected end to end, however, an unconnected set of elements represents a set of inserts, in which each unconnected element must be preceded by the ELEM card.

(NONE)   NEL RADIUS NODE1 ...   NODEN

Status - REQUIRED (if INSE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the radius and connectivity for a single insert element.

Input Variables -

> NEL   (Integer) - REQUIRED
> User insert element number.
>
> RADIUS   (Real)   - REQUIRED
> Radius of the insert.
>
> NODE1 ... NODEN   (Integer)   - REQUIRED
> User node numbers of the two or three insert nodes defining the geometry of the insert. NOTE: Two-noded insert elements are no longer supported.

Additional Information -

> This card is input once for each insert element.
>
> User element numbers, including both surface and insert elements, must be unique within the problem.
>
> All insert element numbers must be less than or equal to 99999.
>
> All of the nodes referenced in the insert element connectivity must have been defined previously as POINTs under the INSE keyword. That is, insert elements can only connect insert nodes. No surface nodes can be referenced by a insert element.
>
> The radius is assumed constant along the entire length of the insert element.
>
> If a fiber is input with an elliptical cross-section or a cross-section that varies along the length of the fiber (see the two insert coordinate cards following the insert 'POINT' card of this section), than the radius value input on this card is ignored. A 'dummy' input value, however, must be input on this card.

Examples of Use -

> 1. Define a set of data for three insert elements on two inserts specifying the element number, the radius and the connectivity. One insert contains one three-noded quadratic element and one two-noded linear element. The second insert contains one three-noded quadratic element.

```
INSERT   30.0E+06
POINTS
```

```
    101   1.0    0.0   1.0
    102   1.0    0.25  1.0
    103   1.0    0.5   1.0
    104   1.0    1.0   1.0
    105   1.0    1.5   1.0
    201   2.0    1.5   0.0
    202   2.0    1.5   0.5
    203   2.0    1.5   1.0
TYPE QUAD
  ELEMENT
    1001  0.2    101   102   103
    1002  0.2    103   104   105
  ELEMENT
    2001  0.2    201   202   203
```

5.3.6	SAMPLING POINT DEFINITION

SAMP   POIN

Status - OPTIONAL

Full Keyword - SAMPLING-POINTS

Function - This card signals the fact that a set of sampling points for which results are requested at any point on or in the body, will be provided for the current GMR.

Input Variables -

POIN   (Alphanumeric)   - REQUIRED

Additional Information -

This card is used to define points at which displacements, stresses, strains, temperatures, pressures and fluxes are to be calculated.

This card is followed by data cards defining the node number and coordinates of the sampling points.

The sampling point definition is also used in conjunction with the sampling surface definition. See next subsection.

The sampling point definition has no effect on the solution of the boundary element system.

Examples of Use -

1. Request result information at three interior points

```
SAMPLING-POINTS
 1001    0.333   0.25   0.0
 1002    0.25    0.1   10.0
 1003    0.2     0.5   20.2
```

(NONE)   NNODE X Y Z

Status - REQUIRED (if SAMP is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines the coordinates of the sampling points for which output will be reported.

Input Variables -

NNODE   (Integer)   - REQUIRED
User number for the node.

X,Y,Z   (Real)   - REQUIRED Cartesian coordinates of the nodal point.

Additional Information -

This card is input once for each point.

User nodal point numbers must be unique, including the surface nodal points and any additional nodal points created for the discretization of insert elements.

Point numbers must be less than or equal to 99999.

5.3.7	SAMPLING SURFACE DEFINITION

SAMP   SURF   NAME

Status - OPTIONAL

Full Keyword - SURFACE

Function - This card initiates the definition of a sampling surface for the current GMR (use in post-processing or results).

Input Variables -

SURF   (Alphanumeric) - REQUIRED

This distinguishes the SAMP SURF definition from the SAMP POINT definition of the last sub-section.

NAME (Alphanumeric) REQUIRED

This is a user defined name for the current sampling surface. May be up to 8-characters long.

Additional Information -

The name of each sampling surface in a GMR must be unique.

The sampling surface is only used for the post-processing display of results. It has no effect on the solution of the boundary element system.

There may be more than one sampling surface in a GMR. The number of sampling surfaces in a GMR is only limited by the total number of sampling points allowed in the total analysis which is set in the PARAM file before compiling BEST-CMS. Use the PRINT LIMIT card in **CASE INPUT to see the current compiled limit.

Examples of Use -

```
SAMPLING POINTS
    1001    1.0    2.0    3.0
    1002    1.0    3.0    3.0
    .
    .

SAMPLING SURFACE SS-TOP
    ELEM
    101    1001    1002    1003    1004    1005    1006    1007    1008
    102    1001    1002    1003    1009    1010    1011    1012    1013
    .
    .

SAMPLING SURFACE SS-BACK
    ELEM
    202    2001    2002    2003    2004    2005    2006
    .
    .
```

ELEM

    Status - REQUIRED (if SAMP SURF is input)

    Full Keyword - ELEMENTS

    Function - Signals the beginning of the connectivity definition for sampling surface elements of the current surface.

    Input Variables - NONE

    Additional Information - NONE

(NONE)   NEL   NODE1   ...   NODEM

Status - REQUIRED (Minimum of one SAMP SURF is input)

Full Keyword - **NO KEYWORD REQUIRED**

Function - Each card defines the connectivity of a single sampling surface element.

Input Variables -

NODE1 ... NODEM (Integer)   REQUIRED

User element number

User node numbers may be 3, 4, 6, 8 or 9 for 3D analysis. the elements should be compatible with the graphic program that will be used in the post-processing of results.

Additional Information -

- This card is input once for each element.

- All nodes specified n the connectivity must be input under the SAMP POINT definition.

- User element numbers should be less than or equal to 99999. The element number does not have to be unique, but it is recommended.

- The sequence of the node numbering is similar to a regular boundary element. However, since the sampling surface element's only purpose is for displaying results in a post-processor, the direction of the order (with respect to the normal) is not important.

Examples of Use -

```
SAMPLING POINTS
    1001   1.0    2.0    3.0
    1002   1.0    3.0    3.0
    :

SAMPLING SURFACE SS-TOP
    ELEM
    101    1001   1002   1003   1004   1005   1006   1007   1008
    102    1001   1002   1003   1009   1010   1011   1012   1013
    :

SAMPLING SURFACE SS-BACK
    ELEM
    202    2001   2002   2003   2004   2005   2006
    :
```

5.4 | DEFINITION OF GMR COMPATIBILITY

When a body is modelled as an assembly of several GMRs suitable conditions must be specified to define the connections among the various regions. In the present version of **BEST-CMS** compatibility is defined between the interface surfaces of each pair of contacting regions. Five types of compatibility are allowed:

1 - Bonded contact : Continuity of all displacement components is imposed across the interface.

2 - Sliding contact : Continuity is required only for the component of displacement normal to the interface. The tractions, in both GMRs, in the tangent plane to the interface are set to zero.

3 - Nonlinear contact: the interface is assumed to be a nonlinear interface defined by a nonlinear interface model.

4 - Spring or resistance contact : Spring or thermal resistance is imposed between regions.

5 - Cyclic contact : Symmetric elements within a cyclic symmetric part can have imposed symmetric deformation on these elements.

Continuity of temperature or pressure, when applicable, is imposed across the interface in a similar manner.

A single nodal point location may be part of at most two GMRs. A single nodal point may be referenced in more than one interface definition data set as long as only two GMRs are involved. A single location must have a unique node number in each GMR. Various acceptable and unacceptable arrangements of GMRs are illustrated in the figure following the **INTE card.

The interface compatibility must be specified in such a way that there is one to one correspondence between the source points (field variable nodes) of the two GMR's that are involved. The input required to specify a single interface between two GMRs is described in the following pages, and a list of keywords recognized in the interface input are given below.

<u>**SECTION**</u>	<u>**KEYWORD**</u>	<u>**PURPOSE**</u>

5.4.1 Interface Definition Input Card
        INTE   Start of interface compatibility condition

5.4.2 Definition of interface surface 1
        GMR   name of first GMR
        SURF   surface on first GMR
        ELEM   element of surface

5.4.3 Definition of interface surface 2
        GMR   name of second GMR
        SURF   surface on second GMR
        ELEM   element of surface

5.4.4 Type of interface conditions
        BOND   bonded interface connection
        SLID   sliding interface connection
        NONL   Nonlinear interface connection
        SPRI   spring constant coefficients
        RESI   thermal resistence across interface
        FRIC   coefficient of Coulomb's friction

5.4.5 Cyclic Symmetry interface definition
        CYCL   cyclic symmetry interface definition
        ANGL   angle for cyclic interface
        DIR   axis of rotation for cyclic interface

5.4.6 Additional Interface Control Options
        TDIF   initial temperature difference across
                                   interface

5.4.1	INTERFACE DEFINITION INPUT CARD

**INTE

Status - OPTIONAL

Full Keyword - **INTERFACE**

Function - Indicates the beginning of an interface definition.

Input Variables - NONE

Additional Information -

A **INTE card must begin each interface definition. The complete definition of the connection between two GMRs may require more than one data set, since each data set can refer to only one surface.

The data set initiated with this card may be repeated as many times as required.

The interface surface reference below must be such that the nodes and elements of one GMR can be superimposed on the nodes and elements of the other GMR by translation and/or rotation, without any deformation.

Note that each of the two GMR's involved in the interface definition must contain elements that lie on the interfacial surface.

The interface data sets must follow all GMR definitions, and must precede any boundary condition data sets.

Examples of Use -

1. Defines the interface of two GMR's (default is perfectly bonded connection).

```
**INTERFACE
  GMR   REG1
        SURFACE   TOP
        ELEMENT   3   4   5
  GMR   REG2
        SURFACE   BOTTOM
        ELEMENT   103   104   105
```
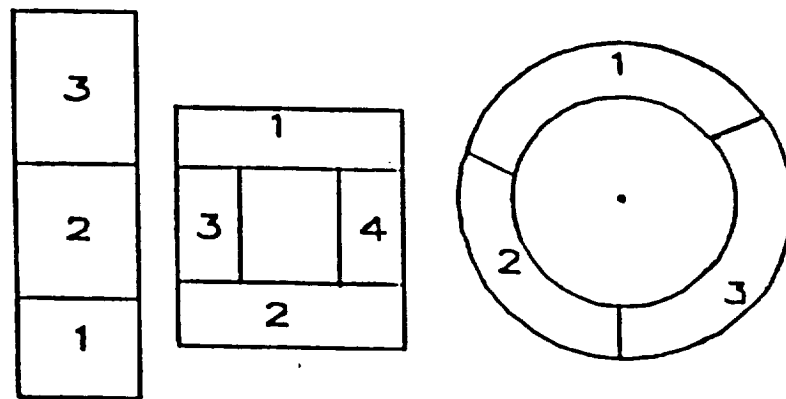
Acceptable Connections

Unacceptable Connection

Figure for **INTE : card   Connections among GMRs
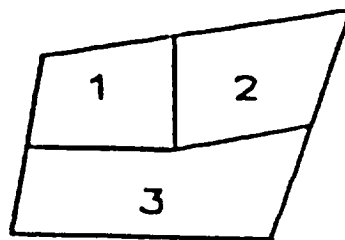
5.4.2	DEFINITION OF INTERFACE SURFACE 1

GMR   IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the first GMR for which the interface surface is to be defined.

Input Variables -

IDGMR   (Alphanumeric)   - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given interface surface must lie entirely on the surface of a single GMR. If an interface compatibility condition is to be applied with more than one GMR, a separate interface compatibility must be defined for each case.

Examples of Use -

1. Identifies the first GMR, say GMR1, of which the interface surface is a part.

```
**INTERFACE
  GMR GMR1
    SURFACE   SURF1
      ELEMENTS 101    102    103    104
```

SURF   IDSUR

    Status - REQUIRED

    Full Keyword - SURFACE

    Function - Identifies the surface within the (first) selected GMR which embodies the interface surface (NAME on SURF card in **GMR input).

    Input Variables -

        IDSUR   (Alphanumeric)   - REQUIRED

    Additional Information -

        An interface surface must be contained entirely within a single surface. If the interface compatibility condition is to be applied to more than one surface, then a separate interface compatibility must be defined for each surface involved.

        The SURF card may conclude the required input for a interface definition. If the SURF card is not followed by a ELEM card, then BEST-CMS will apply the interface compatibility condition to all of the elements in the surface IDSUR.

    Examples of Use -

        1. Identifies the interface surface, say SURF1, as part of the first GMR.

```
**INTERFACE
  GMR GMR1
    SURFACE   SURF1
    ELEMENTS 109    110
  GMR GMR2
    SURFACE   SURF2
```

ELEM   EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the elements of the surface IDSUR to which an interface compatibility condition is to be applied.

Input Variables -

> EL1,EL2,...,ELN   (Integer)   - REQUIRED
>
> User element numbers of the elements of surface IDSUR which forms the interface surface.

Additional Information -

> The effect of this card is to restrict the application of the compatibility condition to a portion of the surface IDSUR.
>
> This input may be continued on more than one card. Each card must begin with the keyword ELEM.
>
> **If the ELEM card is specified, BEST-CMS** will apply the interface compatibility condition only to the elements specified on this list.
>
> In the present version of **BEST-CMS** , interface compatibility can not be specified at individual nodes.

Examples of Use -

> 1. Specifies three elements, 120, 121 and 122, for interfacial compatibility on the surface identified by the preceding SURFACE card.

```
**INTERFACE
  GMR   GMR1
    SURFACE   SURF1
    ELEMENTS 120    121    122
  GMR GMR2
```

| 5.4.3 | DEFINITION OF INTERFACE SURFACE 2 |

GMR   IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the second GMR for which the interface surface is to be defined.

Input Variables -

IDGMR   (Alphanumeric)   - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given interface surface must lie entirely on the surface of a single GMR. If an interface compatibility condition is to be applied with more than one GMR, a separate interface compatibility must be defined for each case.

Examples of Use -

1. Identifies the second GMR, say GMR2, of which the GMR surface is a part.

```
**INTERFACE
  GMR GMR1
    SURFACE   SURF1
    ELEMENTS 101    102
  GMR GMR2
    SURFACE   SURF2
    ELEMENTS 201    202
```

SURF   IDSUR

Status - REQUIRED

Full Keyword - SURFACE

Function - Identifies the surface within the (second) selected GMR which embodies the interface surface (NAME on SURF card in **GMR input).

Input Variables -

   IDSUR   (Alphanumeric)   - REQUIRED

Additional Information -

An interface surface must be contained entirely within a single surface. If the interface compatibility condition is to be applied to more than one surface, then a separate interface compatibility must be defined for each surface involved.

The SURF card may conclude the required input for a interface definition. If the SURF card is not followed by a ELEM card, then BEST-CMS will apply the interface compatibility condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the interface, say SURF2, as part of the second GMR.

```
**INTERFACE
  GMR GMR1
     SURFACE   SURF1
     ELEMENTS 101    102
  GMR GMR2
     SURFACE   SURF2
     ELEMENTS 201    202
```

ELEM   EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - **ELEMENTS**

Function - Specifies the elements of the surface IDSUR to which an interface compatibility condition is to be applied.

Input Variables -

      EL1,EL2,...,ELN   (Integer)   - REQUIRED

      User element numbers of the elements of surface IDSUR which forms the interface surface.

Additional Information -

      The effect of this card is to restrict the application of the compatibility condition to a portion of the surface IDSUR.

      This input may be continued on more than one card. Each card must begin with the keyword ELEM.

      If the ELEM card is specified, BEST-CMS will apply the interface compatibility condition only to the elements specified on this list.

      In the present version of **BEST-CMS** , interface compatibility can not be specified at individual nodes.

Examples of Use -

    1. Specifies three elements, 210, 211 and 212, for interfacial compatibility on the surface identified by the preceding SURFACE card.

```
**INTERFACE
  GMR   GMR1
    SURFACE   SURF1
    ELEMENTS 101    102    103
  GMR   GMR2
    SURFACE SURF2
    ELEMENTS 210    211    212
```

5.4.4	TYPE OF INTERFACE CONDITION

BOND

Status - OPTIONAL

Full Keyword - BONDED

Function - Identifies a fully bonded interface.

Input Variables - NONE

Additional Information -

When this card is input continuity of all variables is imposed across the interface.

This is the default condition when the type of interface is not explicitly defined.

Examples of Use -

1. Defines a perfectly bonded interface of three boundary elements.

```
**INTERFACE
  BOND
  GMR   REG1
    ELEMENT   3    4    5
  GMR   REG2
    ELEMENT  103  104  105
```

SLID

Status - OPTIONAL

Full Keyword - SLIDING

Function - Identifies a sliding interface.

Input Variables - NONE

Additional Information -

When this card is input only normal displacement compatibility is imposed across the interface. The two GMRs are free to move in the plane tangent to the interface, however, the surfaces remain in contact even under tension. This freedom may require the specification of additional boundary conditions to restrain rigid body motion.

Examples of Use -

1. Defines a sliding interface of five boundary elements.

```
**INTERFACE
  SLID
  GMR   REG1
     ELEMENT   101   102   103   104   105
  GMR REG2
     ELEMENT   210   212   213   214   215
```

NONL   ATYPE

Status - OPTIONAL

Full Keyword - **NONLINEAR**

Function - Identifies a nonlinear interface between corresponding surfaces and defines the constitutive relationship used to model the interface.

Input Variables -

ATYPE   (Alphanumeric)   - REQUIRED
Allowable values are FRIC.

FRIC - Specifies a spring-Coulumb friction model (See figure). This model cannot withstand tension. If the interface is subjected to tension, a gap appears and the tractions on the interface are zero.

Additional Information -

Available for ELAS and CTHER analyses only.

Required constitutive property cards are SPRING and FRIC. RESI may be included for CTHERM analysis.

If this card is included, an iterative nonlinear algorithm is used to solve the nonlinear system.

Examples of Use -

1. An nonlinear interface for CTHE analysis

```
**INTERFACE
  GMR   REG1
       SURFACE   TOP
       ELEMENT   101   102
  GMR   REG2
       SURFACE   BOT
       ELEMENT   201   202
  NONLINEAR FRIC
       SPRING       10.0E+08    10.0E+08
       FRIC         0.2
       RESI         1.0E-04
```
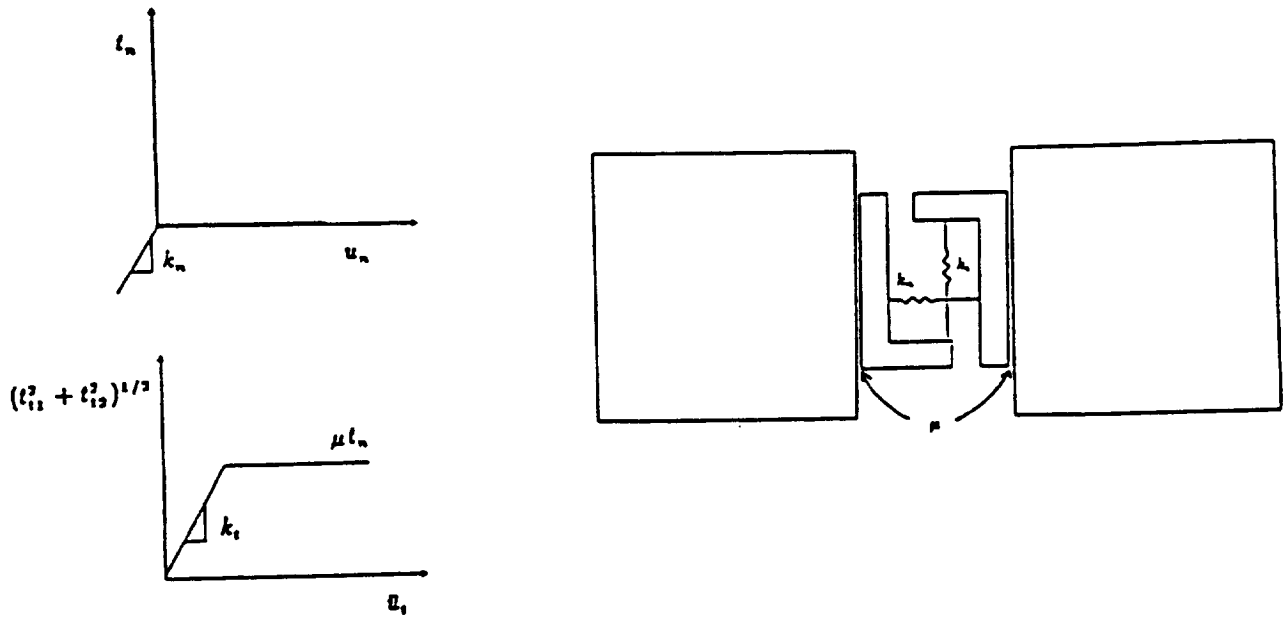
Nonlinear Spring/Coulomb-frictional Interface Connection

Figure for **INTE : NONL card

SPRI   KN KT

Status - OPTIONAL

Full Keyword - SPRING

Function - Defines the spring coefficients on an interface between two regions. The tractions across the interface are linearly related to the difference in displacement between the two surfaces.

Input Variables -

KN   (Real)   - REQUIRED

Spring coefficient normal to the insert

KT   (Real)   - REQUIRED

Spring coefficient tangent to the insert

Additional Information -

The SPRIng option utilizes the relationship

$$t_i = K_{ij}(d_i^I - d_j^{II})$$

where

$$K_{ij} = \begin{bmatrix} K_n & 0 & 0 \\ 0 & K_t & 0 \\ 0 & 0 & K_t \end{bmatrix}$$

d_j^I is the local displacement of the GMR 1

d_j^{II} is the local displacement of the GMR 2

t_i is the local traction between regions

The user is responsible for providing k_n and k_t in proper units, consistent with the specifications of material properties, geometry, and boundary conditions.

If k_t is not specified then $k_t = k_n$ is assumed. The spring coefficient k_n and k_t should be a positive real number ($k > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

May be used in conjunction with the NONL card or without.

Examples of Use -

1. Defines an interface between two regions connected by springs.

```
**INTERFACE
  GMR REG1
    SURFACE SURF1
    ELEMENT  101  102
  GMR REG2
    SURFACE SURF2
    ELEMENT  201  202
      SPRING  10.0E+06  10.0E+05
```

RESI   R1

Status - OPTIONAL

Full Keyword - RESISTANCE

Function - Identifies an interface with thermal resistance between the corresponding surfaces. The flux across this interface is linearly related to the temperature difference between the two surfaces.

Input Variables -

R1   (Real)   - REQUIRED

Thermal resistance coefficient (R)

Additional Information -

The RESIstance option utilizes the relationship:

$$q_1 = \frac{1}{R1}(\theta_1 - \theta_2)$$

where

θ_1	local temperature of GMR 1.
θ_2	local temperature of GMR 2.
q_1	local heat flux from GMR 1.

The user is responsible for providing R in the proper units, consistent with the specification of material properties, geometry and boundary conditions.

The resistance R should be a positive real number ($R > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

May be used in conjunction with the NONL card or without.

Examples of Use -

1. Defines thermal resistance at the interface between two regions which were at the same initial temperature (otherwise, a TDIF card should be inserted after the RESI card).

```
**INTERFACE
   GMR GMR1
      SURFACE  SURF1
      ELEMENTS 12
   GMR GMR2
      SURFACE  SURF2
      ELEMENTS 21
   RESI 1.0
```

FRIC FVAL

Status - OPTIONAL

Full Keyword - FRICTION

Function - Defines the coefficient of Coulumb's friction between the corresponding surfaces.

Input Variables -

FVAL (Real) - REQUIRED

Coefficient at Coulumb's friction

Additional Information -

The value of FVAL should be greater than zero.

This card can only be used in conjunction with the NONL card.

Examples of Use -

1. Defines a frictional interface between two regions

```
**INTERFACE
  GMR GMR1
    SURFACE   SURF1
    ELEMENTS 101 102
  GMR GMR2
    SURFACE   SURF2
    ELEMENTS 201 202
  NONLINEAR FRIC
    SPRING   10.0E+06   10.0E+05
    RESI     2.0E-03
    FRIC     0.2
```

5.4.5	CYCLIC SYMMETRY PARAMETER DEFINITION

CYCL

Status - OPTIONAL

Full Keyword - CYCLIC

Function - Identifies a cyclic symmetry boundary condition.

Input Variables - NONE

Additional Information -

This type of interface condition establishes a relationship between two boundary surfaces. In order for this condition to be applied the two boundary surfaces involved must be such that one can be exactly superimposed on the other by a rotation about a specified axis passing through the origin of the global coordinate system. Further, the imposed boundary conditions of the problem must be such that the deformed shape of one boundary surface can be exactly superimposed on the other by the same rotation. This option is intended for the analysis of (periodic) structures subjected to periodic loading.

Rigid body translation along the cyclic axis and rigid body rotation about that same axis are **not** automatically prevented by invoking the CYCLIC option. Consequently, these motions must be constrained explicitly by the user.

Since a cyclic interface condition involves all components of displacement and traction, **no other boundary condition may be applied to the elements that are involved.**

Local coordinate systems are established for each node on the second boundary surface. As a result, no other local system may be defined for these nodes. Furthermore, in the current version, it is recommended that displacement boundary conditions **not be applied** to any of the second surface nodes.

In the present version of **BEST-CMS**, a boundary surface to which a cyclic interface is applied may not intersect another interface.

A cyclic interface condition is time independent.

Examples of Use -

1. Activate option for cyclic symmetry boundary condition.

```
**INTERFACE
  GMR  GMR1
```

```
SURFACE SURF1
ELEMENT 3
GMR GMR1
SURFACE SURF1
ELEMENT 5
CYCLIC
ANGLE 20
DIRECTION  0.  0.  1.
```

ANGL   THETA

Status - REQUIRED (if CYCL is specified)

Full Keyword - **ANGLE**

Function - Specifies the angle of rotation between the two surfaces referenced in the cyclic symmetry condition.

Input Variables -

THETA   (Real)   - REQUIRED

THETA is the rotation angle (in degrees). A positive rotation is counterclockwise when looking along the positive axis direction.

Additional Information - NONE

Examples of Use -

1. Specifies an angle of 20 degrees between the two surfaces referenced in the cyclic symmetry condition.

```
**INTERFACE
  GMR GMR1
  SURFACE SURF1
  ELEMENT 3
  GMR GMR1
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION  0.  0.  1.
```

DIRE   X Y Z

Status - OPTIONAL

Full Keyword - DIRECTION

Function - Defines the positive direction of the axis of rotation, if CYCL is specified.

Input Variables -

X,Y,Z   (Real)   - REQUIRED

Components of a vector along the positive direction of the axis of rotation.

Additional Information -

This card may be omitted. In this case the rotation axis defaults to the positive z-axis.

Examples of Use -

1. Defines that the positive direction of the axis of rotation is along the z-axis.

```
**INTERFACE
  GMR GMR1
  SURFACE SURF1
  ELEMENT 3
  GMR GMR1
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION  0.  0.  1.
```

5.4.6	ADDITIONAL INTERFACE CONTROL OPTIONS

TDIF

Status - OPTIONAL

Full Keyword - TDIFFERENECE

Function - Signals that there is a difference in the initial temperatures of the two regions involved in the current interface.

Input Variables - NONE

Additional Information -

The TDIF card must be included in the interface definition for any temperature-dependent problem for which the initial temperatures of the adjoining regions are different. In such situations, failure to include this card will produce incorrect results.

It is expected that in future releases of **BEST-CMS**, the necessary checks will be done automatically, and the TDIF card will no longer be needed.

Examples of Use -

1. Indicates that a difference in initial temperatures exists between the GMR's, REG1 and REG2, involved in the current interface.

```
**INTERFACE
  GMR  REG1
     SURFACE   TOP
     ELEMENTS 101    102
  GMR  REG2
     SURFACE   BOTTOM
     ELEMENTS 209    210
  RESI   1.0
  TDIF
```

This section describes the boundary condition input set (BCSET) for the input of boundary conditions applied at the surface of the given structure (or body). The input is designed to allow the specification of time dependent boundary conditions in both local and global coordinate systems. In order to allow the generality required, the input system is necessarily somewhat complex. Considerable simplification is possible for problems with less general requirements.

In the boundary element method, the primary load variable is traction (or flux), which acts over a surface area, not point forces (or sources) as in the finite element method. This means that in defining the region of application of a boundary condition in **BEST-CMS** it is necessary to specify both the nodal points and the elements involved.

A variety of options are provided for the definition of boundary conditions on the surface of the part. Each distinct set of boundary condition data defines either numerical values of variables over some portion of the surface of the part (or body), or establishes a relationship among variables. As many sets of boundary condition data may be used, as are required to completely specify the problem. A nodal point or element may be referenced in more than one set of boundary condition data.

A common process to much of the boundary condition input is the specification of the time dependent variables over the surface. To simplify the subsequent discussion of the various boundary condition types, the recurring definition of space/time variation is described only once in Section 5.5.6.

SECTION   **KEYWORD**   **PURPOSE**

5.5.1 Boundary Condition Set Card
            **BCSET   start of the B.C. definition

5.5.2 Boundary Condition Identification
            ID   name of B.C. set

5.5.3 Identification of Boundary Condition Type
            VALU   for specified B.C. value input
            RELA   for B.C. relation between boundary quantities
            LOCA   for local definition of B.C.

SECTION	**KEYWORD**	**PURPOSE**

5.5.4 Definition of Surface for Application of Boundary Conditions

	GMR	identifies a GMR
	SURF	identifies the surface for this B.C. set
	ELEM	identifies surface elements
	POIN	identifies surface points
	TIME	defines the time for input

5.5.5 Value Boundary Condition for Surface Elements

	TRAC	traction B.C. input
	DISP	displacement B.C. input
	FLUX	flux B.C. input
	TEMP	temp B.C. input
	RIGI	rigid B.C. input
	FORC	force input for rigid B.C.

5.5.6 Definition of Space/Time variation

	SPLI	source (field variable) point list
	T	nodal value of B.C.

5.5.7 Relation Boundary Condition

	SPRI	spring relation (between displacement and traction)
	CONV	convection relation (between temperature and flux)

5.5.1	BOUNDARY CONDITION SET CARD

∗∗BCSE

Status - REQUIRED

Full Keyword - ∗∗BCSET

Function - Identifies the beginning of a boundary condition data set.

Input Variables - NONE

Additional Information -

As many boundary condition data sets may be input as are required. Each must begin with this card.

The boundary condition data sets must follow all GMR and INTERFACE definitions.

Examples of Use -

1. Fix the normal (local) displacement of all elements for on surface SIDE1 of gmr REG2 for all time (no TIME card required)

```
**BCSET
  ID U1FIX
  VALUE
  LOCAL
  GMR REG2
    SURFACE SIDE1
    DISP 1
    SPLIST ALL
    T 1 0.0
```

5.5.2	BOUNDARY CONDITION IDENTIFICATION

ID   NAME

Status - REQUIRED

Full Keyword - **ID**

Function - Defines the identifier for the current boundary condition data set.

Input Variables -

> NAME   (Alphanumeric)   - REQUIRED
> User specified name of for the current data set.

Additional Information -

> The NAME must be unique compared to all other boundary condition data set names defined in the problem.

> The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

Examples of Use -

> 1. Define a set of displacement type boundary conditions with the name DISP1.

```
**BCSET
  ID  DISP1
  SURFACE   SURF1
  ELEMENTS   104
  POINT 108
  DISP 1
  SPLIST 108
  T    1    0.0
**BCSET
```

5.5.3	IDENTIFICATION OF BOUNDARY CONDITION TYPE

VALU

Status - OPTIONAL

Full Keyword - **VALUE**

Function - Identifies the boundary condition as one which will define the numerical values of field variables.

Input Variables - NONE

Additional Information -

> This card must not be used for relational boundary condition sets.

> If VALU, RELA, or VARI do not appear in a boundary condition set, a value-type set is assumed.

Examples of Use -

> 1. Used here to indicate that the value of a local traction type of boundary condition is specified.

```
**BCSET
  ID  TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TRAC 1
  SPLIST ALL
    T 1   -100.0
$end of input data
```

RELA

Status - OPTIONAL

Full Keyword - RELATION

Function - Identifies the boundary condition as one which will define a relationship between field variables (e.g. spring or convection boundary conditions).

Input Variables - NONE

Additional Information -

This card is required for all boundary condition sets which define a relationship between field variables. Therefore, this card must be included for SPRING or CONVECTION boundary conditions.

Examples of Use -

1. The RELATION card is used in the following example to indicate specification of convection type of boundary condition.

```
**BCSET
  ID   BCS1
  RELATION
  GMR   GMR1
  SURFACE   SURF1
    ELEMENTS    1    2    3    4
    CONV  1.26    -100.0  $  H = 1.26, TEMP (AMBIENT) = - 100.0
**BCSET
  ID   BCS2
```

LOCA

Status - OPTIONAL

Full Keyword - LOCAL

Function - Indicates that input for the current boundary condition set will be in local coordinates.

Input Variables - NONE

Additional Information -

In the present version of **BEST-CMS** this option is intended for the specification of displacement, traction or spring constants normal to a (not necessarily plane) surface. Specification of conditions other than zero traction or flux in the tangent plane of the surface is unreliable.

In the local coordinate system the outer normal direction is the first coordinate direction.

Once a local boundary condition is specified on a node, the rest of the required boundary conditions on that node must be specified in local coordinates.

Once a local boundary condition is specified on a node of an element, the rest of the required boundary conditions on that element must be specified in local coordinates.

Local boundary conditions are not applicable to scalar problems (i.e. heat conduction).
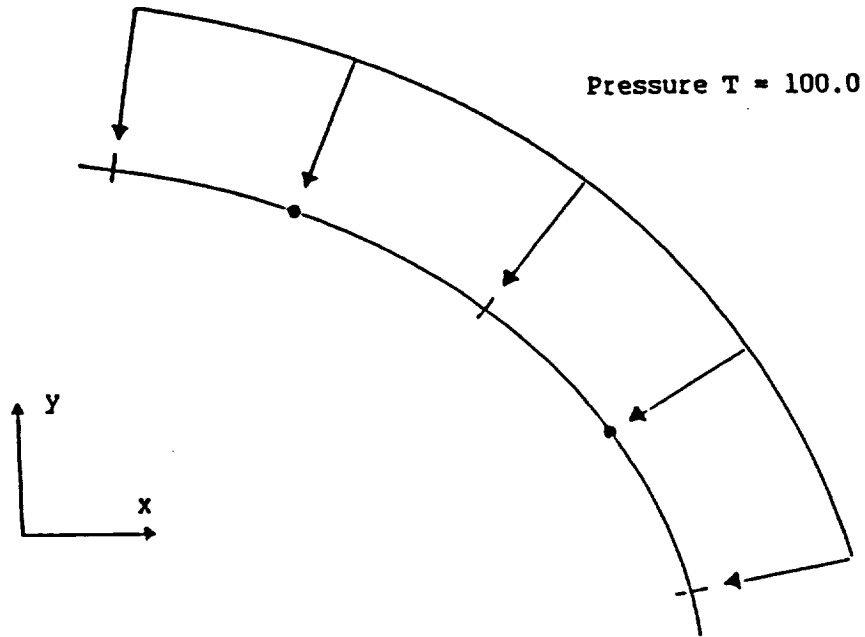
Figure for **BCSET: LOCAL card
Local traction boundary condition input

5.5.4	DEFINITION OF SURFACE FOR APPLICATION OF

BOUNDARY CONDITIONS

In the boundary element method, the primary load variable is traction (or flux), which **acts over a surface area, not nodal forces (or sources)** as in the finite element method. This means that in defining the region of application of a boundary condition in **BEST-CMS** it is necessary to specify both the nodal points and the elements involved.

If no boundary condition is specified (for a particular component) at a node, the primary load variable (of that component) is assumed to be zero.

The input lines involved in defining the element and nodes for a particular boundary condition set are described in this section.

GMR   IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the GMR of the surface on which the boundary condition is to be defined.

Input Variables -

IDGMR   (Alphanumeric)   - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given boundary condition set can involve only a single GMR. If a boundary condition is to be applied to more than one GMR, a separate boundary condition set must be defined for each GMR.

Examples of Use -

1. Identifies the GMR name REG1 in connection with the specification of boundary conditions.

```
**BCSET
  ID  TRAC1
  VALUE
  GMR1  REG1
  SURFACE SURF1
  ELEMENTS   101    102
  TRAC  1
  SPLIST  ALL
  T  1  100.0
$ end of input data
```

SURF   IDSUR

Status - OPTIONAL

Full Keyword - SURFACE

Function - Identifies the surface within the selected GMR on which the boundary condition is to be defined (NAME on SURF card in **GMR input).

Input Variables -

   IDSUR   (Alphanumeric)   - REQUIRED

Additional Information -

   Either this keyword or the HOLE keyword must be input for each boundary condition set.

   A boundary condition set can involve only a single surface. If a boundary condition is to be applied to more than one surface, then a separate boundary condition set must be defined for each surface involved.

   It is recommended that, whenever possible, surfaces be made to coincide with the regions over which boundary conditions are to be applied. This considerably simplifies the definition of surface for application of boundary condition.

   If the SURF card is not followed by an ELEM or POIN card, then BEST-CMS will apply the boundary condition to all of the elements in the surface IDSUR.

Examples of Use -

   1. Identifies the surface name SURF1 relevant to the specification of boundary conditions.

```
**BCSET
  ID   TRAC1
  GMR   GMR1
  SURFACE SURF1
  ELEMENTS  101   102    103
  TRAC  1
  SPLIST  ALL
  T   1  100.0
```
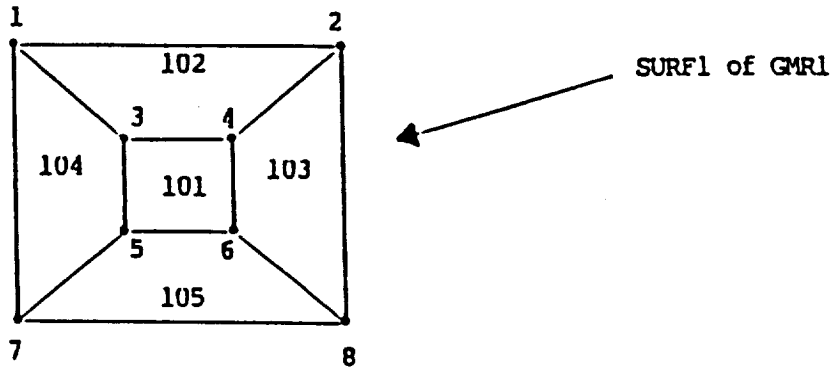
Case 1 Input   Result

GMR1   Boundary condition applied at nodes 1
SURF1   through 8.
                       Over elements 101 through 105.

Case 2 Input
GMR1   Boundary condition applied at nodes 3,
SURF1   4, 5 and 6..
ELEM   101   Over element 101 only.

Case 3 Input   Boundary condition applied at nodes 3,
GMR1   4, 5 and 6.
SURF1   Over elements 102, 103 and 105.
ELEM   102 103 105
POIN   3   4   5   6

Case 4 Input   Boundary condition applied at node 3.
GMR1   Over element 101 only.
SURF1
ELEM   101
POIN 3

Figure for **BCSET: SURF, ELEM and POIN cards
3-D boundary subset definition

ELEM   EL1 EL2 ...   ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the elements of the surface IDSUR to which a boundary condition is to be applied.

Input Variables -

> EL1,EL2,...,ELN   (Integer)   - REQUIRED
>
> User element numbers of the elements of surface IDSUR which are to be included within the boundary condition set.

Additional Information -

> The effect of this card is to restrict the application of the boundary condition to a portion of the surface IDSUR.
>
> This input may be continued on more than one card. Each card must begin with the keyword ELEM.
>
> **If the ELEM card is not followed by a POIN card, then BEST-CMS** will apply the boundary condition to all of the source points in the specified elements.

Examples of Use -

1. Specifies three elements on the surface SURF1 on which traction boundary conditions are given.

```
**BCSET
  ID  DISP2
  SURFACE  SURF1
  ELEMENTS  101   102   103
  DISP  2
  SPLIST  ALL
  T  1  0.0
**BCSET
```

POIN   P1 P2 ... PN

Status - OPTIONAL

Full Keyword - POINTS

Function - Restricts the application of a boundary condition to a subset of the source points lying on the surface IDSUR.

Input Variables -

P1,P2,...,PN   (Integer)   - REQUIRED

Additional Information -

This card restricts the application of the boundary condition to the source points specified.

This card may be repeated as often as required. Each card must begin with the keyword.

If the POIN card is specified, then BEST-CMS will apply the boundary condition to and only to the source points specified in this list.

This card is not allowed in a rigid element specification.

Examples of Use -

1. Time-dependent input (in the x-direction) for points 5, 6, 7, and 8 over elements 102 and 103.

```
**BCSET
  ID BC1
  VALUE
  GMR REG3
    SURFACE BOTTOM
    ELEMENT 102 103
    POINT 5 6 7 8
    TIME 2.0 5.0 10.0
    TRAC 1
    SPLIST 6       7       5       8
    T 1 100.0 100.0 100.0 100.0
    T 2 200.0 200.0 300.0 300.0
    T 3 500.0 600.0 700.0 200.0
```

```
TIME      T1 T2 ...  TN
```

Status - OPTIONAL

Full Keyword - TIMES

Function - Specifies the times at which the variable involved in the boundary condition set will be specified.

Input Variables -

T1   (Real)   - REQUIRED

First time point for boundary condition specification.

T2,...,TN   (Real)   - OPTIONAL

Subsequent time points for boundary condition specification.

Additional Information -

This input may be continued on more than one card if required. Each card must begin with the keyword TIME.

The time values input on this card need not agree with the times at which output was requested in the case control input. Different sets of time points may be used for different boundary conditions in the same analysis.

The time points must be specifed in ascending order.

Boundary condition values at other than input times are calculated by linear interpolation.

If a time card does not appear, the variables involved in the boundary condition are assumed to be time independent. Consequently, only a single time point may be specified for the SPACE/TIME VARIATION (as defined in Section 5.5.6).
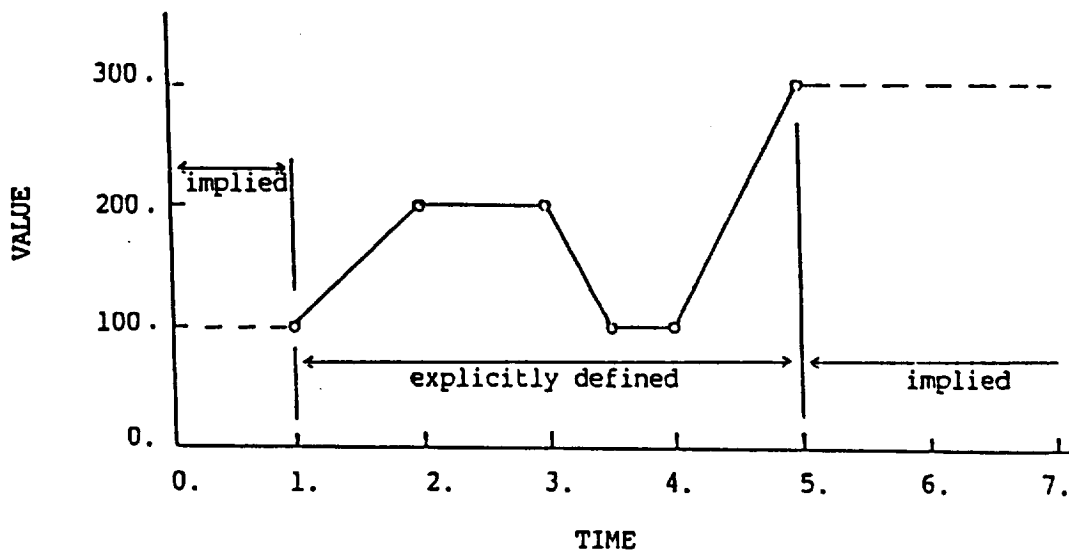
Example:

```
**BCSET
  ID TRAC1
  VALUE
    CHR CHR1
    SURF SURF1
    ELEMENT 3
      TIME 1.0  2.0  3.0  3.5  4.0  5.0
      TRAC 1
      SPLIST ALL
        T 1  100.0
        T 2  200.0
        T 3  200.0
        T 4  100.0
        T 5  100.0
        T 6  300.0
```

Figure for **BCSET: (all specified values)
VALUE vs. TIME for boundary condition input

5.5.5	VALUE BOUNDARY CONDITIONS FOR SURFACE ELEMENTS

TRAC   IDIR

Status - OPTIONAL

Full Keyword - TRACTION

Function - Indicates that the IDIR component of traction will be specified for all nodes of the current boundary condition set.

Input Variables -

IDIR   (Integer)   - REQUIRED

Defines the component direction in which traction is specified. For cartesian coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

Additional Information -

This card can only be used in a boundary condition data set containing the VALU card.

Up to three sets of traction and/or displacement specifications may be included in the same boundary condition data set. All must refer to the same boundary condition set. Only one condition (displacement or traction) may be applied in a given component direction.

The default condition is always to set traction to zero. After all boundary condition data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction conditions.

The TRACTION input line must be immediately followed by the space/time variation.

Examples of Use -

1. Defines a traction of magnitude 100.0 units in the positive y direction.

```
**BCSET
  ID TRAC1
  VALUE
  GMR   REG1
  SURFACE   SURF1
  ELEMENTS  105   106
  POINTS    112   113   114
  TRAC 2
  SPLIST    112   113   114
  T   1   100.0   100.0   100.0
```

Examples of Use -

2. Defines a traction of magnitude 100.0 units in the direction of the outward normal.

```
**BCSET
  VALUE
  LOCAL
  GMR   GMR1
  SURFACE SURF1
  ELEMENTS  101  102
  TRAC 1
  SPLIST ALL
  T   1   100.0
$ end of data set
```

DISP   IDIR

Status - OPTIONAL

Full Keyword - DISPLACEMENT

Function - Indicates that the IDIR component of displacement will be specified for all nodal points contained in the current boundary condition set.

Input Variables -

IDIR   (Integer)   - REQUIRED

Defines the component direction in which displacement is specified. For global coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

Additional Information -

This card can only be used in a boundary condition data set containing the VALU card.

Up to three sets of traction and/or displacement specifications may be included in the same boundary condition data set. All must refer to the same boundary condition set. Only one condition (displacement or traction) may be applied in a given component direction.

The default condition is always to set traction to zero. After all boundary condition data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction conditions.

The DISPLACEMENT input line must be immediately followed by space/time variation.

Examples of Use -

1. Defines complete fixity of nodes, (i.e., displacements in x and y directions are zero) of three elements for a two-dimensional elasticity problem.

**BCSET

```
ID  DISP1
VALUE
GMR  GMR1
SURFACE  SURF1
ELEMENTS  104  105  106
DISP  1
SPLIST  ALL
T  1  0.0
DISP  2
SPLIST  ALL
T  2  0.0
**BCSET
ID  TRAC1
```

FLUX

Status - OPTIONAL

Full Keyword - FLUX

Function - Indicates that the flux will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

FLUX is only valid for HEAT or CTHErmal analysis.

This card can only be used in a boundary condition data set containing the VALU card.

The specification of flux may be included with up to three sets of traction and/or displacement specifications in the same boundary condition data set for concurrent analyses. All must refer to the same boundary condition set.

When applicable, the default condition is to set flux to zero.

The FLUX input line must be immediately followed by the space/time variation.

Examples of Use -

1. Defines zero flux conditions across three elements.

```
**BCSET
  ID  ENTER1
  VALUE
  GMR  GMR1
  SURFACE  SURF1
  ELEMENTS  22  23  24
  FLUX
  SPLIST  ALL
  T  1  0.0
```

TEMP

Status - OPTIONAL

Full Keyword - TEMPERATURE

Function - Indicates that the temperature will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

TEMP is only valid for HEAT or CTHErmal analysis.

This card can only be used in a boundary condition data set containing the VALU card.

The specification of temperature may be included with up to three sets of traction and/or displacement specifications in the same boundary condition data set for concurrent analyses. All must refer to the same boundary condition set.

When applicable, the default condition is to set flux to zero.

The TEMP input line must be immediately followed by the space/time variation.

Examples of Use -

1. Indicates that a constant temperature is specified on the relevant elements.

```
**BCSET
  ID   TOP
  VALUE
  GMR   GMR2
  SURFACE   SURF2
  ELEMENTS   218   219
  VELO   1
  SPLIST   ALL
  T   1   0.0
  TEMP
  SPLIST ALL
  T   1   0.0
```

`RIGI       IDIR`

Status - OPTIONAL

Full Keyword - `RIGID`

Function - Indicates that the IDIR component of displacement, for all nodes of all the elements specified, will be constrained to have the same displacement.

Input Variables -

IDIR   (Integer)   - REQUIRED

Defines the global component direction in which displacement is specified.

1 - x direction
2 - y direction
3 - z direction

Additional Information -

Rigid B.C. input is an element based B.C. specification and all source points that are contained in the elements specified in the B.C. set are automatically included in the rigid B.C. definition. Use of the POINT card and the SPLIST card is NOT allowed.

Use of a LOCAL system is NOT allowed with the RIGID B.C. specification. This card can only be used in a boundary condition data set containing the VALU card.

One, two, or three components may be specified in the same rigid boundary condition data set. All must refer to the same boundary elements list in the B.C. set.

DISP, TRAC, TEMP, and FLUX are not allowed in the same B.C. input set as the RIGID specification. If a separate B.C. set is used, DISP, TRAC, TEMP and FLUX may be applied to an element which also has a RIGID condition specified; however, it can not be in the same direction as the RIGID specification and specifications must be with respect to a global direction.

RIGID elements do not rotate.

RIGID elements will develop resultant traction over the elements listed in the direction specified in order to satisfy the RIGID condition.

Tractions and forces (also see FORC card) acting over the elements of a RIGID specification are self-equilibrating.

The default condition is always to set traction to zero. After all boundary condition data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction conditions.

Examples of Use -

1. Define a rigid specification for elements 101 through 105 in the global x and z directions.

```
**BCSET
  ID RIGID-A
  VALUE
  GMR  GMR1
  SURFACE  SURF1
  ELEMENTS   101   102   103   104   105
  RIGID  1
  RIGID  3

  (end of BC set)
```

FORC

Status - OPTIONAL

Full Keyword - FORC

Function - Indicates that a force will be specified on the rigid elements of this BC set in the direction specified on the RIGID card immediately preceding this card.

Input Variables - NONE

Additional Information -

> This card can only be used in conjunction with the RIGID card, and the RIGID card must immediately precede this card.

> Up to three RIGID cards (corresponding to the three global directions) may be contained in a B.C. set. A force specification may or may not be associated with each direction (i.e., one direction may have a force associated with it and another one may not).

> The traction in a direction specified on a RIGID card integrated over the elements of the B.C. set is equal to the force associated with that direction. If there is no force associated with a direction, the force is assumed to be zero.

> The value of the force associated with the rigid elements is input on the 'T' card (Section 5.5.6). This force may be time dependent; however, the spatial variation specified with the SPLIST card is not applicable. Therefore, the SPLIST card should NOT be input and only one force value should be input (per time) on the 'T' card.

Examples of Use -

1. Define a B.C. set with a rigid definition in the global x and z directions. A time-dependent force is associated with the rigid definition in the x direction. (The z direction has zero force.)

```
**BCSET
  ID RIGID-B
  VALUE
  GMR   GMR1
  SURFACE   SURF1
  ELEMENTS    101   102   103
  RIGID  1
  FORCE
     T   1    10.0
     T   2    20.0
  RIGID  3

  (end of BC set)
```

5.5.6	DEFINITION SPACE/TIME VARIATION

SPLI   N1 N2 ...   NN

Status - REQUIRED

Full Keyword - SPLIST (source point list)

Function - Defines the order in which nodal values of the variable will be input.

Input Variables -

       N1   (Integer or Alphanumeric)   - REQUIRED

       User nodal point number of first node for which data will be input. Optional values are ALL or SAME, described under Additional Information.

       N2,...,NN   (Integer)   - REQUIRED (if ALL or SAME are not used)

       Users nodal point number of all remaining nodes that are defined by the definition of surface for application of Boundary Conditions (Section 5.5.4).

Additional Information -

       This input may be continued on more than one card if required. Each card must begin with the keyword SPLI.

       If N1 = ALL, then **BEST-CMS** assigns the same value of the input variable to all nodes defined by the definition of surface for application of Boundary Condition (Section 5.5.4).

       If N1 = SAME, then the nodal point ordering is taken to be the same as that defined for the immediately preceding boundary condition specification within the same boundary condition set. N1 = SAME may not be used for the first boundary condition specification within a boundary condition set.

       If the node number input is used (i.e. if ALL or SAME are not used) then the **total number of points in SPLI must equal the number of nodes defined by the SURF, ELEM, and POINT cards (Section 5.5.4).**

       Do not use this card if the value to be specified on the 'T' card (see next card) represents a force on a rigid element set.

T   IT V1 V2 ... VN

Status - REQUIRED

Full Keyword - T

Function - Identifies a data card containing values of a variable specified in a boundary condition at time point IT.

Input Variables -

IT   (Integer)   - REQUIRED

Time point as specified on the TIME card in the definition of the surface for application of boundary condition (section 5.5.4). IT = 1 refers to the first time point, IT = 2 the second, etc.

V1,V2,...,VN   (Real)   - REQUIRED

Nodal values of the variable in the nodal point order defined on the SPLI card.

Additional Information -

This input may continue for as many cards as required. Each additional card must begin with T and the time point IT. The input for each new time point must begin on a new card.

The results of various uses of the SPLI and T cards are shown in the figure below.

The results of various uses of the SPLI and T cards are shown in the following figure:
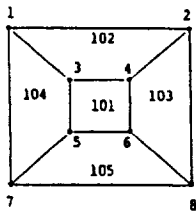
Note differences in traction input of nodes. 3 and 4 over elements 101 and 102

```
Case 1 Input
TIME   0.0
TRAC 1
SPLIST ALL
T 1   100.0
```

```
Case 2 Input
TIME   0.0
TRAC 1
SPLIST 3  4
T 1   100.0  50.0
```

```
Case 3 Input
TIME   0.0  1.0  2.0  4.0
TRAC 1
SPLIST ALL
T 1   50.0
T 2   150.0
T 3   150.0
T 4   50.0
```
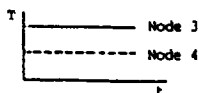
```
Case 4 Input
TIME   0.0  1.0  2.0  4.0
TRAC   1
SPLIST 3  4
T 1   75.0   50.0
T 2   100.0  25.0
T 3   75.0   50.0
T 4   75.0   25.0
```

5.5.7		RELATION BOUNDARY CONDITION

SPRI   IDIR CONST

Status - OPTIONAL

Full Keyword - SPRING

Function - Identifies a boundary condition in which a specified component of displacement and traction are linearly related by a spring constant for all nodal points defined in the current boundary condition.

Input Variables -

IDIR   (Integer)   - REQUIRED

Defines the component direction in which the relationship between displacement and traction is specified. For global coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

CONST   (Real)   - REQUIRED

Additional Information -

The SPRI card can only be used if the RELA card has been input for the current boundary condition data set.

This option is intended primarily to allow a boundary condition that accounts for the response of other structures which are not modelled in **BEST-CMS**, through their stiffnesses (i.e. spring constants).

Currently, only time independent spring boundary conditions are available.

The real portion of the spring constant should be positive. If zero is input, the real portion will be automatically reset to 1.0E-10.

Examples of Use -

1. Specifies a spring of constant 1.0E+03 units in the x direction.

```
**BCSET
  ID  BCS1
  RELATION
  GMR GMR1
  SURFACE SURF1
     ELEMENTS  1  2  3  4
     SPRING    1  1.0E+03
**BCSET
  ID BCS2
```

```
CONV      FCOEFF TAMBT
```

Status - OPTIONAL

Full Keyword - CONVECTION

Function - Identifies a boundary condition in which surface temperature minus ambient temperature is linearly related to flux via a film coefficient for all nodal points defined in the current boundary condition.

Input Variables -

FCOEFF   (Real)   - REQUIRED

Convective film coefficient (h)

TAMBT   (Real)   - OPTIONAL

Ambient temperature of convective fluid (T_a)

Additional Information -

CONV is only valid for HEAT or CTHErmal analysis.

The CONV card can only be used if the RELA card has been input for the current boundary condition data set.

The CONVection option utilizes the relationship:

$$Q = -H * (T_a - T)$$

The film coefficient must be time independent.

If a TIME card was **not** included in the current boundary condition set, then the ambient temperature is time independent and TAMBT must be specified in the CONV card.

If a TIME card was included in the current BCSET, then T card(s) must follow the CONV card to define the time variation of ambient temperature.

No spatial variation of film coefficient nor ambient temperature is permitted within an individual BCSET.

The film coefficient should be set to a positive value. If zero is input, the coefficient will be automatically reset to 1.0E-10.

Examples of Use -

1. Defines a film coefficient of 1.26 units and a surface to ambient temperature difference of 100 units.

```
**BCSET
  ID   BCS1
```

```
RELATION
GMR   GMR1
SURFACE SURF1
   ELEMENTS  1  2  3  4
   CONV      1.26  -100.0
**BCSET
 ID  BCS2
```

| 6.0 | BEST-CMS DATA SET EXAMPLES |

In this chapter example problems are presented to illustrate data preparation for **BEST-CMS**. An attempt has been made to keep the problem geometry as simple as possible so the user is not burdened with undue complexity.

Section	Highlight
6.1	Elastic Analysis with Standard Insert Elements
6.2	Elastic Analysis, Frictional Insert Interface
6.3	Elastic Analysis, Multi-region, Multi-time
6.4	Elastic Analysis, Frictional Insert Interface
6.5	Steady-state Heat Transfer
6.6	Transient Heat Transfer, Multi-region
6.7	Transient Heat Transfer with Symmetry
6.8	Steady-state Uncoupled Thermoelasticity, Multi-region
6.9	Steady-state Uncoupled Thermoelasticity, Frictional Interface
6.10	Transient Uncoupled Thermoelasticity, Multi-region
6.11	Elastoplastic Analysis
6.12	Creep Analysis
6.13	Elastic Analysis, Elliptic Fibers, Sampling Surfaces
6.14	Elastic Analysis Using Micro-surface Fiber, Rigid BCs

```
$  VERIFICATION PROBLEM:  inse001.dat
$------------------------------------------------------------------------

$  ANALYSIS TYPE:
$    ELASTIC-INSERT PROBLEM (USING INSERT ELEMENTS)
$    3-D, STATIC, ELASTIC ANALYSIS OF A BODY WITH INSERTS
$
$  PROBLEM DISCRIPTION:
$    A 3-D, FIXED END THICK CYLINDER WITH FIBERS IN THE CIRCUMFERENTIAL
$    DIRECTION IS ANALYZED SUBJECTED TO INTERNAL PRESSURE IN THE
$    CYLINDER.
$
$  BOUNDARY ELEMENT MODEL:
$    THE INNER AND OUTER RADII OF THE CYLINDER ARE 10 AND 20,
$    RESPECTIVELY,  THE THICKNESS IS 2 AND THE RADII OF THE INSERTS ARE
$    0.5.  BY USING ROLLER BOUNDARY CONDITIONS ON THE FACES OF SYMMETRY,
$    ONLY A FIFTEEN DEGREE SLICE OF THE THICK CYLINDER NEEDS TO BE
$    MODELED.  SIXTEEN EIGHT-NODED QUADRATIC BOUNDARY ELEMENTS ARE USED
$    TO DEFINE THE SIDES OF THE MODEL.  A NINE-NODED ELEMENT IS USED ON
$    BOTH THE INTERNAL AND EXTERNAL FACES OF THE CYLINDER,  AND ONE
$    INSERT ELEMENT IS USED PER INSERT.  NOTE THE INSERT ELEMENTS ARE
$    CURVILINEAR IN GEOMETRY.
$
$  REFERENCE FOR SOLUTION:
$    RESULTS FROM A MULTI-REGION, AXISYMMETRIC BEM ANALYSIS ARE USED
$    FOR COMPARISON WITH THE 3-D INSERT RESULTS FROM THE PRESENT DATA.
$    'DEVELOPMENT OF BEM FOR CERAMIC COMPOSITES', FIRST ANNUAL REPORT,
$    NASA (LEWIS RESEARCH CENTER) - GRANT NUMBER: NAG3-888 (DEC. 1988).
$
$  SOLUTION POINTS TO VERIFY:
$
$                                 RADIAL           CIRCUMFERENTIAL
$                      RADIAL    X-DISPLACEMENT        YY-STRESS
$            NODE     POSITION   AXIS.     BESTCMS    AXIS.    BESTCMS
$            ----     --------   ----------------    ----------------
$            9020      10.0      .913      .97722    5.11     5.5404
$            9021      12.5      .712      .80990    4.28     3.7734
$            9022      15.0      .602      .71539    3.46     3.1350
$            9023      17.5      .537      .66136    3.04     2.9225
$            9024      20.0      .500      .63550    2.72     2.8191
$
$
$  RUN TIME:
$      6.50 X BASE PROBLEM
$
$  MISCELLANEOUS:
$    NONE
$
$------------------------------------------------------------------------
```

```
$

**CASE CONTROL
  TITL         THICK CYLINDER W/ INSERTS UNDER INTERNAL PRESSURE
  PRINT BOUND
  PRINT LOAD
  ECHO

**MATERIAL INPUT
  ID MAT1
  TEMP 70.0
  EMOD 100.0
  POIS 0.3

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
     1       9.9144      -1.3053      0.0
     2      11.1538      -1.4684      0.0
     3      12.3931      -1.6316      0.0
     4      13.6324      -1.7947      0.0
     5      14.8717      -1.9579      0.0
     6      16.1110      -2.1211      0.0
     7      17.3503      -2.2842      0.0
     8      18.5896      -2.4474      0.0
     9      19.8289      -2.6105      0.0
    10      20.0          0.0         0.0
    11      19.8289       2.6105      0.0
    12      18.5896       2.4474      0.0
    13      17.3503       2.2842      0.0
    14      16.1110       2.1211      0.0
    15      14.8717       1.9579      0.0
    16      13.6324       1.7947      0.0
    17      12.3931       1.6316      0.0
    18      11.1538       1.4684      0.0
    19       9.9144       1.3053      0.0
    20      10.0          0.0         0.0
    21      12.5          0.0         0.0
    22      15.0          0.0         0.0
    23      17.5          0.0         0.0
   101       9.9144      -1.3053      2.0
   102      11.1538      -1.4684      2.0
   103      12.3931      -1.6316      2.0
   104      13.6324      -1.7947      2.0
   105      14.8717      -1.9579      2.0
   106      16.1110      -2.1211      2.0
   107      17.3503      -2.2842      2.0
   108      18.5896      -2.4474      2.0
   109      19.8289      -2.6105      2.0
   110      20.0          0.0         2.0
   111      19.8289       2.6105      2.0
   112      18.5896       2.4474      2.0
   113      17.3503       2.2842      2.0
   114      16.1110       2.1211      2.0
   115      14.8717       1.9579      2.0
   116      13.6324       1.7947      2.0
```

```
117      12.3931      1.6316     2.0
118      11.1538      1.4684     2.0
119       9.9144      1.3053     2.0
120      10.0          0.0       2.0
121      12.5          0.0       2.0
122      15.0          0.0       2.0
123      17.5          0.0       2.0
201       9.9144      -1.3053    1.0
203      12.3931      -1.6316    1.0
205      14.8717      -1.9579    1.0
207      17.3503      -2.2842    1.0
209      19.8289      -2.6105    1.0
210      20.0          0.0       1.0
211      19.8289      2.6105     1.0
213      17.3503      2.2842     1.0
215      14.8717      1.9579     1.0
217      12.3931      1.6316     1.0
219       9.9144      1.3053     1.0
220      10.0          0.0       1.0
$
$   DEFINE SURFACE BOUNDARY ELEMENTS
$
 SURFACE SURF1
    TYPE QUAD
    ELEMENTS
 1      1     20     19     18     17     21      3      2
 2      3     21     17     16     15     22      5      4
 3      5     22     15     14     13     23      7      6
 4      7     23     13     12     11     10      9      8
 5    101    102    103    121    117    118    119    120
 6    103    104    105    122    115    116    117    121
 7    105    106    107    123    113    114    115    122
 8    107    108    109    110    111    112    113    123
 9    101    201      1      2      3    203    103    102
10    103    203      3      4      5    205    105    104
11    105    205      5      6      7    207    107    106
12    107    207      7      8      9    209    109    108
13     19    219    119    118    117    217     17     18
14     17    217    117    116    115    215     15     16
15     15    215    115    114    113    213     13     14
16     13    213    113    112    111    211     11     12
17      1    201    101    120    119    219     19     20  220
20    109    209      9     10     11    211    111    110  210
    NORMAL 1 +
$
$   DEFINE INSERT POINTS AND ELEMENTS
$
 INSERT 1000.0
  POINTS
   401   11.1538   1.4684   1.
   402   13.6324   1.7947   1.
   403   16.1110   2.1211   1.
   404   18.5896   2.4474   1.
   501   11.25    0.   1.
   502   13.75    0.   1.
   503   16.25    0.   1.
   504   18.75    0.   1.
   601   11.1538  -1.4684   1.
```

```
  602  13.6324 -1.7947  1.
  603  16.1110 -2.1211  1.
  604  18.5896 -2.4474  1.
  TYPE QUAD
  ELEMENTS
701 .5   401 501 601
  ELEMENTS
702 .5   402 502 602
  ELEMENTS
703 .5   403 503 603
  ELEMENTS
704 .5   404 504 604
```

$ RESTRICT DISPLACEMENT IN THE Z DIRECTION

```
**BCSET
  ID DISP1
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 1 2 3 4 5 6 7 8
    DISP 3
    SPLIST ALL
      T 1 0.0
```

$ ROLLER B.C. ON SYMMETRICAL FACES (SPECIFIED IN THE LOCAL SYSTEM)

```
**BCSET
  ID DISP2
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 9 10 11 12 13 14 15 16
    DISP 1
    SPLIST ALL
      T 1 0.0
```

$ INTERNAL PRESSURE (SPECIFIED IN THE LOCAL SYSTEM)

```
**BCSET
  ID TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TRAC 1
    SPLIST ALL
      T  1 -10.
```

```
$  VERIFICATION PROBLEM :   inse004.dat
$
$  CREATED BY:  D. HENRY, APRIL 1990
$
$  ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$       ELASTOSTATICS  WITH SPRINGS/FRICTION INSERT INTERFACE (1 INSERT)
$
$  PROBLEM DESCRIPTION :
$    CUBE WITH ONE INSERT
$    HOMOGENEOUS   I.E.   MATRIX AND INSERT ARE SAME MATERIAL
$    SPRINGS/FRICTION INSERT INTERFACE (INFINITE SPRING CONSTANTS)
$    CUBE IN TENSION
$    (SO ONLY GAPS OPEN AT INTERFACE)
$
$  BOUNDARY ELEMENT MODEL :
$    6 QUADRATIC ELEMENTS ON THE BOUNDARY
$    1 QUADRATIC INSERT ELEMENT
$
$  REFERENCE FOR ANALYTICAL SOLUTION :  SIMPLE TENSION
$
$  SOLUTION POINTS TO VERIFY :  3-D BEM WITH HOLE ELEMENTS
$
$              (HOLE)          (HOLE)          (INSERT)        (INSERT)
$  POINT  DISPLACEMENT-X  DISPLACEMENT-Y  DISPLACEMENT-X  DISPLACEMENT-Y
$   1006       E-07          1.0288            E-07          1.0288
$ 203003       E-07           .64067           E-07           .46460
$ 303003       .009656        .44569           .009567        .44579
$
$  RUN TIME :
$    X BASE PROBLEM
 **CASE
  ECHO
  TITLE (3-D)    CUBE W/ INTERFACE 1 INSERT (ELASTOSTATICS) IN TENSION
  TIMES  1.
  PRINT BOUND
  PRINT LOAD


 **MATE
  ID MAT1
  TEMP    70.0
  EMOD   100.
  POIS     0.3
  ALPHA    0.01
  DENS     1.
  SPEC  1000.
  COND     1.
```

```
**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
        1          .0000         .0000         .0000
        2          .5000         .0000         .0000
        3         1.0000         .0000         .0000
        4         1.0000         .5000         .0000
        5         1.0000        1.0000         .0000
        6          .5000        1.0000         .0000
        7          .0000        1.0000         .0000
        8          .0000         .5000         .0000
     1001          .0000         .0000         .5000
     1002          .5000         .0000         .5000
     1003         1.0000         .0000         .5000
     1004         1.0000         .5000         .5000
     1005         1.0000        1.0000         .5000
     1006          .5000        1.0000         .5000
     1007          .0000        1.0000         .5000
     1008          .0000         .5000         .5000
     2001          .0000         .0000        1.0000
     2002          .5000         .0000        1.0000
     2003         1.0000         .0000        1.0000
     2004         1.0000         .5000        1.0000
     2005         1.0000        1.0000        1.0000
     2006          .5000        1.0000        1.0000
     2007          .0000        1.0000        1.0000
     2008          .0000         .5000        1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
      101       1      2      3   1003   2003   2002   2001   1001 1002
      102       3      4      5   1005   2005   2004   2003   1003 1004
      103       5      6      7   1007   2007   2006   2005   1005 1006
      104       7      8      1   1001   2001   2008   2007   1007 1008
        1       1      2      3      4      5      6      7      8
      201    2001   2002   2003   2004   2005   2006   2007   2008
    NORMAL  201 +

  INSERT 100.  1.   .01
   INTERFACE   NONLINEAR
    FRIC 0.0
    SPRING 1.0E+06 1.0E+06
    POINT
      3001    .5 .5 0.
      3003    .5 .5  .5
      3005    .5 .5 1.
    TYPE QUAD
    ELEMENT
      301   .05  3001 3003 3005

**BCSET
  ID   BCE
  GMR   GMR1
  SURFACE   SURF11
```

```
     ELEMENTS    103
  TRAC   2
    SPLIST  ALL
      T   1    100.0


**BCSET
  ID   BC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    101
  DISP   2
    SPLIST  ALL
      T   1    0.0


**BCSET
  ID   BC2
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    101 103
    POINT 6 2006 2 2002
  DISP   1
    SPLIST  ALL
      T   1    0.0


**BCSET
  ID   BC3
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    103 101
    POINTS    1001 1007 1003 1005
  DISP   3
    SPLIST 1001 1007 1003 1005
      T   1    0.0   0.0   0.0   0.0
```

```
$  VERIFICATION PROBLEM :   inse005.dat
$
$  CREATED BY:  D. HENRY, APRIL 1990
$
$  ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                 ELASTOSTATICS
$
$  PROBLEM DESCRIPTION :
$    MULTI-REGION CUBE WITH TWO INSERTS IN TENSION
$
$    REGION 1:
$      E-INSERT = 500.        E-MATRIX = 100.
$    REGION 2:
$      E-INSERT = 400.        E-MATRIX = 200.
$
$  BOUNDARY ELEMENT MODEL :
$    2 GMR's
$   12 QUADRATIC ELEMENTS ON THE BOUNDARY
$    4 LINEAR  INSERT ELEMENT
$
$  REFERENCE FOR ANALYTICAL SOLUTION :  NONE
$
$  SOLUTION POINTS TO VERIFY :
$
$    GMR     POINT  TIME    DISPLACEMENT-X   DISPLACEMENT-Y
$     2      1005   3.         -0.26730          4.1706
$     2     303003  3.          .021335          3.3593
$
$  RUN TIME :
$    X BASE PROBLEM

**CASE
  ECHO
  TITLE   MULTI-REGION CUBE W/ HOMOG. 2 INSERT
  TIME  1.  2.  3.


**MATE
  ID MAT1
  TEMP    70.0
  EMOD   100.
  POIS     0.3
  ALPHA    0.01
  DENS     1.
  SPEC  1000.
  COND     1.

**MATE
  ID MAT2
```

```
TEMP      70.0
EMOD     200.
POIS       0.3
ALPHA      0.03
DENS       1.
SPEC    1000.
COND       4.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
          1           .0000           .0000           .0000
          2           .5000           .0000           .0000
          3          1.0000           .0000           .0000
          4          1.0000           .5000           .0000
          5          1.0000          1.0000           .0000
          6           .5000          1.0000           .0000
          7           .0000          1.0000           .0000
          8           .0000           .5000           .0000
       1001           .0000           .0000           .5000
       1002           .5000           .0000           .5000
       1003          1.0000           .0000           .5000
       1004          1.0000           .5000           .5000
       1005          1.0000          1.0000           .5000
       1006           .5000          1.0000           .5000
       1007           .0000          1.0000           .5000
       1008           .0000           .5000           .5000
       2001           .0000           .0000          1.0000
       2002           .5000           .0000          1.0000
       2003          1.0000           .0000          1.0000
       2004          1.0000           .5000          1.0000
       2005          1.0000          1.0000          1.0000
       2006           .5000          1.0000          1.0000
       2007           .0000          1.0000          1.0000
       2008           .0000           .5000          1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
       101       1       2       3    1003    2003    2002    2001    1001 1002
       102       3       4       5    1005    2005    2004    2003    1003 1004
       103       5       6       7    1007    2007    2006    2005    1005 1006
       104       7       8       1    1001    2001    2008    2007    1007 1008
         1       1       2       3       4       5       6       7       8
       201    2001    2002    2003    2004    2005    2006    2007    2008
    NORMAL   201 +

  INSERT   500.  3. 0.02
    POINT
       3001    .5 .5 0.
       3002    .5 .5 0.25
       3003    .5 .5  .5
       3004    .5 .5 0.75
       3005    .5 .5 1.
    TYPE LINEAR
```

```
ELEMENT
  301   .15   3001 3002 3003
  302   .15   3003 3004 3005




**GMR
  ID GMR2
  MAT MAT2
  TREF 70.0
  TINI 0.0
  POINTS
         1            .0000        1.0000          .0000
         2            .5000        1.0000          .0000
         3           1.0000        1.0000          .0000
         4           1.0000        1.5000          .0000
         5           1.0000        2.0000          .0000
         6            .5000        2.0000          .0000
         7            .0000        2.0000          .0000
         8            .0000        1.5000          .0000
      1001            .0000        1.0000          .5000
      1002            .5000        1.0000          .5000
      1003           1.0000        1.0000          .5000
      1004           1.0000        1.5000          .5000
      1005           1.0000        2.0000          .5000
      1006            .5000        2.0000          .5000
      1007            .0000        2.0000          .5000
      1008            .0000        1.5000          .5000
      2001            .0000        1.0000         1.0000
      2002            .5000        1.0000         1.0000
      2003           1.0000        1.0000         1.0000
      2004           1.0000        1.5000         1.0000
      2005           1.0000        2.0000         1.0000
      2006            .5000        2.0000         1.0000
      2007            .0000        2.0000         1.0000
      2008            .0000        1.5000         1.0000

  SURFACE SURF21
    TYPE QUAD
    ELEMENTS
      101        1       2       3    1003   2003   2002   2001   1001 1002
      102        3       4       5    1005   2005   2004   2003   1003 1004
      103        5       6       7    1007   2007   2006   2005   1005 1006
      104        7       8       1    1001   2001   2008   2007   1007 1008
        1        1       2       3       4      5      6      7      8
      201     2001    2002    2003    2004   2005   2006   2007   2008
    NORMAL   201 +

  INSERT   500.  3.  0.02
    POINT
      3001     .5 1.5 0.
      3002     .5 1.5 0.25
      3003     .5 1.5  .5
      3004     .5 1.5 0.75
      3005     .5 1.5 1.
    TYPE LINEAR
```

```
ELEMENT
  301   .15   3001 3002 3003
  302   .15   3003 3004 3005




**INTERFACE
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   103
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS   101

**BCSET
  ID   BCZ
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS   103
    TIME 1. 2. 3.
  TRACTION 2
    SPLIST   ALL
      T   1    100.0
      T   2    200.0
      T   3    300.0

**BCSET
  ID   BC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   101
  DISP   2
    SPLIST   ALL
      T   1    0.0

**BCSET
  ID   BC2
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   101
    POINT   2 2002
  DISP   1
    SPLIST   ALL
      T   1    0.0

**BCSET
  ID   BC3
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS   103
    POINTS   1005 1007
  DISP   3
    SPLIST 1005 1007
      T   1    0.0   0.0
```

```
**BCSET
  ID  BC4
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS     103
    POINT 6 2006
  DISP  1
    SPLIST  ALL
      T  1    0.0

**BCSET
  ID  BC5
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS    101
    POINTS     1003 1001
  DISP  3
    SPLIST 1003 1001
      T  1    0.0  0.0
```

```
$   VERIFICATION PROBLEM :   inse007.dat
$
$   CREATED BY:  D. HENRY, APRIL 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   ELASTOSTATICS
$
$   PROBLEM DESCRIPTION :
$     CUBE WITH ONE INSERT
$     HOMOGENEOUS   I.E.  MATRIX AND INSERT ARE SAME MATERIAL
$     SPRINGS-FRICTION ON INSERT/MATRIX INTERFACE
$     AXIAL COMPRESSION
$
$   BOUNDARY ELEMENT MODEL :
$     6 QUADRATIC ELEMENTS ON THE BOUNDARY
$     2 QUADRATIC INSERT ELEMENT
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$     POINT       X-DISP.     Y-DISP
$     1006        E-06        -20.27
$     POINT       X-DISP.     Y-DISP     X-TRAC.    Y-TRAC.
$     203003      E-06        -6.4103    E-03       173.00
$
$   RUN TIME :
$     X BASE PROBLEM

**CASE
  ECHO
  TITLE (3-D)    CUBE W/ HOMOG. 2 INSERT
  TIMES  1.


**MATE
  ID MAT1
  TEMP    70.0
  EMOD    100.
  POIS     0.3

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
        1          .0000         .0000         .0000
```

```
        2          .5000         .0000         .0000
        3         1.0000         .0000         .0000
        4         1.0000         .5000         .0000
        5         1.0000        1.0000         .0000
        6          .5000        1.0000         .0000
        7          .0000        1.0000         .0000
        8          .0000         .5000         .0000
     1001          .0000         .0000         .5000
     1002          .5000         .0000         .5000
     1003         1.0000         .0000         .5000
     1004         1.0000         .5000         .5000
     1005         1.0000        1.0000         .5000
     1006          .5000        1.0000         .5000
     1007          .0000        1.0000         .5000
     1008          .0000         .5000         .5000
     2001          .0000         .0000        1.0000
     2002          .5000         .0000        1.0000
     2003         1.0000         .0000        1.0000
     2004         1.0000         .5000        1.0000
     2005         1.0000        1.0000        1.0000
     2006          .5000        1.0000        1.0000
     2007          .0000        1.0000        1.0000
     2008          .0000         .5000        1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
      101          1      2      3   1003   2003   2002   2001  1001 1002
      102          3      4      5   1005   2005   2004   2003  1003 1004
      103          5      6      7   1007   2007   2006   2005  1005 1006
      104          7      8      1   1001   2001   2008   2007  1007 1008
        1          1      2      3      4      5      6      7     8
      201       2001   2002   2003   2004   2005   2006   2007  2008
    NORMAL  201 +

  INSERT 100.  1.  0.01
   INTERFACE   NONLINEAR
    FRIC .8
    SPRING 300.0 300.0
    POINT
      3001    .5 .5 0.
      3002    .5 .5 0.25
      3003    .5 .5  .5
      3004    .5 .5 0.75
      3005    .5 .5 1.
    TYPE QUAD
    ELEMENT
      301   .05   3001 3002 3003
      302   .05   3003 3004 3005

 **BCSET
   ID  BC1
   GMR  GMR1
   SURFACE  SURF11
     ELEMENTS  101
   DISP 2
     SPLIST  ALL
       T  1   0.0
```

```
**BCSET
  ID   BC2
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    101 103
    POINT 6 2006 2 2002
  DISP  1
    SPLIST   ALL
       T  1    0.0

**BCSET
  ID   BC3
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    103 101
    POINTS     1001 1007 1003 1005
  DISP  3
    SPLIST 1001 1007 1003 1005
       T  1    0.0   0.0   0.0   0.0

**BCSET
  ID   BC9
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    103
    TRAC 2
    SPLIST   ALL
       T  1    -1000.0
```

```
$   VERIFICATION PROBLEM :    inse101.dat
$
$   CREATED BY:  D. HENRY, APRIL 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   S.S. HEAT CONDUCTION
$
$   PROBLEM DESCRIPTION :
$     CUBE WITH ONE INSERT
$     COND-MATRIX = 100. COND-INSERT = 500.
$     LINEAR TEMPERATURE DISTRIBUTION
$
$   BOUNDARY ELEMENT MODEL :
$     6 QUADRATIC ELEMENTS ON THE BOUNDARY
$     4 QUADRATIC INSERT ELEMENT
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$     ELEMENT      POINT         TEMPERATURE        FLUX
$       103        1005             .0000          10039.
$      1002       200095          46.5600         -17106.
$      1002       300095          51.7190          8553.6
$
$   RUN TIME :
$      X BASE PROBLEM
  **CASE
  ECHO
  TITLE       3-D  CUBE - HEAT TRANSER WITH 1 INSERT  (LINEAR TEMP.)
  HEAT STEADY
  TIMES  1.

**MATE
  ID MAT1
  TEMP 70.0
  COND 100.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
          1          .0000          .0000          .0000
          2          .5000          .0000          .0000
          3         1.0000          .0000          .0000
          4         1.0000          .5000          .0000
          5         1.0000         1.0000          .0000
          6          .5000         1.0000          .0000
          7          .0000         1.0000          .0000
          8          .0000          .5000          .0000
       1001          .0000          .0000          .5000
       1003         1.0000          .0000          .5000
       1005         1.0000         1.0000          .5000
```

```
     1007           .0000        1.0000           .5000
     2001           .0000         .0000          1.0000
     2002           .5000         .0000          1.0000
     2003          1.0000         .0000          1.0000
     2004          1.0000         .5000          1.0000
     2005          1.0000        1.0000          1.0000
     2006           .5000        1.0000          1.0000
     2007           .0000        1.0000          1.0000
     2008           .0000         .5000          1.0000
   SURFACE SURF11
     TYPE QUAD
     ELEMENTS
     101          1      2      3   1003   2003   2002   2001   1001
     102          3      4      5   1005   2005   2004   2003   1003
     103          5      6      7   1007   2007   2006   2005   1005
     104          7      8      1   1001   2001   2008   2007   1007
       1          1      2      3      4      5      6      7      8
     201       2001   2002   2003   2004   2005   2006   2007   2008
   NORMAL   201 +

   INSERT  1.  500.  1.
     POINT
       91   .5   .5   0.
       92   .5   .5   0.05
       93   .5   .5   0.1
       94   .5   .5    .3
       95   .5   .5    .5
       96   .5   .5    .7
       97   .5   .5   0.9
       98   .5   .5   0.95
       99   .5   .5   1.
     TYPE QUAD
     ELEMENTS
      1000   .1   91 92 93
      1001   .1   93 94 95
      1002   .1   95 96 97
      1003   .1   97 98 99

**BCSET
   ID  BC1
   GMR   GMR1
   SURFACE   SURF11
     ELEMENTS   101
     TEMP  1
     SPLIST ALL
       T  1    100.0

**BCSET
   ID  BC2
   GMR   GMR1
   SURFACE   SURF11
     ELEMENTS   103
     TEMP  1
     SPLIST ALL
       T  1    0.0
```

```
$   VERIFICATION PROBLEM :  inse106.dat
$
$   CREATED BY:  D. HENRY, SEPTEMBER 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   TRANSIENT HEAT TRANSFER
$
$   PROBLEM DESCRIPTION :
$     MULTI-REGION
$     CUBE WITH INSERTS
$     HOMOGENEOUS   I.E.  MATRIX AND INSERT ARE SAME MATERIAL
$     LINEAR TEMPERATURE VARIATION AT STEADY-STATE
$
$   BOUNDARY ELEMENT MODEL :
$     2 REGIONS
$     6 QUADRATIC ELEMENTS PER REGION
$     2 LINEAR INSERTS IN FIRST REGION
$     1 (2 ELEMENT) QUADRATIC INSERT IN SECOND REGION
$
$   REFERENCE FOR ANALYTICAL SOLUTION :  NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$   TIME = 2000.0
$
$     POINT        TEMP.
$     5001       75.9459
$
$
$   RUN TIME :
$     X BASE PROBLEM


**CASE
  ECHO
  TITLE MULTI-REGION CUBE W/ HOMOG. INSERT   (TRANSIENT HEAT CONDUCTION)
  HEAT TRANS
  TIME STEP 20 100.


**MATE
  ID MAT1
  TEMP    70.0
  EMOD    100.
  POIS     0.3
  ALPHA    0.01
  DENS     1.
  SPEC   1000.
  COND     1.

**MATE
  ID MAT2
```

```
TEMP     70.0
EMOD    100.
POIS      0.3
ALPHA     0.01
DENS      1.
SPEC   1000.
COND      1.


**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
        1           .0000          .0000          .0000
        2           .5000          .0000          .0000
        3          1.0000          .0000          .0000
        4          1.0000          .2500          .0000
        5          1.0000          .5000          .0000
        6           .5000          .5000          .0000
        7           .0000          .5000          .0000
        8           .0000          .2500          .0000
     1001           .0000          .0000          .5000
     1002           .5000          .0000          .5000
     1003          1.0000          .0000          .5000
     1004          1.0000          .2500          .5000
     1005          1.0000          .5000          .5000
     1006           .5000          .5000          .5000
     1007           .0000          .5000          .5000
     1008           .0000          .2500          .5000
     2001           .0000          .0000         1.0000
     2002           .5000          .0000         1.0000
     2003          1.0000          .0000         1.0000
     2004          1.0000          .2500         1.0000
     2005          1.0000          .5000         1.0000
     2006           .5000          .5000         1.0000
     2007           .0000          .5000         1.0000
     2008           .0000          .2500         1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
       101      1      2      3   1003   2003   2002   2001   1001 1002
       102      3      4      5   1005   2005   2004   2003   1003 1004
       103      5      6      7   1007   2007   2006   2005   1005 1006
       104      7      8      1   1001   2001   2008   2007   1007 1008
         1      1      2      3      4      5      6      7      8
       201   2001   2002   2003   2004   2005   2006   2007   2008
    NORMAL  201 +

  INSERT  100.  1.  0.01
    POINT
       3001    .25 .25 0.
       3002    .25 .25  .5
       3003    .25 .25 1.
       4001    .75 .25 0.
       4002    .75 .25  .5
```

```
   4003    .75 .25 1.
TYPE LINEAR
ELEMENT
   301   .1  3001  3002 3003
ELEMENT
   401   .1  4001  4002 4003




**GMR
  ID GMR2
  MAT MAT2
  TREF 70.0
  TINI 0.0
  POINTS
          1          .0000          .5000          .0000
          2          .5000          .5000          .0000
          3         1.0000          .5000          .0000
          4         1.0000          .7500          .0000
          5         1.0000         1.0000          .0000
          6          .5000         1.0000          .0000
          7          .0000         1.0000          .0000
          8          .0000          .7500          .0000
       1001          .0000          .5000          .5000
       1002          .5000          .5000          .5000
       1003         1.0000          .5000          .5000
       1004         1.0000          .7500          .5000
       1005         1.0000         1.0000          .5000
       1006          .5000         1.0000          .5000
       1007          .0000         1.0000          .5000
       1008          .0000          .7500          .5000
       2001          .0000          .5000         1.0000
       2002          .5000          .5000         1.0000
       2003         1.0000          .5000         1.0000
       2004         1.0000          .7500         1.0000
       2005         1.0000         1.0000         1.0000
       2006          .5000         1.0000         1.0000
       2007          .0000         1.0000         1.0000
       2008          .0000          .7500         1.0000

  SURFACE SURF21
    TYPE QUAD
    ELEMENTS
      101          1       2       3   1003  2003  2002  2001  1001 1002
      102          3       4       5   1005  2005  2004  2003  1003 1004
      103          5       6       7   1007  2007  2006  2005  1005 1006
      104          7       8       1   1001  2001  2008  2007  1007 1008
        1          1       2       3      4     5     6     7     8
      201       2001    2002    2003   2004  2005  2006  2007  2008
    NORMAL  201 +

  INSERT  100.  1. 0.01
    POINT
      3001     .5 .75 0.
      3002     .5 .75 0.25
      3003     .5 .75  .5
      3004     .5 .75 0.75
```

```
   3005    .5 .75 1.
TYPE QUAD
ELEMENT
   301   .1   3001 3002 3003
   302   .1   3003 3004 3005




**INTERFACE
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    103
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS    101

**BCSET
  ID   BCZ
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS    103
  TEMP
    SPLIST   ALL
      T  1    0.0

**BCSET
  ID   BC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    101
  TEMP
    SPLIST   ALL
      T  1   100.0
```

SYMMETRY

```
$    VERIFICATION PROBLEM :  inse108.dat
$
$    CREATED BY:  D. HENRY, OCTOBER, 1990
$
$    ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERT/HOLE
$                    TRANSIENT HEAT TRANSFER
$
$    PROBLEM DESCRIPTION :
$       CUBE WITH ONE INSERT/HOLE
$       HOMOGENEOUS   I.E.  MATRIX AND INSERT ARE SAME MATERIAL
$       UNIFORM TEMPERATURE AT STEADY-STATE
$       HALF SYMMETRY
$
$    BOUNDARY ELEMENT MODEL :
$       5 QUADRATIC ELEMENTS ON THE BOUNDARY
$       1 QUADRATIC INSERT ELEMENT
$
$    REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$    SOLUTION POINTS TO VERIFY :
$
$    TIME = 2000.0
$
$      POINT      TEMP.      FLUX
$       1006     98.617     0.0
$     403003     99.214    -1.4318
$
$
$    RUN TIME :
$      X BASE PROBLEM

**CASE
  ECHO
  TITLE   HALF SYMMETRY -CUBE W/ HOMOG. INSERT   (TRANSIENT HEAT CONDUCTION)
  HEAT TRANS
  TIME STEP  20  100.
  PRECISION LOW
  SYMM HALF

**MATE
  ID MAT1
  TEMP    70.0
  EMOD    100.
  POIS     0.3
  ALPHA    0.01
  DENS     1.
  SPEC  1000.
  COND     1.

**GMR
  ID GMR1
```

```
MAT MAT1
TREF 70.0
TINI 0.0
POINTS
        1           .0000           .0000           .0000
        2           .5000           .0000           .0000
        3          1.0000           .0000           .0000
        4          1.0000           .5000           .0000
        5          1.0000          1.0000           .0000
        6           .5000          1.0000           .0000
        7           .0000          1.0000           .0000
        8           .0000           .5000           .0000
     1001           .0000           .0000           .5000
     1002           .5000           .0000           .5000
     1003          1.0000           .0000           .5000
     1004          1.0000           .5000           .5000
     1005          1.0000          1.0000           .5000
     1006           .5000          1.0000           .5000
     1007           .0000          1.0000           .5000
     1008           .0000           .5000           .5000
     2001           .0000           .0000          1.0000
     2002           .5000           .0000          1.0000
     2003          1.0000           .0000          1.0000
     2004          1.0000           .5000          1.0000
     2005          1.0000          1.0000          1.0000
     2006           .5000          1.0000          1.0000
     2007           .0000          1.0000          1.0000
     2008           .0000           .5000          1.0000

SURFACE SURF11
  TYPE QUAD
  ELEMENTS
     101        1        2        3     1003     2003     2002     2001     1001  1002
     102        3        4        5     1005     2005     2004     2003     1003  1004
     103        5        6        7     1007     2007     2006     2005     1005  1006
       1        1        2        3        4        5        6        7        8
     201     2001     2002     2003     2004     2005     2006     2007     2008
  NORMAL  201 +

INSERT 100. 1. .01
  POINT
     3001    .5 .5 0.
     3002    .5 .5  .5
     3003    .5 .5 1.
  TYPE QUAD
  ELEMENT
     301   .1  3001  3002  3003

**BCSET
  ID  BC1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS    101
    TEMP
    SPLIST  ALL
      T  1   100.0
```

```
$   VERIFICATION PROBLEM :   inse205.dat
$
$   CREATED BY:  D. HENRY, APRIL 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   S.S. UNCOUPLED THERMOELASTICITY
$
$   PROBLEM DESCRIPTION :
$     MULTI-REGION CUBE WITH TWO INSERTS
$     LINEAR  TEMPERATURE DISTRIBUTION
$     REGION 1:
$        E-INSERT = 500.        E-MATRIX = 100.
$        ALPHA-INSERT = 0.02    ALPHA-MATRIX = 0.01
$        K-INSERT = 3.          K-MATRIX = 1.
$     REGION 2:
$        E-INSERT = 400.        E-MATRIX = 200.
$        ALPHA-INSERT = 0.05    ALPHA-MATRIX = 0.03
$        K-INSERT = 5.          K-MATRIX = 4.
$
$   BOUNDARY ELEMENT MODEL :
$     2 GMR's
$    12 QUADRATIC ELEMENTS ON THE BOUNDARY
$     4 LINEAR  INSERT ELEMENT
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$    GMR     POINT  TIME     DISPLACEMENT-X   DISPLACEMENT-Y
$     2      1005   3.          4.4104           8.7265
$     2     303003  3.         -0.81265          4.4633
$
$   RUN TIME :
$      X BASE PROBLEM

**CASE
   ECHO
   TITLE   MULTI-REGION CUBE W/ HOMOG. 2 INSERT (C-THERMAL)
   CTHERM  STEADY  3  1.


**MATE
   ID MAT1
   TEMP     70.0
   EMOD     100.
   POIS      0.3
   ALPHA     0.01
   DENS      1.
   SPEC     1000.
```

```
     COND      1.

**MATE
  ID MAT2
  TEMP      70.0
  EMOD      200.
  POIS      0.3
  ALPHA     0.03
  DENS      1.
  SPEC    1000.
  COND      4.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
          1             .0000          .0000          .0000
          2             .5000          .0000          .0000
          3            1.0000          .0000          .0000
          4            1.0000          .5000          .0000
          5            1.0000         1.0000          .0000
          6             .5000         1.0000          .0000
          7             .0000         1.0000          .0000
          8             .0000          .5000          .0000
       1001             .0000          .0000          .5000
       1002             .5000          .0000          .5000
       1003            1.0000          .0000          .5000
       1004            1.0000          .5000          .5000
       1005            1.0000         1.0000          .5000
       1006             .5000         1.0000          .5000
       1007             .0000         1.0000          .5000
       1008             .0000          .5000          .5000
       2001             .0000          .0000         1.0000
       2002             .5000          .0000         1.0000
       2003            1.0000          .0000         1.0000
       2004            1.0000          .5000         1.0000
       2005            1.0000         1.0000         1.0000
       2006             .5000         1.0000         1.0000
       2007             .0000         1.0000         1.0000
       2008             .0000          .5000         1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
      101         1      2      3   1003   2003   2002   2001   1001 1002
      102         3      4      5   1005   2005   2004   2003   1003 1004
      103         5      6      7   1007   2007   2006   2005   1005 1006
      104         7      8      1   1001   2001   2008   2007   1007 1008
        1         1      2      3      4      5      6      7      8
      201      2001   2002   2003   2004   2005   2006   2007   2008
    NORMAL   201 +

  INSERT  500.  3.  0.02
    POINT
      3001    .5 .5 0.
      3002    .5 .5 0.25
```

```
 3003    .5 .5  .5
 3004    .5 .5 0.75
 3005    .5 .5 1.
TYPE LINEAR
ELEMENT
 301   .15  3001 3002 3003
 302   .15  3003 3004 3005




**GMR
 ID GMR2
 MAT MAT2
 TREF 70.0
 TINI 0.0
 POINTS
         1          .0000       1.0000        .0000
         2          .5000       1.0000        .0000
         3         1.0000       1.0000        .0000
         4         1.0000       1.5000        .0000
         5         1.0000       2.0000        .0000
         6          .5000       2.0000        .0000
         7          .0000       2.0000        .0000
         8          .0000       1.5000        .0000
      1001          .0000       1.0000        .5000
      1002          .5000       1.0000        .5000
      1003         1.0000       1.0000        .5000
      1004         1.0000       1.5000        .5000
      1005         1.0000       2.0000        .5000
      1006          .5000       2.0000        .5000
      1007          .0000       2.0000        .5000
      1008          .0000       1.5000        .5000
      2001          .0000       1.0000       1.0000
      2002          .5000       1.0000       1.0000
      2003         1.0000       1.0000       1.0000
      2004         1.0000       1.5000       1.0000
      2005         1.0000       2.0000       1.0000
      2006          .5000       2.0000       1.0000
      2007          .0000       2.0000       1.0000
      2008          .0000       1.5000       1.0000

 SURFACE SURF21
  TYPE QUAD
  ELEMENTS
   101         1     2     3  1003  2003  2002  2001  1001 1002
   102         3     4     5  1005  2005  2004  2003  1003 1004
   103         5     6     7  1007  2007  2006  2005  1005 1006
   104         7     8     1  1001  2001  2008  2007  1007 1008
     1         1     2     3     4     5     6     7     8
   201      2001  2002  2003  2004  2005  2006  2007  2008
  NORMAL  201 +

 INSERT  500.  3. 0.02
  POINT
    3001    .5 1.5 0.
    3002    .5 1.5 0.25
```

```
    3003    .5 1.5  .5
    3004    .5 1.5 0.75
    3005    .5 1.5 1.
 TYPE LINEAR
 ELEMENT
    301   .15  3001 3002 3003
    302   .15  3003 3004 3005




**INTERFACE
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   103
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS   101

**BCSET
  ID  BCZ
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS   103
   TIME 1. 2. 3.
  TEMP
    SPLIST  ALL
      T  1   100.0
      T  2   200.0
      T  3   300.0

**BCSET
  ID  BC1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   101
  DISP  2
    SPLIST  ALL
      T  1    0.0
  TEMP
    SPLIST  ALL
      T  1    0.0

**BCSET
  ID  BC2
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   101
    POINT  2 2002
  DISP  1
    SPLIST  ALL
      T  1    0.0

**BCSET
  ID  BC3
  GMR  GMR2
  SURFACE  SURF21
```

```
     ELEMENTS    103
     POINTS    1005 1007
   DISP  3
     SPLIST 1005 1007
       T  1    0.0  0.0


**BCSET
  ID  BC4
  GMR   GMR2
  SURFACE   SURF21
     ELEMENTS    103
     POINT 6 2006
   DISP  1
     SPLIST  ALL
       T  1    0.0

**BCSET
  ID  BC5
  GMR   GMR1
  SURFACE   SURF11
     ELEMENTS    101
     POINTS    1003 1001
   DISP  3
     SPLIST 1003 1001
       T  1    0.0  0.0
```

```
$   VERIFICATION PROBLEM :   inse207.dat
$
$   CREATED BY:  D. HENRY, APRIL 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   S.S. UNCOUPLED THERMOELASTICITY
$
$   PROBLEM DESCRIPTION :
$     CUBE WITH ONE INSERT
$     HOMOGENEOUS   I.E.   MATRIX AND INSERT ARE SAME MATERIAL
$     SPRINGS ON INSERT/MATRIX INTERFACE
$     LINEAR  TEMPERATURE DISTRIBUTION AXIAL TENSION
$
$   BOUNDARY ELEMENT MODEL :
$     6 QUADRATIC ELEMENTS ON THE BOUNDARY
$     2 QUADRATIC INSERT ELEMENT
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$     POINT           X-DISP.        Y-DISP.        Z-DISP.
$      1006            E-08           10.851          E-06
$    303003            .07244         4.5909          E-06
$
$   RUN TIME :
$     X BASE PROBLEM

**CASE
  ECHO
  TITLE (3-D)      CUBE W/ HOMOG. 2 INSERT (C-THERMAL)
  CTHERM  STEADY
  TIMES  1.


**MATE
  ID MAT1
  TEMP     70.0
  EMOD    100.
  POIS      0.3
  ALPHA     0.01
  DENS      1.
  SPEC   1000.
  COND      1.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
```

```
TINI 0.0
POINTS
        1           .0000           .0000           .0000
        2           .5000           .0000           .0000
        3          1.0000           .0000           .0000
        4          1.0000           .5000           .0000
        5          1.0000          1.0000           .0000
        6           .5000          1.0000           .0000
        7           .0000          1.0000           .0000
        8           .0000           .5000           .0000
     1001           .0000           .0000           .5000
     1002           .5000           .0000           .5000
     1003          1.0000           .0000           .5000
     1004          1.0000           .5000           .5000
     1005          1.0000          1.0000           .5000
     1006           .5000          1.0000           .5000
     1007           .0000          1.0000           .5000
     1008           .0000           .5000           .5000
     2001           .0000           .0000          1.0000
     2002           .5000           .0000          1.0000
     2003          1.0000           .0000          1.0000
     2004          1.0000           .5000          1.0000
     2005          1.0000          1.0000          1.0000
     2006           .5000          1.0000          1.0000
     2007           .0000          1.0000          1.0000
     2008           .0000           .5000          1.0000

SURFACE SURF11
  TYPE QUAD
  ELEMENTS
    101        1      2      3   1003   2003   2002   2001   1001 1002
    102        3      4      5   1005   2005   2004   2003   1003 1004
    103        5      6      7   1007   2007   2006   2005   1005 1006
    104        7      8      1   1001   2001   2008   2007   1007 1008
      1        1      2      3      4      5      6      7      8
    201     2001   2002   2003   2004   2005   2006   2007   2008
  NORMAL  201 +

INSERT  100.  1.  0.01
 INTERFACE   NONLINEAR
  FRIC .8
  SPRING 300.0E+06 300.0E+06
  CONV 4.0
  POINT
    3001    .5 .5 0.
    3002    .5 .5 0.25
    3003    .5 .5  .5
    3004    .5 .5 0.75
    3005    .5 .5 1.
  TYPE QUAD
  ELEMENT
    301    .05  3001 3002 3003
    302    .05  3003 3004 3005

**BCSET
 ID  BCZ
 GMR  GMR1
 SURFACE  SURF11
```

```
        ELEMENTS    103
      TEMP
        SPLIST  ALL
          T  1   100.0

  **BCSET
     ID  BC1
     GMR   GMR1
     SURFACE   SURF11
        ELEMENTS    101
     DISP  2
        SPLIST  ALL
          T  1   0.0
     TEMP
        SPLIST  ALL
          T  1   0.0

  **BCSET
     ID  BC2
     GMR   GMR1
     SURFACE   SURF11
        ELEMENTS    101 103
        POINT 6 2006 2 2002
     DISP  1
        SPLIST  ALL
          T  1   0.0

  **BCSET
     ID  BC3
     GMR   GMR1
     SURFACE   SURF11
        ELEMENTS    103 101
        POINTS    1001 1007 1003 1005
     DISP  3
        SPLIST 1001 1007 1003 1005
          T  1   0.0  0.0  0.0  0.0

  **BCSET
     ID  BC9
     GMR   GMR1
     SURFACE   SURF11
        ELEMENTS    103
        TRAC 2
        SPLIST  ALL
          T  1   1000.0
```

MULTI-REGION

```
$   VERIFICATION PROBLEM :   inse211.dat
$
$   CREATED BY:  D. HENRY, SEPTEMBER 1990
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH INSERTS
$                   TRANSIENT UNCOUPLED THERMOELASTICITY
$
$   PROBLEM DESCRIPTION :
$     MULTI-REGION
$     CUBE WITH INSERTS
$     HOMOGENEOUS   I.E.   MATRIX AND INSERT ARE SAME MATERIAL
$     LINEAR TEMPERATURE VARIATION AT STEADY-STATE
$
$   BOUNDARY ELEMENT MODEL :
$     2 REGIONS
$     6 QUADRATIC ELEMENTS PER REGION
$     2 LINEAR INSERTS IN FIRST REGION
$     1 (2 ELEMENT) QUADRATIC INSERT IN SECOND REGION
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$   SOLUTION POINTS TO VERIFY :
$
$
$
$
$   RUN TIME :
$     X BASE PROBLEM


   **CASE
     ECHO
     TITLE MULTI-REGION CUBE W/ HOMOG. INSERT   (TRANSIENT UNCOUPLED THERMOELAS-
TICITY)
     CTHERM TRANSIENT
     TIME STEP 20 100.


   **MATE
     ID MAT1
     TEMP    70.0
     EMOD    100.
     POIS     0.3
     ALPHA    0.01
     DENS     1.
     SPEC  1000.
     COND     1.

   **MATE
```

```
ID MAT2
TEMP     70.0
EMOD    100.
POIS      0.3
ALPHA     0.01
DENS      1.
SPEC   1000.
COND      1.


**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
          1           .0000          .0000          .0000
          2           .5000          .0000          .0000
          3          1.0000          .0000          .0000
          4          1.0000          .2500          .0000
          5          1.0000          .5000          .0000
          6           .5000          .5000          .0000
          7           .0000          .5000          .0000
          8           .0000          .2500          .0000
       1001           .0000          .0000          .5000
       1002           .5000          .0000          .5000
       1003          1.0000          .0000          .5000
       1004          1.0000          .2500          .5000
       1005          1.0000          .5000          .5000
       1006           .5000          .5000          .5000
       1007           .0000          .5000          .5000
       1008           .0000          .2500          .5000
       2001           .0000          .0000         1.0000
       2002           .5000          .0000         1.0000
       2003          1.0000          .0000         1.0000
       2004          1.0000          .2500         1.0000
       2005          1.0000          .5000         1.0000
       2006           .5000          .5000         1.0000
       2007           .0000          .5000         1.0000
       2008           .0000          .2500         1.0000

  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
       101      1      2      3   1003   2003   2002   2001   1001 1002
       102      3      4      5   1005   2005   2004   2003   1003 1004
       103      5      6      7   1007   2007   2006   2005   1005 1006
       104      7      8      1   1001   2001   2008   2007   1007 1008
         1      1      2      3      4      5      6      7      8
       201   2001   2002   2003   2004   2005   2006   2007   2008
    NORMAL   201 +

  INSERT   100.  1. 0.01
    POINT
       3001    .25 .25 0.
       3002    .25 .25  .5
       3003    .25 .25 1.
       4001    .75 .25 0.
```

```
  4002    .75 .25   .5
  4003    .75 .25 1.
 TYPE LINEAR
 ELEMENT
  301   .1   3001  3002 3003
 ELEMENT
  401   .1   4001  4002 4003




**GMR
 ID GMR2
 MAT MAT2
 TREF 70.0
 TINI 0.0
 POINTS
        1         .0000        .5000        .0000
        2         .5000        .5000        .0000
        3        1.0000        .5000        .0000
        4        1.0000        .7500        .0000
        5        1.0000       1.0000        .0000
        6         .5000       1.0000        .0000
        7         .0000       1.0000        .0000
        8         .0000        .7500        .0000
     1001         .0000        .5000        .5000
     1002         .5000        .5000        .5000
     1003        1.0000        .5000        .5000
     1004        1.0000        .7500        .5000
     1005        1.0000       1.0000        .5000
     1006         .5000       1.0000        .5000
     1007         .0000       1.0000        .5000
     1008         .0000        .7500        .5000
     2001         .0000        .5000       1.0000
     2002         .5000        .5000       1.0000
     2003        1.0000        .5000       1.0000
     2004        1.0000        .7500       1.0000
     2005        1.0000       1.0000       1.0000
     2006         .5000       1.0000       1.0000
     2007         .0000       1.0000       1.0000
     2008         .0000        .7500       1.0000

 SURFACE SURF21
  TYPE QUAD
  ELEMENTS
   101        1     2     3  1003  2003  2002  2001  1001 1002
   102        3     4     5  1005  2005  2004  2003  1003 1004
   103        5     6     7  1007  2007  2006  2005  1005 1006
   104        7     8     1  1001  2001  2008  2007  1007 1008
     1        1     2     3     4     5     6     7     8
   201     2001  2002  2003  2004  2005  2006  2007  2008
  NORMAL   201 +

 INSERT  100.  1. 0.01
  POINT
   3001    .5 .75 0.
   3002    .5 .75 0.25
   3003    .5 .75   .5
```

```
   3004    .5 .75 0.75
   3005    .5 .75 1.
TYPE QUAD
ELEMENT
   301  .1  3001 3002 3003
   302  .1  3003 3004 3005




**INTERFACE
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   103
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS   101

**BCSET
  ID  BCZ
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS   103
  TEMP
    SPLIST  ALL
      T  1   0.0

**BCSET
  ID  BC1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   101
  TEMP
    SPLIST  ALL
      T  1   100.0


**BCSET
  ID  BCU1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   101
    POINT  1   3   2001 2003
    DISP 2
    SPLIST  ALL
      T  1   0.0

**BCSET
  ID  BCU2
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   101
    POINT  2 2002
    DISP 1
    SPLIST  ALL
      T  1   0.0
```

```
**BCSET
  ID   BCU3
  GMR   GMR1
  SURFACE  SURF11
    ELEMENTS   101
    POINT  1001 1003
    DISP 3
    SPLIST  ALL
      T  1   0.0

**BCSET
  ID   BCU4
  GMR   GMR2
  SURFACE  SURF21
    ELEMENTS   103
    POINT  6 2006
    DISP 1
    SPLIST  ALL
      T  1   0.0

**BCSET
  ID   BCU5
  GMR   GMR2
  SURFACE  SURF21
    ELEMENTS   103
    POINT  1005 1007
    DISP 3
    SPLIST  ALL
      T  1   0.0
```

```
$
$ VERIFICATION PROBLEM :    inse301.dat
$
$ CREATED BY:  DON HENRY,  1992
$
$ ANALYSIS TYPE : ELASTOPLASTIC FIBER ANALYSIS
$
$ PROBLEM DESCRIPTION :
$ CUBE W/ 1 FIBER OF RADIUS = 0.3
$ VON MISES MODEL,   TRANSVERSE LOADING
$
$ BOUNDARY ELEMENT MODEL :
$    6 QUADRATIC ELEMENTS ON THE BOUNDARY
$
$ REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$ SOLUTION POINTS TO VERIFY :


**CASE
  TITLE 3D PLAS. W/ FIBERS, R=0.3
  TIME  95. 100. 105. 110. 115. 120. 125. 130. 135. 140.
  TIME 145. 150. 155. 160. 165. 170. 175. 180. 185. 190.
  PLAS
  CVCP 1.66

**MATE
  ID MAT1
  TEMP    70.0
  EMOD    15.0E+06
  POIS     0.25
 INELASTIC
  VON MISES
  YIELD   0.1E+06
  CURVE
  15.1E+06  1.

**GMR
  ID GMR1
  MAT MAT1
  TREF 0.0
  POINTS
        1           .0000         .0000         .0000
        2           .5000         .0000         .0000
        3          1.0000         .0000         .0000
        4          1.0000         .5000         .0000
        5          1.0000        1.0000         .0000
        6           .5000        1.0000         .0000
        7           .0000        1.0000         .0000
        8           .0000         .5000         .0000
     1001           .0000         .0000        2.5000
     1002           .5000         .0000        2.5000
```

```
      1003          1.0000        .0000      2.5000
      1004          1.0000        .5000      2.5000
      1005          1.0000       1.0000      2.5000
      1006           .5000       1.0000      2.5000
      1007           .0000       1.0000      2.5000
      1008           .0000        .5000      2.5000
      2001           .0000        .0000      5.0000
      2002           .5000        .0000      5.0000
      2003          1.0000        .0000      5.0000
      2004          1.0000        .5000      5.0000
      2005          1.0000       1.0000      5.0000
      2006           .5000       1.0000      5.0000
      2007           .0000       1.0000      5.0000
      2008           .0000        .5000      5.0000
   SURFACE SURF11
     TYPE QUAD
     ELEMENTS
      101        1      2      3   1003   2003   2002   2001   1001 1002
      102        3      4      5   1005   2005   2004   2003   1003 1004
      103        5      6      7   1007   2007   2006   2005   1005 1006
      104        7      8      1   1001   2001   2008   2007   1007 1008
        1        1      2      3      4      5      6      7      8
      201     2001   2002   2003   2004   2005   2006   2007   2008
     NORMAL   201 +

   INSERT   60.0E+06
     POINT
      3001    .5 .5 0.
      3002    .5 .5 0.5
      3003    .5 .5 1.
      3004    .5 .5 2.5
      3005    .5 .5 4.
      3006    .5 .5 4.5
      3007    .5 .5 5.
     TYPE QUAD
     ELEMENT
      301   .3   3001 3002 3003
      302   .3  ·3003 3004 3005
      303   .3   3005 3006 3007
 **BCSET
   ID  BC1
   GMR  GMR1
   SURFACE  SURF11
     ELEMENTS   101
   DISP  2
     SPLIST  ALL
       T  1    0.0

 **BCSET
   ID  BC2
   GMR  GMR1
   SURFACE  SURF11
     ELEMENTS   103
   TIMES  0. 1000.
   TRAC  2
     SPLIST  ALL
```

```
          T  1  0.0
          T  2 1000.0E+03

**BCSET
  ID  BC3
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS    101 103
    POINT 6 2006 2 2002
  DISP  1
    SPLIST  ALL
      T  1   0.0

**BCSET
  ID  BC4
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS    101 102 103 104
    POINTS    1001 1007 1003 1005
  DISP  3
    SPLIST 1001 1007 1003 1005
      T  1   0.0  0.0  0.0  0.0
```

```
$
$ VERIFICATION PROBLEM :   inse401.dat
$
$ CREATED BY:  DON HENRY,  1993
$
$ ANALYSIS TYPE :  CREEP ANALYSIS WITH FIBERS
$
$ PROBLEM DESCRIPTION :
$ CUBE W/ 4 FIBER OF RADIUS = 0.15,   CREEP MODEL
$ LOADING PERPINDICULAR TO FIBERS
$
$ BOUNDARY ELEMENT MODEL :
$   6 QUADRATIC ELEMENTS ON THE BOUNDARY
$
$ REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$
$ SOLUTION POINTS TO VERIFY :


**CASE
  TITLE 3D CREEP W/ 4 FIBERS, R=0.15
  CREEP 500 0.025 40
  CVCP 1.62

**MATE
  ID MAT1
  TEMP    70.0
  EMOD    15.0E+06
  POIS       0.25
 INELASTIC
  CREEP 1.8E-20 3.0

**GMR
  ID GMR1
  MAT MAT1
  TINT 0.0
  TREF 0.0
  POINTS
        1          .0000         .0000         .0000
        2          .5000         .0000         .0000
        3         1.0000         .0000         .0000
        4         1.0000         .5000         .0000
        5         1.0000        1.0000         .0000
        6          .5000        1.0000         .0000
        7          .0000        1.0000         .0000
        8          .0000         .5000         .0000
     1001          .0000         .0000        2.5000
     1002          .5000         .0000        2.5000
     1003         1.0000         .0000        2.5000
     1004         1.0000         .5000        2.5000
     1005         1.0000        1.0000        2.5000
     1006          .5000        1.0000        2.5000
```

```
       1007            .0000         1.0000        2.5000
       1008            .0000          .5000        2.5000
       2001            .0000          .0000        5.0000
       2002            .5000          .0000        5.0000
       2003           1.0000          .0000        5.0000
       2004           1.0000          .5000        5.0000
       2005           1.0000         1.0000        5.0000
       2006            .5000         1.0000        5.0000
       2007            .0000         1.0000        5.0000
       2008            .0000          .5000        5.0000
   SURFACE SURF11
     TYPE QUAD
     ELEMENTS
       101         1       2       3   1003   2003   2002   2001   1001 1002
       102         3       4       5   1005   2005   2004   2003   1003 1004
       103         5       6       7   1007   2007   2006   2005   1005 1006
       104         7       8       1   1001   2001   2008   2007   1007 1008
         1         1       2       3      4      5      6      7      8
       201      2001    2002    2003   2004   2005   2006   2007   2008
     NORMAL   201 +

   INSERT   60.0E+06
     POINT
       3011    .25        .25         .0
       3012    .25        .25        2.5
       3013    .25        .25        5.0
       3021    .75        .25         .0
       3022    .75        .25        2.5
       3023    .75        .25        5.0
       3031    .25        .75         .0
       3032    .25        .75        2.5
       3033    .25        .75        5.0
       3041    .75        .75         .0
       3042    .75        .75        2.5
       3043    .75        .75        5.0
     TYPE QUAD
     ELEMENT
       311   .15   3011 3012 3013
     ELEMENT
       321   .15   3021 3022 3023
     ELEMENT
       331   .15   3031 3032 3033
     ELEMENT
       341   .15   3041 3042 3043


**BCSET
  ID   BC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   101
  DISP   2
    SPLIST   ALL
      T   1    0.0

**BCSET
  ID   BC2
```

```
   GMR   GMR1
   SURFACE   SURF11
     ELEMENTS   103
   TRAC   2
     SPLIST   ALL
       T   1 1000.0E+03

**BCSET
  ID   BC3
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   101 103
    POINT 6 2006 2 2002
  DISP   1
    SPLIST   ALL
       T   1     0.0

**BCSET
  ID   BC4
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    101 102 103 104
    POINTS    1001 1007 1003 1005
  DISP   3
    SPLIST 1001 1007 1003 1005
       T   1    0.0   0.0   0.0   0.0
```

```
$   VERIFICATION PROBLEM :   inse501.dat
$
$   CREATED BY:  D. HENRY, MARCH 1997
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH ELLIPTICAL FIBER INSERTS
$                   ELASTOSTATICS ANALYSIS
$                   SAMPLING SURFACES ARE EMPLOYED
$
$   PROBLEM DESCRIPTION :
$      ONE REGION RECTANGULAR CUBE IN TENSION
$      WITH ONE INSERT PER REGION
$
$   BOUNDARY ELEMENT MODEL :
$      6 QUADRATIC ELEMENTS ON THE BOUNDARY
$      1 QUADRATIC INSERT ELEMENT
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$


**CASE
   TITLE  INSERT W/ WITH ELLIPTIC CROSS-SECTION
   ECHO
   NEUTRAL

**MATE
   ID MAT1
   TEMP 70.0
   EMOD  3.791E+06
   POIS  0.17

**GMR
   ID GMR1
   MAT MAT1
   TREF 70.0
   POINTS
           1           .0000          .0000          .0000
           2           .5000          .0000          .0000
           3          1.0000          .0000          .0000
           4          1.0000          .5000          .0000
           5          1.0000         1.0000          .0000
           6           .5000         1.0000          .0000
           7           .0000         1.0000          .0000
           8           .0000          .5000          .0000
           9           .5000          .5000          .0000
        1001           .0000          .0000          .5000
        1002           .5000          .0000          .5000
        1003          1.0000          .0000          .5000
        1006           .5000         1.0000          .5000
```

```
          1005         1.0000       1.0000          .5000
          1007          .0000       1.0000          .5000
          2001          .0000        .0000         1.0000
          2002          .5000        .0000         1.0000
          2003         1.0000        .0000         1.0000
          2004         1.0000        .5000         1.0000
          2005         1.0000       1.0000         1.0000
          2006          .5000       1.0000         1.0000
          2007          .0000       1.0000         1.0000
          2008          .0000        .5000         1.0000
          2009          .5000        .5000         1.0000
     SURFACE SURF11
       TYPE QUAD
       ELEMENTS
          101         1      2      3   1003   2003   2002   2001   1001 1002
          102         3      4      5   1005   2005   2004   2003   1003
          103         5      6      7   1007   2007   2006   2005   1005 1006
          104         7      8      1   1001   2001   2008   2007   1007
            1         1      2      3      4      5      6      7      8    9
          201      2001   2002   2003   2004   2005   2006   2007   2008 2009
     NORMAL   201 +

       INSERT  60.0E+06 1.  1.  .22
          POINTS
$ NOTE: the elliptical radii are 0.15 and 0.2
$        the 0.15 radius is aligned with the 0. 1. 0. directions
$        for all points
          401  0.2 0.5 0.5    .15 .2  0. 1. 0.
          501  0.5 0.5 0.5    .15 .2  0. 1. 0.
          601  0.8 0.5 0.5    .15 .2  0. 1. 0.

          TYPE QUAD
          ELEMENTS
$ NOTE: THE RADIUS '0.15' SPECIFIED BELOW ON THE ELEMENT CONNECTIVITY
$        CARD IS IGNORED WHEN CROSS SECTIONAL PROPERTIES ARE SPECIFIED
$        ON THE POINT CARDS (however, this dummy radius must be input)

          701 0.15   401 501 601

       SAMPLING POINTS
          101          .0000        .0000          .2500
          102          .5000        .0000          .2500
          103         1.0000        .0000          .2500
          104         1.0000        .5000          .2500
          105         1.0000       1.0000          .2500
          106          .5000       1.0000          .2500
          107          .0000       1.0000          .2500
          108          .0000        .5000          .2500
          201          .0000        .0000          .7500
          202          .5000        .0000          .7500
          203         1.0000        .0000          .7500
          204         1.0000        .5000          .7500
          205         1.0000       1.0000          .7500
          206          .5000       1.0000          .7500
          207          .0000       1.0000          .7500
          208          .0000        .5000          .7500

       SAMPLING SURFACE SS1
```

```
   101        101   102   103   104   105   106   107   108

       SAMPLING SURFACE SS2
       201        201   202   203   204   205   206   207   208


**BCSET
  ID   BC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   104
  DISP   1
    SPLIST ALL
      T   1    0.0


**BCSET
  ID   BC2
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   104 102
    POINTS   4 2004 8 2008
  DISP   2
    SPLIST   4 2004 8 2008
      T   1    0.0 0.0 0.0 0.0


**BCSET
  ID   BC3
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   104 102
    POINTS   1001 1003 1005 1007
  DISP   3
    SPLIST   1001 1003 1005 1007
      T   1    0.0 0.0 0.0 0.0

**BCSET
  ID   TRAC1
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS   102
  TRAC   1
    SPLIST ALL
      T   1    100.0
```

FIBER, RIGID BCs

```
$   VERIFICATION PROBLEM :   inse502.dat
$
$   CREATED BY:  D. HENRY, MARCH 1997
$
$   ANALYSIS TYPE : THREE-DIMINSIONAL BODY WITH FIBER INSERTS
$                   MULTI-REGION, ELASTOSTATICS ANALYSIS
$                   FIBERS ARE MODELED WITH MICRO-SURFACES
$                   BOUNDARY CONDITIONS INCLUDED A RIGID CONSTRAINT WITH
$                   A TIME DEPENDENT FORCE
$
$   PROBLEM DESCRIPTION :
$     TWO REGION RECTANGULAR CUBE IN TENSION
$     WITH ONE INSERT PER REGION
$
$   BOUNDARY ELEMENT MODEL :
$     6 QUADRATIC ELEMENTS ON THE BOUNDARY / REGION
$     1 QUADRATIC INSERT ELEMENT / REGION
$
$   REFERENCE FOR ANALYTICAL SOLUTION :   NONE
$

**CASE
  TITLE 3-D CUBE - RIGID BCs TEST WITH MULTI-REGION MICRO-SURFACES
  TIME 1. 2.

**MATE
  ID MAT1
  TEMP 70.0
  EMOD  3.791E+06
  POIS  0.17

**GMR MICRO
  ID GMR2
  MAT MAT1
  TREF 70.0
  POINTS
          1          1.0000          .0000          .0000
          2          1.5000          .0000          .0000
          3          2.0000          .0000          .0000
          4          2.0000          .5000          .0000
          5          2.0000         1.0000          .0000
          6          1.5000         1.0000          .0000
          7          1.0000         1.0000          .0000
          8          1.0000          .5000          .0000
       1001          1.0000          .0000          .5000
       1003          2.0000          .0000          .5000
       1005          2.0000         1.0000          .5000
       1007          1.0000         1.0000          .5000
       2001          1.0000          .0000         1.0000
```

```
            2002             1.5000            .0000             1.0000
            2003             2.0000            .0000             1.0000
            2004             2.0000            .5000             1.0000
            2005             2.0000           1.0000             1.0000
            2006             1.5000           1.0000             1.0000
            2007             1.0000           1.0000             1.0000
            2008             1.0000            .5000             1.0000
     SURFACE SURF21
       TYPE QUAD
       ELEMENTS
         101          1      2      3   1003   2003   2002   2001   1001
         102          3      4      5   1005   2005   2004   2003   1003
         103          5      6      7   1007   2007   2006   2005   1005
         104          7      8      1   1001   2001   2008   2007   1007
           1          1      2      3      4      5      6      7      8
         201       2001   2002   2003   2004   2005   2006   2007   2008
     NORMAL   201 +

     INSERT   60.0E+06 1.  1.  .22
       POINTS
         401  1.1 0.5 0.5
         501  1.5 0.5 0.5
         601  1.9 0.5 0.5
       TYPE LINEAR
       ELEMENTS
         701 .2   401 501 601

     SAMPLING POINTS
     1999 1.5 .75 .75

   **GMR MICRO
      ID GMR1
      MAT MAT1
      TREF 70.0
      POINTS
            1             .0000            .0000             .0000
            2             .5000            .0000             .0000
            3            1.0000            .0000             .0000
            4            1.0000            .5000             .0000
            5            1.0000           1.0000             .0000
            6             .5000           1.0000             .0000
            7             .0000           1.0000             .0000
            8             .0000            .5000             .0000
         1001             .0000            .0000             .5000
         1003            1.0000            .0000             .5000
         1005            1.0000           1.0000             .5000
         1007             .0000           1.0000             .5000
         2001             .0000            .0000            1.0000
         2002             .5000            .0000            1.0000
         2003            1.0000            .0000            1.0000
         2004            1.0000            .5000            1.0000
         2005            1.0000           1.0000            1.0000
         2006             .5000           1.0000            1.0000
         2007             .0000           1.0000            1.0000
         2008             .0000            .5000            1.0000
     SURFACE SURF11
       TYPE QUAD
       ELEMENTS
```

```
      101          1      2      3   1003   2003   2002   2001   1001
      102          3      4      5   1005   2005   2004   2003   1003
      103          5      6      7   1007   2007   2006   2005   1005
      104          7      8      1   1001   2001   2008   2007   1007
        1          1      2      3      4      5      6      7      8
      201       2001   2002   2003   2004   2005   2006   2007   2008
   NORMAL   201 +

   INSERT   60.0E+06 1.  1.  .22
     POINT
     401    .1 0.5 0.5
     501    .5 0.5 0.5
     601    .9 0.5 0.5
     TYPE LINEAR
     ELEMENTS
     701 .2   401 501 601

   SAMPLING POINTS
   2999    .5 .75 .75


**INTERFACE
  GMR   GMR1
  SURFACE  SURF11
    ELEMENTS   102
  GMR   GMR2
  SURFACE  SURF21
    ELEMENTS   104

**BCSET
  ID  BC1
  GMR   GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS   1001 1007
  DISP  3
    SPLIST  ALL
      T  1    0.0

**BCSET
  ID  BC2
  GMR   GMR2
  SURFACE  SURF21
    ELEMENTS   102
    POINTS   1003 1005
  DISP  3
    SPLIST  ALL
      T  1    0.0

**BCSET
  ID  BC3
  GMR   GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS   8 2008
  DISP  2
    SPLIST  ALL
      T  1    0.0
```

```
**BCSET
  ID   BC4
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS    102
    POINTS   4 2004
  DISP   2
    SPLIST   ALL
      T   1    0.0

**BCSET
  ID   BC5
  GMR   GMR1
  SURFACE   SURF11
    ELEMENTS    104
    DISP   1
    SPLIST   ALL
    T 1   0.

**BCSET
  ID   RIGIDBC
  GMR   GMR2
  SURFACE   SURF21
    ELEMENTS    102
  TIME 0. 10.
    RIGID   1
    FORCE
    T 1   0.
    T 2   1000.
```

7.0	APPLICATIONS

The analyses developed in **BEST-CMS** have been applied to a vast array of fiber composite problems. A few representative examples are described in the following chapters:

7.1 | INTRODUCTION

In the mesh diagrams of the models containing the fiber inserts, a double line is used to indicate the centerline of the insert elements. The length of these elements are shown in proper proportion for the three-dimensional views, however, the radii of the inserts are not indicated on these diagrams. The double line is a symbolic representation of the insert elements and does not in any way indicate the diameter of the insert. Refer to the example description for the values of the radii.

Throughout this section consistent units are used in the definition of the examples. This means all lengths are defined in the same units and the tractions and the elastic moduli are defined in terms of these lengths as force/length^2. No confusion should arise since the results are reported as non-dimensional quantities.

The first test of the formulation is on a unit cube with a single insert of radius 0.1 through the center of the cube. The cube is subjected to tension and shear in the direction parallel and perpendicular to the insert. The cube has a modulus of 100.0 and a Poisson ratio of 0.3. Consistent units are used for all information described in this problem. An insert with two different moduli of 1,000 and 10,000 is studied. The Poisson ratio of the insert is assumed to be the same as that of the cube.

The problem is analyzed by both the present formulation and by a full three-dimensional multi-region BEM approach. As shown in Fig. 7.2.1, the model for the insert formulation consists of fourteen quadratic boundary elements and the insert contains three quadratic insert elements. The two-region, three-dimensional model shown in Fig. 7.2.2 contains twenty quadratic boundary elements in the first region and sixteen in the second. Note 9-noded elements are used in describing the insert and hole to accurately capture the curvilinear geometry.

In Fig. 7.2.3, the profile of the end displacement of the cube under a uniform normal traction of 100.0 (in parallel with the insert) is shown. The present formulation is in good agreement with the full three-dimensional results for $Ei/E = 10$. For the case $Ei/E = 100$, the insert formulation exhibits greater stiffness than the 3-D results. This difference is contributed to the way the load is distributed from the insert to the composite matrix. In the full 3-D model, the applied traction and the resulting reactions at the fixed end act directly on the end of the insert. In the composite formulation, the insert is assumed not to intersect the boundary surface and therefore the insert is moved back slightly from the end of the cube. The load is therefore transferred through the composite matrix to the end of the insert and to its sides in a manner that is slightly different from the full 3-D analysis.

In Fig. 7.2.4, the stress distribution through the center of the cube (from A to B as indicated in the figure) is shown. Again the results are very good for $Ei/E = 10$, and deviates slightly from the full 3-D results in the second case.

In Figs. 7.2.5 and 7.2.6, the lateral displacements along the side of the cube are shown for a cube subjected to a shear traction of 100. For the case of applied shear perpendicular to the insert (Fig. 7.2.5), the results for both the insert and full 3-D model show good agreement. Once again a slight deviation is observed for $Ei/E = 100$. In the case of the shear traction in the plane of the insert (Fig. 7.2.6) the insert has little effect on the displacement (as anticipated) and all results fall in close proximity.

Existing methods of analysis of composite material based on mechanics of materials have been relatively successful in predicting the behavior of composite material for loading in the longitudinal direction. The properties perpendicular to the direction of the fibers are not so readily predictable by present means. The focus of the present example concerns this lateral behavior.

Four cubes (Fig. 7.3.1) with one, two, five and nine inserts are fixed with a roller boundary condition on one side and subjected to a uniform traction, perpendicular to the inserts. The material properties, given in consistent units, are

$$E^{insert} = 1000 \qquad E^{matrix} = 100$$
$$\nu^{insert} = 0.3 \qquad \mu^{matrix} = 0.3$$

For the cube with one and two inserts, the boundary mesh consists of two quadratic surface elements on each lateral side and four elements on the top and bottom. For the cubes with five and nine inserts, one additional element was added to the side with the applied traction and to the side with the roller boundary condition. The top and bottom faces contain six elements to match the pattern of the sides. In all cases, each insert contained three one-dimensional quadratic elements.

The profile for the end displacement of the cube with one insert and five inserts are shown in Figs. 7.3.2 and 7.3.3. The results are seen to be in good agreement with the two-dimensional results. The 2-D results are approximations since plane stress is assumed. The 3-D solutions for the one insert are within 2% error of the 2-D solution and within 3% for the case of 5 inserts.

Also shown in Fig. 7.3.4 are the average end displacements for the one, two, five and nine inserts. Results show good agreement with 2-D results. For one, two and five inserts, the solutions are within 2% error of the 2-D results and 6% for the case of nine inserts where the volume ratio of the inserts to the total volume is 28.2%. The result is also displayed in a plot of Effective Modulus vs. Insert Volume Ratio in Fig. 7.3.5. The effective modulus is defined as the average stress/average strain. The three-dimensional results follow closely to the two-dimensional solution.

The strength of a cylinder under internal pressure can be increased by adding stiff circumferential insert supports. In the present example, a three-dimensional, open ended thick cylinder are 10 and 20 respectively, the thickness is 2 and the radius of the fully-bonded inserts is 0.5. By using roller boundary conditions on the faces of symmetry, only a fifteen degree slice of the thick cylinder needs to be modeled. As shown in Fig. 7.4.1, sixteen eight-noded quadratic boundary elements are used to define the sides of the model, a nine-noded element is used on both the internal and external faces of the cylinder, and three insert elements are used per insert. Note, the inserts in this problem are curvilinear in geometry. The elastic modulus of the cylinder is assumed to be 100, and the effect of inserts with five different moduli of 100, 250, 500, 750 and 1000 is studied. The Poisson ratio is 0.3 for both the composite matrix and insert, and the internal pressure in the cylinder is 100.

Results from a multi-region, axisymmetric BEM analysis were used for comparison with the 3-D insert results of the present example. The axisymmetric model consists of twenty quadratic boundary elements on the outer surface, and six boundary elements per hole and per insert (Fig. 7.4.2). The radial displacement through the thick cylinder along the top face is shown in Fig. 7.4.3 for all five moduli. The displacement for the composites with low Ei/E ratios are in good agreement with the axisymmetric results, and diverge slightly for higher Ei/E ratios. In Fig. 7.4.4, the circumferential stress is shown for the same points along the top edge. This stress is smooth for the homogeneous case $(Ei/E = 1.0)$ and exhibits increasing fluctuations as the Ei/E ratio increases and the inserts take on more of the load. The circumferential stress of the 3-D insert model is in good agreement with the axisymmetric results for all cases. In Fig. 7.4.5, the radial stress is displayed for the two models. The inserts have little effect on this stress and the curves of the five moduli for both approaches lie in close proximity.

7.5 CUBE WITH MULTIPLE INSERTS WITH RANDOM ORIENTATION

In an attempt to analyze a material with a random fiber structure, cubes with multiple inserts oriented in random directions are studied. The cubes are of unit length and have four boundary elements per side (Fig. 7.5.1a). Randomly oriented fibers of variable length with radii of 0.05 are placed in five cubes in quantities of 5, 10, 15, 20 and 25 (Fig. 7.5.1b-f). Three cases of material properties are considered for each cube. The modulus of the composite matrix is 100 for all cases, however, the modulus of the inserts are 500, 10,000 and 200,000 for the three cases studied. Poisson's ratio is uniformly 0.3 throughout. Roller boundary conditions are employed on three adjacent sides and a uniform normal traction of 100 is applied to a fourth face.

The normal end displacement at the center of the face on the side with the applied traction is plotted against the number of inserts in a cube for the three materials (Fig. 7.5.2). The displacement decreases with increasing number of inserts per cube and increasing Ei/E values as expected.

7.6	A BEAM WITH INSERT REINFORCEMENT
	IN BENDING

In the last example, the applicability of the present formulation to the study of the micromechanical behavior of the ceramic composite is apparent. The present formulation, however, is equally applicable to typical problems encountered by civil engineers. Reinforced concrete can now be modeled exactly as a three-dimensional body and studied in detail for the first time. The present example considers a reinforced concrete beam. Here the concrete plays the role of the composite matrix and the reinforcement bars play the role of the fiber insert. In Fig. 7.6.1, a $4 \times 1 \times 1$ beam with four inserts is modeled using twenty-eight quadratic boundary elements. The effect of the ratio of insert modulus to matrix modulus (Ei/E) is studied for a range of values between 1 and 100. The Poisson ratio is 0.3 for both the beam and reinforced rods.

The beam is completely fixed at one end and a downward shear traction of 100 is applied to the other end. The non-dimensional vertical displacement of the end obtained from the present analysis is shown in Fig. 7.6.2 as a function of Ei/E. The non-dimensional displacement is defined as the end displacement of the reinforced beam divided by the displacement of a homogeneous beam under similar boundary conditions.

The end displacement obtained from the mechanics of material solution is also displayed in Fig. 7.6.2 in non-dimensional form. The curvature of the two plots are very similar but differ in magnitude. This difference is contributed to the fact that although the mechanics of material solution accounts for the stiffening due to the inserts, it does not include the effect of interaction between inserts.

7.7 | LAMINATED FIBER COMPOSITE

A laminated composite fabricated from a fiber composite material is shown in Fig. 7.7.1. The fiber composite is constructed with a single row of fully-bonded fibers oriented in the same direction. A two-ply laminate is then constructed from the fiber composite with the fibers of the two layers oriented at 90° angles. A boundary element model created for the study of this material is shown in Fig. 7.7.2. A small slice containing two inserts in each layer is used. The model consists of two regions. The outer surface of each region is modeled with sixteen quadratic boundary elements and each insert contains two quadratic insert elements. The interface between the two regions is assumed to be a perfect bond, however, the present version of the program also allows for sliding and spring connections.

The composite structure is subjected to bi-axial tension. This is accomplished with normal tractions of 100 applied to two adjacent roller boundary conditions applied to the opposite ends. The elastic modulus of the composite matrix of both regions are assumed to be 100, and the moduli of the inserts very between 100 and 10,000. The Poisson ratio is 0.3 for both the composite matrix and inserts at all times.

Figure 7.7.3 displays the displacement as a function of insert moduli for a point on the interface at the corner of the plate adjacent to the sides with the applied traction. The material exhibits less displacement as the modulus is increased, as expected.

7.8 | HEAT CONDUCTION: CUBE WITH RANDOM INSERTS

The cube with randomly oriented inserts shown in Fig. 7.5.1 was analyzed for heat conduction analysis. The left end was subjected to a prescribed temperature of 100°F and for the right end a temperature of 0° was specified. All other surfaces were assumed to be insulated. Figure 7.8.1 shows the equivalent thermal conductivity of a cube for different insert arrangements. In this analysis the ratio of thermal conductivities between the insert and the matrix was assumed to be 100.

The addition of fiber inserts in a material alters the thermal expansion of the material. The effective coefficient of thermal expansion of a composite material is dependent on many factors such as: The elastic and thermal properties of the individual constituents; the size, shape, orientation, and number of fibers; and the interface interaction between the fiber and matrix. Some investigators [Hopkins and Chamis, 1985] have formulated equations for the determination of the effective material properties of fiber composite materials, including the coefficient of thermal expansion. These equations are, however, limited to specific types of insert arrangements and interface connections. For general arrangements, experiments can be carried out for the prediction of effective material properties. Experiments, however, are expensive and time consuming. The finite element method may be employed for this purpose, however, the convergence of the solution may be slow and the solution may be expensive and difficult to achieve. The present uncoupled thermoelastic BEM implementation is an ideal alternative for the prediction of effective material properties for general, tubular fiber composite materials. The analysis is relatively simple and cost efficient.

A boundary element model of a cube with nine insert fibers is shown in Fig. 7.9.1. The fibers are assumed to be perfectly bonded to the composite matrix. Sixteen boundary elements are used to model the outer surface of the cube and one insert element is used to model each of the nine fibers. The cube is subjected to a uniform temperature increase by simply specifying the temperature on one face of the cube and zero flux on the other five sides. The cube is allowed to expand freely, however, rigid body translation is prevented. The user specified radius of the insert fibers is easily changed to simulate various void ratio, therefore, minimizing the effort required for re-analysis of the cube with different insert to total volume ratios.

In Fig. 7.9.2 the effective coefficient of thermal expansions in the lateral and transverse directions are shown as a function of the insert to total volume ratio for a fiber composite material with the following material properties:

$$E^{matrix} = 17.0 \times 10^6 \text{psi} \qquad E^{insert} = 28.0 \times 10^6 \text{psi}$$

$$\nu^{matrix} = 0.24 \qquad \nu^{insert} = 0.24$$

$$\alpha^{matrix} = 0.5 \times 10^{-6}/F \qquad \alpha^{insert} = 2.1 \times 10^{-6}/F$$

Also shown are the predictions by Hopkins and Chamis (1985). The solutions of the two analysis are in good agreement at low insert to total volume ratios and deviate slightly from one another as the insert to total volume ratio is increased. A graph for a similar

analysis is shown in Fig. 7.9.3 for a perfectly-bonded fiber composite material with the following material properties of aluminum and steel are used.

$$E^{matrix} = 10.0 \times 10^6 \text{ psi} \qquad E^{insert} = 30.0 \times 10^6 \text{ psi}$$
$$\nu^{matrix} = 0.3 \qquad \nu^{insert} = 0.3$$
$$\alpha^{matrix} = 12.0 \times 10^{-6}/F \qquad \alpha^{insert} = 6.0 \times 10^{-6}/F$$

This time the coefficient of thermal expansion of the inserts is less than that of the matrix, and therefore, the effective coefficient of thermal expansion decreases with increasing insert to total volume ratio. Once again good agreement is seen between the solutions of the two analyses.

A cube with five inserts is used to demonstrate the linear thermal resistant interface relation between the matrix and the insert. The boundary element mesh containing sixteen surface elements and five insert elements is shown in Fig. 7.10.1. The radius of the inserts is 0.1. The conductivity of the insert and matrix are both 100.0, however, the conductivity across the matrix/insert interface is varied. The cube is subjected to a temperature of 100.0 on a face parallel to the inserts and a temperature of 0.0 on the opposite face. The other four sides are insulated.

A graph of the flux at the center of the zero temperature face versus the thermal conductivity of the interface is shown in Fig. 7.10.2. For very large values of the interface conductivity the interface connection approaches a perfect bond and the overall solution approaches the solution of a homogeneous cube with a conductivity of 100.0. As the interface conductivity decreases the flux decreases. For an interface conductivity of 0.0 the cube exhibits the solution of a cube with perfectly insulated holes.

The cooling of a cube containing five fiber inserts is analyzed using the thermoelastic formulation. Shown in Fig. 7.10.1 is a boundary element model with sixteen surface elements and five insert elements of radius 0.1. The inserts are assumed to be connected by a linear spring-thermal resistant interface. To demonstrate the effect of the interface connection, the insert and the matrix are assumed to be constructed from the same material with an elastic modulus, a Poisson ratio, a thermal conductivity and a coefficient of thermal expansion of 100.0, 0.3, 100.0, and 0.01, respectively. The thermal conductivity across the matrix-insert interface is held constant at 100.0, however, the spring constants between the insert and the matrix are varied. A roller boundary condition is applied on two opposite faces of the cube and the other four sides are free to expand, however, rigid body translation is prevented. The cube is then subjected to a uniform decrease in temperature of 100.0.

A graph is shown in Fig. 7.11.1 of the resultant tractions at the center of the roller boundary condition faces versus the interface spring constants ($k_n = k_t$). For large spring constant values the interface connection approaches a perfect bond, and since the material properties of the insert and matrix are equal, the overall solution approaches the solution of a homogeneous cube. As the spring constants decrease, the tractions at the roller boundary conditions decrease.

7.12 NONLINEAR MATRIX-INSERT INTERFACE: BEAM IN BENDING

A boundary element model of a 4x1x1 beam with four longitudinal inserts from example 7.6 is shown in Fig. 7.6.1. The effect of the ratio of the insert modulus to matrix modulus (Ei/E) is studied for a range of values between 1.0 and 100.0. The Poisson ratio is 0.3 throughout. A nonlinear frictional interface between the insert and matrix is assumed. The coefficient of friction is 0.2 and the normal tangential spring coefficients are both 1000.0 times the elastic modulus of the matrix.

The beam is completely fixed at one end and a downward shear traction of 100.0 is applied to the other end. The non-dimensional vertical displacement of the end obtained from the present analysis is shown in Fig. 7.12.1 as a function of Ei/E. The non-dimensional displacement is defined as the end displacement of the fiber composite beam divided by the displacement of a homogeneous beam subjected to the same boundary conditions. The plot of end displacement obtained in example 7.6 for a reinforced beam perfectly bonded insert-matrix interface connections is also displayed in Fig. 7.12.1. The beam with the perfectly-bonded inserts exhibits greater stiffness than the beam with the nonlinear insert interface connections, as expected.

7.13 | EFFECT OF POISSON RATIO IN FIBER COMPOSITES

The goal of the present five year project is to develop an accurate, yet practical boundary element program for the analysis of ceramic composite material. A conventional BEM approach is computationally expensive and mesh generation is tedious for an analysis of a solid which may have hundreds of fiber inserts. Therefore the challenge of the present work is to introduce certain approximations in the formulation which will produce an efficient BEM analysis and will not compromise accuracy. One approximation unique to this BEM implementation is the assumption that boundary quantities vary as a trigonometric function around the circumference of the insert and hole. This was shown to be an accurate approximation through the numerical applications of this section. Secondly, the steady-state BEM formulations derived in Chapters 2-4 assume that the Poisson ratio of the insert is equal to the Poisson ratio of the matrix. (This limitation was removed in the formulations of Chapters 5, however, these formulations are more expensive.) This second assumption is investigated in this example using a two-dimensional BEM analysis.

A boundary element model of a cube with a single insert is shown in Fig. 7.13.1. Sixteen quadratic boundary elements are used to model the outer boundary and eight elements are used to model both the hole and insert. The modulus of elasticity of the matrix is 10^6 and the Poisson ratio is 0.25. The elastic modulus of the insert is assumed to range from 10^4 to 10^8 and the Poisson ratio from 0.0 to 0.5. The cube is secured by roller boundary conditions on two adjacent sides, a traction of 10^5 is applied on a third side, and the normal displacement is measured at the center of the remaining face. The ratio of insert diameter to side of cube is 0.2 for the volume of insert to total volume ratio of 0.031. The displacement versus the variation in Poisson ratio ($\nu^{insert}/\nu^{matrix}$) is shown in Fig. 7.13.2 for the elastic modulus ratios (E^{insert}/E^{matrix}) of 0.01, 0.1, 1.0, 10.0, and 100.0. The displacement in these curves are normalized by the displacement obtained for the respective elastic modulus ratios when the Poisson ratio of the insert and matrix are the same ($\nu^I/\nu^M = 1.0$). The displacement is observed to be minimal in all cases. The effect is greatest for the modulus ratio E^I/E^M of 1.0 at the extreme values of ν^I/ν^M. The maximum error, however, is less than 4.5%.

This same problem is analyzed for the insert diameter to side of cube ratio of 0.6 for an insert volume to total volume ratio of 0.283 as shown in Fig. 7.13.3. The normalized displacement versus ν^I/ν^M for five modulus ratios is shown in Fig. 7.13.4. Very minute change with variation of ν^I/ν^M is observed for modulus ratios of 0.01 and 100.0 which are almost indistinguishable from one another on the graph. The variation in Poisson ratio is seen to have a modest effect on the displacement values for modulus ratios of 0.1 and 10.0. For an elastic modulus of 1.0, the effect of Poisson ratio is somewhat tolerable for the midrange values of ν^I/ν^M, however, it has a strong effect for extreme values of ν^I/ν^M.

Next the effect of the Poisson ratio in a cube with multiple fibers interacting with one another is investigated. A cube with nine inserts (Fig. 7.13.5) is analyzed under similar conditions as the analyses of a cube with a single fiber. The volume of insert to total volume ratio is again 0.283. In Fig. 7.13.6 the displacement versus ν^I/ν^M is shown for the multiple fiber composite for a modulus ratio E^I/E^M of 1.0. For comparison, the curve for a cube with a single insert with the same modulus and insert to total volume ratio is shown. The effect with the multiple fiber inserts is just slightly less than the effect with a single insert.

In conclusion, we note that a difference in Poisson ratio between an insert and a matrix should not be ignored if the difference in the modulus of elasticity is small and the volume of insert to total volume is large. Nevertheless, fiber inserts are generally added to material in order to strengthen the material, and the elasticity modulus ratio (E^I/E^M) is usually greater than five. Furthermore, many composite constituents often have comparable Poisson ratio values. Therefore, the assumption that the Poisson ratio of the insert must be equal to the Poisson ratio of the matrix is not very limiting.

In Fig. 7.14.1 a boundary element mesh is shown for the outer surface of a rectangular block containing fifty small insert fibers and one large fiber (insert elements not shown). The insert fibers are aligned parallel to one another in the arrangement shown in Fig. 7.14.2. Due to the relative size of the large central fiber, this fiber is explicitly modeled as a separate region in the conventional boundary element sense. The other fifty fibers are each modeled with one quadratic insert element. The total number of nodes in the problem is five-hundred and ninety-eight. The material properties of the composite constituents are

Matrix	Fiber Type 1	Fiber Type 2
$E^M = 17.2 \times 10^6$ psi	$E^{I1} = 60.0 \times 10^6$ psi	$E^{I2} = 28.0 \times 10^6$ psi
$\nu^M = 0.24$	$\nu^{I1} = 0.24$	$\nu^{I2} = 0.24$
$\alpha^M = 0.5 \times 10^6/\text{F}$	$\alpha^{I1} = 4.0 \times 10^{-6}/\text{F}$	$\alpha^{I2} = 2.1 \times 10^{-6}/\text{F}$

In Fig. 7.14.3 through 7.14.9 displacement contours are shown for a model constrained with roller boundary conditions on three adjacent sides (bottom, left, and back faces) and subjected to a normal traction of 10^5 on the small (right side) face perpendicular to the fibers. In. Figs. 7.14.4, 7.14.6, 7.14.8 and 7.14.9 a uniform temperature decrease of 2400°F has also been applied in addition to the normal traction. Contours for displacement (of the top, traction free face) in the direction parallel to the normal traction is shown in Figs. 7.14.3 and 7.14.4. In Fig. 7.14.5 and Fig. 7.14.6 the contours are shown for the displacement (of the top face) in the direction perpendicular to the normal traction and the fibers. In Fig. 7.14.7 and Fig. 7.14.8 the contours are shown for the displacement (of the top face) parallel to the fibers. Fig. 7.14.9 displays a three-dimensional view of 7.14.8.

In Fig. 7.14.10 a contour plot of displacement is shown for a model subjected to a normal traction on the top face perpendicular to the fibers. The displacement shown is of the top face in the direction parallel to the loading. The opposite face is fixed on rollers, and the lateral faces are traction free (rigid body translation is prevented). The fifty small insert fibers primarily strengthen the matrix uniformly, and the large insert interacts with the strengthened matrix resulting in the deformation displayed in the figures.

An uncoupled thermoelastic analysis of a cube with one and nine inserts is studied. A unit cube is discretized with six quadratic boundary elements on the surface of the cube and one insert element per insert. The radius of the insert is 0.2 for the cube with a single insert and 0.1 for the cube with nine inserts. The material properties, given in consistent units are

$$E^{insert} = 100.0 \qquad E^{matrix} = 1.0$$

$$\nu^{insert} = 0.3 \qquad \nu^{matrix} = 0.3$$

$$k^{insert} = 1000.0 \qquad k^{matrix} = 1.0$$

$$\rho c_{\epsilon}^{insert} = 1.0 \qquad \rho c_{\epsilon}^{matrix} = 1.0$$

The temperature of the cube is initially at zero degrees. On a face parallel to the length of the insert, the temperature is held at $0°$ degrees and the face is supported by rollers. On the opposite face the temperature is suddenly raised to $100°$. All other sides are insulated and rigid body translation is prevented. The results of the flux and end displacement versus nondimensionalized time for the traction free face is given in Fig. 7.15.1 through 7.15.4.

The three-dimensional fiber composite BEM results are compared with conventional, two-dimensional BEM results. In Figs. 7.15.1 and 7.15.2 the flux on the free face is shown for a cube with a single insert and a cube with nine inserts, respectively. Similarly, Figs. 7.15.3 and 7.15.4 display the results for end displacement versus time. The fiber composite analysis is in good agreement with the conventional, two-dimensional results. Slight deviation is expected due to the 2-D assumption and the use of a somewhat crude 3-D discretization.

A boundary element discretization of a turbine blade is shown in Fig. 7.16.1. Half symmetry is employed in this model which consists of a single region with ninety-two quadratic boundary elements. The model is 58.2 mm long, 13.9 mm wide, the radius of the base is 6.95 mm, and the tip is 1.98 mm (from the axis of symmetry) in thickness at the largest point.

A transient heat conduction analysis is first carried out on a homogeneous blade with a conductivity of $k = 0.0216$. The blade initially rests in thermodynamic equilibrium at zero temperature. Then, a gas at a temperature of $1200°$ is assumed to flow over the blade while a gas at a temperature of $500°$ surrounds the base of the blade. At the leading edge of the blade the film coefficient is $h = 0.003395$ and tapers off to $h = 0.00064$ at the trailing edge. At the base of the blade the film coefficient of $h = 0.00005$ is assumed. Figures 7.16.3a through 7.16.7a contain contour plots describing the temperature flow through the blade at 1.0, 2.0, 4.0, 6.0, and 8.0 seconds.

The blade is reanalyzed, this time with eight fibers (four fibers per side) running from the tip of the blade to half way through the base. The radius of the fibers is 0.15 mm and their cross-sectional locations are shown in Fig. 7.16.2. Five (9-noded) insert elements are used to model each fiber. The conductivity of the fibers is 100 times the conductivity of the blade. In Figs. 7.16.3b through 7.16.7b the resulting temperature distributions are shown in contour plots. Overall, the higher conductivity of the fibers increases the heat flow through the blade. Hence, the heat at the tip of the blade is carried to the base faster than in the homogeneous blade. This results in a temperature distribution which is lower than the homogeneous blade near the tip and higher near the base.

7.17	HEAT FLOW IN A CUBE WITH
	MULTIPLE FIBERS

The steady-state heat flow through a unit cube with one, five, and nine fibers is analyzed. The radii of the fibers is 0.1 and the arrangement of the fibers is shown in Figure 7.17.1. The conductivity of the fibers is assumed to be 10 times the conductivity of the composite matrix. A uniform flux of 100 is applied to a side of the cube parallel to the fibers. The opposite side is maintained at a temperature of 0°. The remaining four sides are insulated.

The resulting temperature profiles on the face subjected to the flux is shown in Figure 7.17.2 for the three fiber arrangements. Note, the effect of the number of fibers on the temperature profile. As the number of fibers are increased, the overall conductivity of the cube is increased. Hence, the heat from the applied flux is carried away (to the face which is maintained at 0°) at a higher rate, resulting in lower temperatures. Also note, the local temperature minimums in the temperature profiles are associated with the close proximity of the fibers near the flux boundary.

7.18	EFFECTIVE CONDUCTIVITY IN A
	FIBER COMPOSITE

In problem 7.9, the effective coefficient of thermal expansion was calculated for a fiber composite with nine fibers. In the present example, the conductivity of heat flow in a similar specimen is examined. The boundary element model is shown in Figure 7.9.1. The fibers are assumed to be perfectly bonded to the composite matrix so that the resistance of the heat flow across the interface is zero. Twenty-four boundary elements are used to model the outer surface of the cube and one fiber element is used to model each of the nine fibers. The cube is subjected to a temperature of $0°F$ on a face perpendicular to the fibers and $100°F$ on the opposite face. The remaining four faces are insulated. The total heat flux through the cube is calculated and the effective conductivity in the direction parallel to the fibers can be found. When the boundary conditions are rearranged to produce heat flow perpendicular to the fibers, the effective conductivity in the transverse direction can be determined.

In figure 7.18.1, the effective conductivity in both the lateral and transverse directions are shown as a function of the ratio of the fiber volume to the total volume of the specimen. The fiber composite has the following conductivities:

$$k^{matrix} = 8.2 \text{ Btu/hr ft F} \qquad k^{fiber} = 32.0 \text{ Btu/hr ft F}$$

The solutions are in good agreement with the approximate solutions by Hopkins and Chamis (1985) at low fiber to total volume ratios and deviates slightly from one another when the fiber to total volume ratio is increased.

A similar analysis was carried out for a fiber composite with the following conductivities:

$$k^{matrix} = 25.0 \text{ Btu/hr ft F} \qquad k^{fiber} = 5.8 \text{ Btu/hr ft F}$$

Once again, the results shown in Figure 7.18.2 are in good agreement with the solutions obtained with the formulas given by Hopkins and Chamis (1985).

In problem 7.9, the effective coefficient of thermal expansion was calculated for a fiber composite, and in the above example (7.18) the effective conductivity was determined. In the present example, the effective modulus of elasticity will be calculated for a similar specimen with 9 perfectly bonded fibers. The effective modulus of elasticity can be calculated in both the longitudinal and transverse directions by subjecting the cube to tension in the directions parallel and perpendicular to the fibers, respectively.

In Figure 7.19.1, the effective modulus of elasticity in the longitudinal and transverse directions are shown as a function of the ratio of the fiber volume to the total volume of the specimen. The fiber composite has the following material properties:

$$E^{matrix} = 17.0 \times 10^6 \text{ psi} \qquad E^{fiber} = 28.0 \times 10^6 \text{ psi}$$

$$\nu^{matrix} = 0.24 \qquad \nu^{fiber} = 0.24$$

The solutions are again in good agreement with the approximate solutions by Hopkins and Chamis (1985) for the low fiber to total volume ratios, and deviates slightly from one another when the volume ratio is increased.

A similar analysis is also carried out for the aluminum-steel composite matrix of example 7.9.

$$E^{matrix} = 10.0 \times 10^6 \text{ psi} \qquad E^{fiber} = 30.0 \times 10^6 \text{ psi}$$

$$\nu^{matrix} = 0.3 \qquad \nu^{fiber} = 0.3$$

Once again, good agreement is seen between the solutions as displayed in Figure 7.19.2.

SINGLE FIBER

The elastoplastic analysis of a unit cube with a single fiber of diameter 0.6 is analyzed using the fiber element approach developed in Chapter 6. In order to gage the accuracy of the formulation, a von Mises constitutive model was employed and the solution was compared to a two-dimensional plane stress boundary element analysis. The discretization of the fiber element model is shown in Figure 7.20.1. Six quadratic boundary elements are used to model the outer surface of the composite. Three fiber elements (with cylindrical volume cells of diameter 0.98) were used to model the fiber. The two-dimensional discretization is shown in Figure 7.20.2. Three regions modeled with quadratic boundary elements and volume cells were used. The first region contains twelve boundary elements and is used to model the fiber. A second region (surrounding the fiber) contains twenty-four boundary elements and twelve volume cells and a third region contains twenty-four boundary elements and sixteen volume cells and is used to model the matrix. The elastic modulus of the fiber and the matrix are 60×10^6 psi and 15.0×10^6 psi., respectively, and the Poisson's ratio is assumed to be 0.25 in both materials. The fiber is assumed to remain elastic and the matrix is assumed to have a yield strength of 10.0×10^4 psi.

Two different analyses were carried out. In the first case the matrix is assumed to have a strain hardening parameter of 15.0×10^6 psi, and in the second case zero strain hardening is assumed. The cube is subjected to a normal traction on a side of the cube parallel to the fiber and a roller boundary condition is applied to the opposite face. In figure 7.20.3, the results of the end displacement vs. traction is shown for the two analyses. The 3D strain hardening iterative solution is stiffer than the 2D results. The difference in the solution is partly contributed to the plane stress assumption and to the fact that a finer discretization is used in the 2D analysis. Also, the cylindrical volume cell of the fiber element approach does not completely cover the entire volume of the composite matrix, and therefore, some of the nonlinear behavior is not accounted for.

As a check of the iterative algorithm, the 3D fiber element analysis with strain hardening was conducted using the variable stiffness plasticity algorithm [Henry and Banerjee, 1988b]. The variable stiffness solution is in excellent agreement with the iterative solution (Figure 7.20.3). The analysis with zero strain hardening is also shown in Figure 7.20.3. Once again the 3D solution is stiffer than the 2D results, however, both solutions collapse close to the same load level.

It is important to note that even though the fiber element analysis is three-dimensional, the computational time was slightly less than the solution time of the 2D approach. For problems with additional fibers, the fiber element approach will produce even greater savings!

7.21	ELASTOPLASTICITY: CUBE WITH A
	THIN FIBER

An analysis similar to the previous example is carried out for a unit cube with a fiber of diameter 0.2. The material properties from the previous example are used. The model shown in Figure 7.20.1 is employed with a cylindrical volume cell of diameter 0.98.

A fiber composite analysis is carried out for four different hardening parameters: 15.0×10^6 psi, 8.0×10^6 psi, 2.0×10^6 psi, and 0.0 psi. Figure 7.21.1 displays the end displacement vs. traction results for a cube subjected to a normal traction on a face parallel to the fibers with the opposite face constrained against normal displacement. The thin fiber strengthens the composite to a lesser degree than the larger fiber in the previous problem, resulting in a softer elastic and nonlinear solution and a lower collapse load for the zero hardening case.

An elastoplastic analysis of a cube with 9 fibers in a square packing arrangement is analyzed. The discretization of the fiber element model is shown in Figure 7.22.1. Six quadratic boundary elements are used to model the outer surface of the unit cube and one fiber element is used to model each fiber. The fibers have a diameter of 0.2 and a volume cylinder diameter of 0.32. The elastic modulus of the fiber and matrix is 60.0×10^6 psi and 15.0×10^6 psi, respectively, and the Poisson's ratio is assumed to be 0.25 in both materials. The fibers are assumed to remain elastic and the matrix is assumed to have a yield strength of 10.0×10^4 psi.

The analysis was carried out for a number of different hardening parameters. The cube is subjected to a normal traction on a face parallel to the fibers and a roller boundary condition is applied on the opposite face. In Figure 7.22.2, the end displacement vs. traction is shown. It is interesting to compare these results to the solution of the single fiber in problem 7.20 (Figure 7.20.3) which involved similar material properties, similar loading, and had the same volume of fiber to total volume ratio. The analysis of the specimen with 9 fibers yields at a slightly lower stress level, however, exhibits a greater stiffness in both the elastic and nonlinear solution, than the analysis of the single fiber. The results of the zero hardening case indicate a lower collapse load in the present analysis than in example 7.20.

The elastoplastic fracture model is used to model the constitutive behavior of a unit cube with a fiber of diameter 0.6. The modulus of elasticity of the fiber and the matrix are 1000.0 psi and 100.0 psi, respectively, and the Poisson's ratio is 0.3 for both materials. The parameters associated with the elastoplastic-fracture model are given below in units of psi:

Compression Zone:

$$\tau_y = 1000.0 \quad \tau_{ult} = 10,000.0 \quad A_y = 1000.0 \quad A_{ult} = 2000.0 \quad H = 1000.0$$

Tension Zone:

$$\tau_y = 100.0 \quad \tau_{ult} = 5000.0 \quad A_y = 100.0 \quad A_{ult} = 200.0 \quad H = 100.0$$

Two analyses are carried out. In the first analysis, the cube is subjected to a normal tensile traction on a face parallel to the fiber and roller boundary conditions are applied on the opposite face. In the second analysis, a normal compressive traction is applied in a similar manner. 2D and 3D approaches are used in both analyses. The discretization shown in Figure 7.20.1 is used for the 3D case and the discretization of Figure 7.20.2 is employed for the 2D approach.

The results of both analyses are shown in Figure 7.23.1. In Figure 7.23.2, a close up view is shown for the tension case. The results of the 2D and 3D approaches agree quite well in both the tension and compression analyses.

7.24	ELASTOPLASTIC-FRACTURE: FIBER
	COMPOSITE UNDER AXIAL LOADING

A 1 x 1 x 4 rectangular block has a single fiber at its center running through its length (Figure 7.24.1). The elastic and inelastic material properties of the composite of example 7.23 are assumed in the present analysis. The effect of the fiber radius on the deformation of the composite is carried out. Three cases were examined in which the radius of the fiber is 0.3, 0.2 and 0.0 (homogeneous case).

In Figure 7.24.2, the axial strain at the center of the specimen (inside the fiber) is shown for the three cases under both tension and compression loading. Significant stiffening is observed in both loading configurations as the radius is increased. In previous examples (7.20-7.23) it was observed that the presence of the fibers resulted in a significant loss of strength when the composite was loaded perpendicular to the fibers beyond its elastic limit. In the present analysis, a specimen loaded in tension yields at a very low level. For a specimen in compression the yield is higher and the hardening parameter is greater resulting in a very slight deviation from the elastic solution. Since the fiber remains elastic and the strain hardening is constant, the inelastic deformation is relatively linear once yielding occurs.

7.25 | CREEP ANALYSIS

One of the well known advantages of using a generic viscoplastic model is that at large times the steady state solution of the viscoplastic problem turns out to be identical to the elastoplastic solution with an appropriate yield function and plastic potential. The nine inclusion problems shown in figures 7.25.1 and 7.25.2 was subjected to a traction loading in the direction perpendicular to the longitudinal axes of the fibers. Two analyses were carried out. In the first analysis the material was modeled as an elastoplastic material and the response was investigated under an incremental loading. In the second analysis each load increment was allowed to settle to a steady state creep solution. The final steady state creep solutions are as expected in agreement with the elastoplastic response (Figure 7.25.3).

Next the loading was increased linearly with the time and the average strain-time response is plotted in Figure 7.25.4.

The analysis of three different fiber arrangements is studied in this example. In the first case shown in Figure 7.26.1, straight fibers with circular cross-sections are arranged in a cross pattern. In the second and third cases the fibers are curved and arranged in a weave pattern (Figure 7.26.1). In case 2, the fibers have a nearly circular cross-section and in case 3 the fibers have an elliptical cross-section with a B/A axis ratio of 2.18. The matrix has material properties of $E = 115 \times 10^3$ MPa and $\nu = 0.22$ and the fiber $E = 390 \times 10^3$ MPa and $\nu = 0.17$. The edge of the matrix is given a uniform displacement in the in-plane direction of the fibers as the opposite edge is restrained. The stiffness of the composite, defined as the average normal stress resulting from the deformation divided by the average strain, is displayed in Figure 7.26.2 for the three analyses.

A cube with 16 loosely bundled, circular fibers is shown in Figure 7.27.1. The composite is assumed to have the material properties for SiC/RBSN defined in the previous example. A side of the cube is given a uniform displacement in the principal direction of the fibers while the opposite end is restrained. Analysis is carried out for fibers with various size radii. Figure 7.27.2 displays the axial stiffness (average stress/average strain) versus the fiber-volume ratio.

A unit cube with 13 unidirectional fibers is subjected to a displacement in the direction perpendicular to the fibers. The arrangement of the fibers is shown in Figure 7.28.1. The fibers are assumed to have an elliptical cross-section defined by elliptical radii A and B. The axis B (see Figure 7.28.1) is assumed to be aligned with the direction of the displacement loading. Analyses were carried out for a number of different cross-sectional parameters A and B, in which the cross-sectional area of the fiber is held constant, i.e., $A \cdot B = 0.01$, and hence, the fiber-volume ratio remains constant at 40.84%.

The fiber is assumed to have material properties $E = 390 \times 10^3$ MPa and $\nu = 0.2$ and the matrix has $E = 115 \times 10^3$ MPa and $\nu = 0.2$. Figure 7.28.1 displays a graph of the lateral stiffness, which is defined as the average normal stress across the lateral face divided by the strain, versus the ratio of cross-section parameters A/B. Note when the large axis (B) is aligned with the loading (low values of A/B) the composite exhibits greater stiffness which is expected since $E_{fiber} > E_{matrix}$.

A cube with 13 uniaxial circular fibers is subjected to a uniform normal displacement on one edge while restrained on the opposite edge. The arrangement of the fibers is shown on Figure 7.29.1. The material properties of the matrix are $E = 115 \times 10^3$ MPa and $\nu = 0.25$. The modulus of elasticity for the fiber is $E = 390 \times 10^3$ MPa. Five analyses are carried out in which the Poisson's ratio of the fiber varied between 0.0 and 0.5. Figure 7.29.1 displays the effect that the variation of Poisson's ratios have on the average lateral strain in the fiber composite.

7.30	THERMOELASTIC BEHAVIOR OF SiC/RSBN
	COMPOSITES (Goldberg & Hopkins, 1993,1994)

BESTCMS includes provisions to model mechanical and thermal fiber/matrix interfaces. The interface behavior is incorporated directly into the boundary integral formulation, obviating the need for cumbersome discretization requirements. The mechanical interface is modeled by defining linear or nonlinear spring constants normal to the fiber and in the fiber longitudinal directions. A frictional mechanical interface can also be represented. The thermal interface is modeled by defining a thermal resistance (the ratio of the heat flux to the temperature difference) between the fiber and the matrix.

To examine the longitudinal thermal expansion behavior of unidirectional SiC/RBSN ([0°] fiber orientation) with a fiber volume ratio of 0.24, the average composite strain due to temperature only was computed for several applied temperatures, and the results were compared to experimental values obtained by Bhatt and Palczer. Both strong and weak fiber/matrix interface conditions were examined. For the composite with a weak interface, prior work by Bhatt, et al. (1992) found that the thermal expansion mismatch between the fiber and the matrix results in an interfacial gap forming between the fiber and the matrix during processing. To model the weak interface with its debonding, a combined mechanical/thermal interface (1992) was utilized in the BESTCMS model. A thermal resistance was applied in order to simulate the thermal barrier that the interfacial gap creates between the fiber and the matrix (Aboudi, Pindera, and Arnold, 1993 and Bhatt, et al., 1992). Appropriate spring and frictional parameters were also defined in order to model the effect of the weak interface on the mechanical behavior. For the material with the strong interface, a perfect bonding was assumed and no special interface was applied.

The variation of the thermal strain with temperature is plotted, along with the experimental values for comparison, in Figure 7.30.1 for the material with a strong interface and in Figure 7.30.2 for the material with the weak interface. For both interfacial conditions, the boundary element results compared favorably to the experimental values, indicating that the boundary element technique are able to simulate the thermal expansion behavior for both strongly and weakly bonded materials. One interesting note for the material with the weak interface is that varying the thermal resistance value significantly affected the results (more so than any of the mechanical interface parameters), possibly indicating that the temperature distribution between the constituents plays a significant role in the thermal expansion behavior of the material.

For composites with large diameter fibers and a finite number of fibers through the thickness, Aboudi, et al. (1993) found that classical homogenization theory does not sufficiently account for the microstructural effects on the macromechanical behavior and does

not properly account for the local fluctuations in the macroscopic behavior. To alleviate this difficulty, they developed the FG theory, which explicitly couples microstructural details with the composite macrostructure. To test out this theory, they computed the through-the-thickness temperature and stress profiles due to a 500°C (932°F) temperature gradient for SiC/Ti-15.3 with a fiber volume ratio of 0.40 and a fiber thermal conductivity fifty times that of the matrix. Both uniform and non-uniform fiber spacings were considered. The boundary element method was utilized by Goldberg and Hopkins (1994) in order to verify the results obtained using the FG theory.

Two cross sections were considered, one which included both fiber and matrix, and one in which only matrix material was included. For the material with constant fiber spacing, the temperature profiles for both cross sections are shown in Figure 7.30.3, and temperature profiles for both fiber spacing conditions for the cross section containing both fiber and matrix are shown in Figure 7.30.4. For the cross-section containing both fiber and matrix, the temperature profile exhibits a distinct "staircase" pattern, with a smaller variation seen for the cross section containing matrix only. This pattern is very similar to that found using FG theory, and is very different from that predicted using continuum theory, in which a purely linear temperature distribution would be expected. For the composite with linear fiber spacing (concentrated towards the low temperature end of the composite), the local temperatures are lower throughout the cross section then that which is seen for the material with constant fiber spacing. This result is consistent with FG theory, and is expected due to the concentration of the fibers (with their significantly higher conductivity compared to the matrix), near the low temperature end of the composite. The only significant qualitative difference found between the FG results and the BEM results is that for the fiber/matrix cross-section the FG theory predicted piecewise linear temperature distributions, while the boundary element results were much more curvilinear. The curvilinear BEM results were found to be an artifact of the fact that the BEM results were taken on the surface of the model, and the insert element formulation in BESTCMS includes the assumption that fiber ends are not free surfaces. Boundary element models constructed using explicitly modeled fibers showed the same piecewise linear distribution predicted by FG theory, and the temperature within the *fiber* region was the average of the temperature range in the *fiber* region found using the insert element models.

The model shown in Figure 7.31.1a was fixed in all directions at one end and subjected to transverse bending by applying a traction of $t_x = 100$. Figures 7.31.3 and 7.31.4 show the results of the shear stress distribution on a cutting plane placed along the span at $L = 15$ for linear fiber-matrix interface (bonded and sliding) and the nonlinear fiber-matrix interface (gap and contact), respectively. Obviously for straight fibers once the bond stress is exceeded the pattern of shear stress distribution for the analysis with sliding and that with gaps and contacts are somewhat similar because the matrix dominates the shear behavior. These types of sampling planes are likely to be extremely useful for the practical utility of BESTCMS since the user can now examine the detailed results at different cross-sections.

Figures 7.31.5 and 7.31.6 show the results of Von Mises stresses at the fiber-matrix interface for the geometry shown in Figure 7.31.1bb subjected to uniaxial tension $t_z = 100$. It is extremely interesting to note that the stress distribution changes significantly for each case and the distribution is far more complex than the approximately uniform variation that is likely to occur for a straight fiber. Most interestingly the maximum Von-Mises stresses which are nearly equal for the sliding and bonded contacts drops nearly by a factor of two for the analysis with gaps and contacts. It is therefore extremely important to include the correct definition of precise geometry and interface conditions for the micromechanical analysis of woven composites.

Figures 7.31.7 and 7.31.8 show similar results of Von Mises stresses at the fiber-matrix interface of fiber 1 for the two curvilinear fiber problem defined in Figure 7.31.2 subjected to transverse bending by a traction $t_x = 100$ applied at one end and completely fixed at the other end. These once again show that high stress concentrations occur at the fiber matrix interface and the distribution is quite complex. In addition, the magnitude of the maximum Von Mises stresses are similar for the bonded and sliding interfaces and drops considerably when nonlinear analysis with gaps and contacts are carried out.
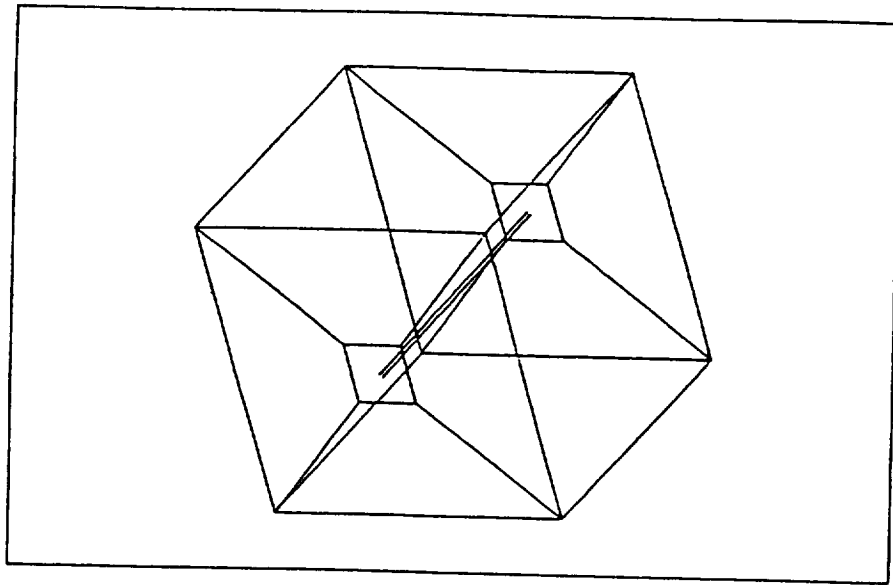
Fig. 7.2.1 Discretization of an Insert in a Unit Cube Utilizing Quadratic Insert Elements
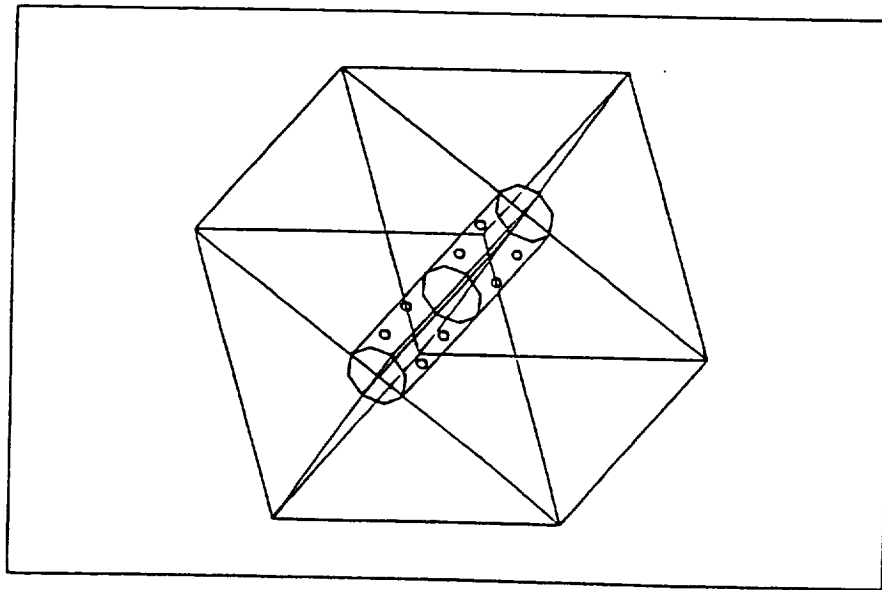
Fig. 7.2.2 Full Three-dimensional, Multi-region Discretization of an Insert in a Unit Cube
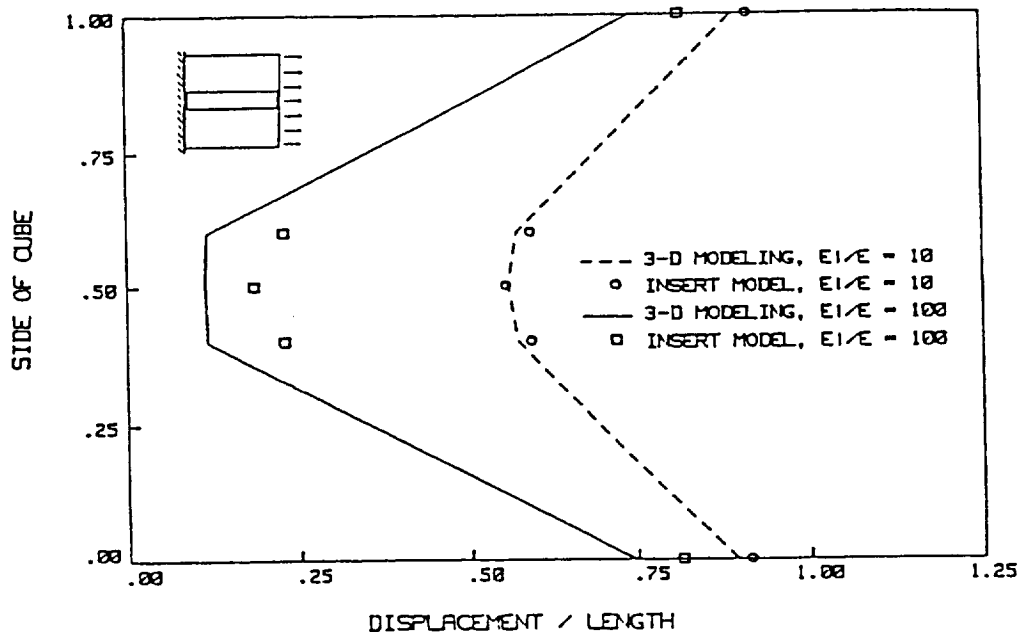
Fig. 7.2.3 Comparison of Displacement Profiles Between the Full
3-D Model and the Insert Element Model for the End of
a Cube in Tension
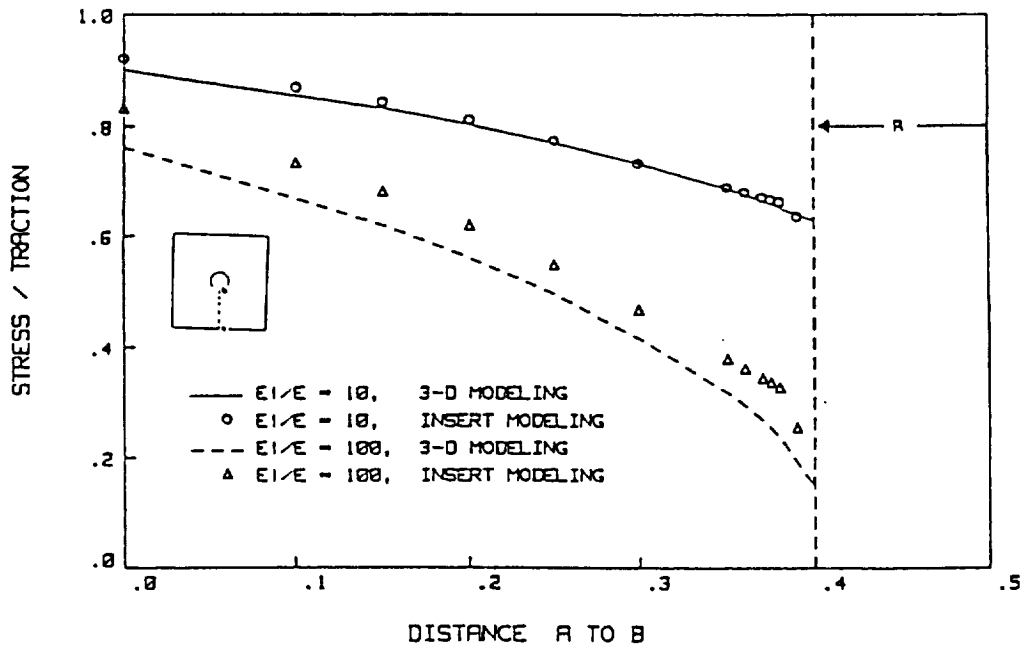
Fig. 7.2.4 Axial Stress Through the Cross Section of a Unit Cube
in Tension with a Single Insert in Parallel with the
Loading
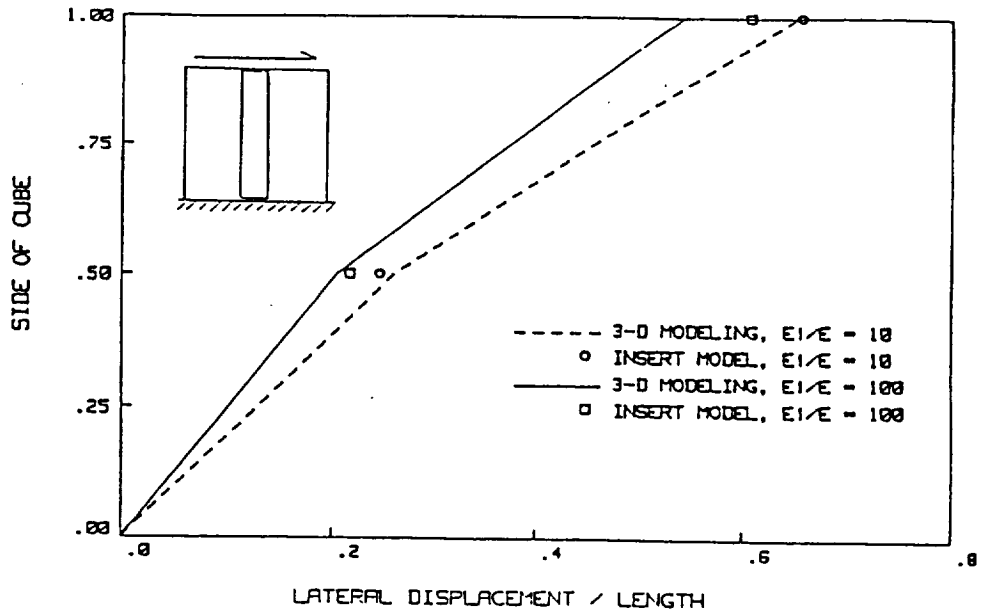
Fig. 7.2.5 Lateral Displacement Along a Side of a Cube Subjected to a Shear Force Perpendicular to an Insert
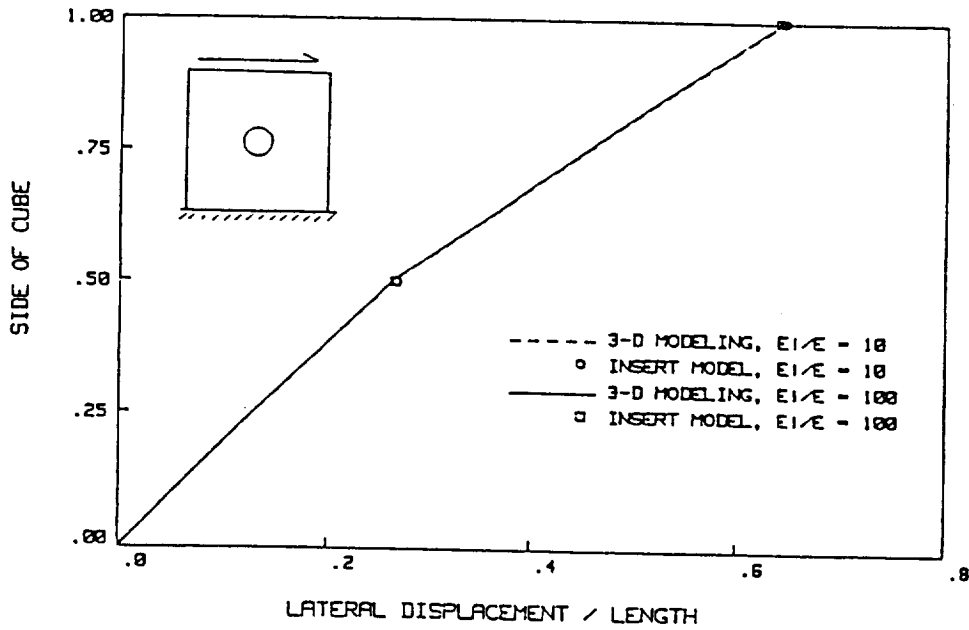
Fig. 7.2.6 Lateral Displacement Along a Side of a Cube Subjected to a Shear Force in the Cross-Plane of an Insert
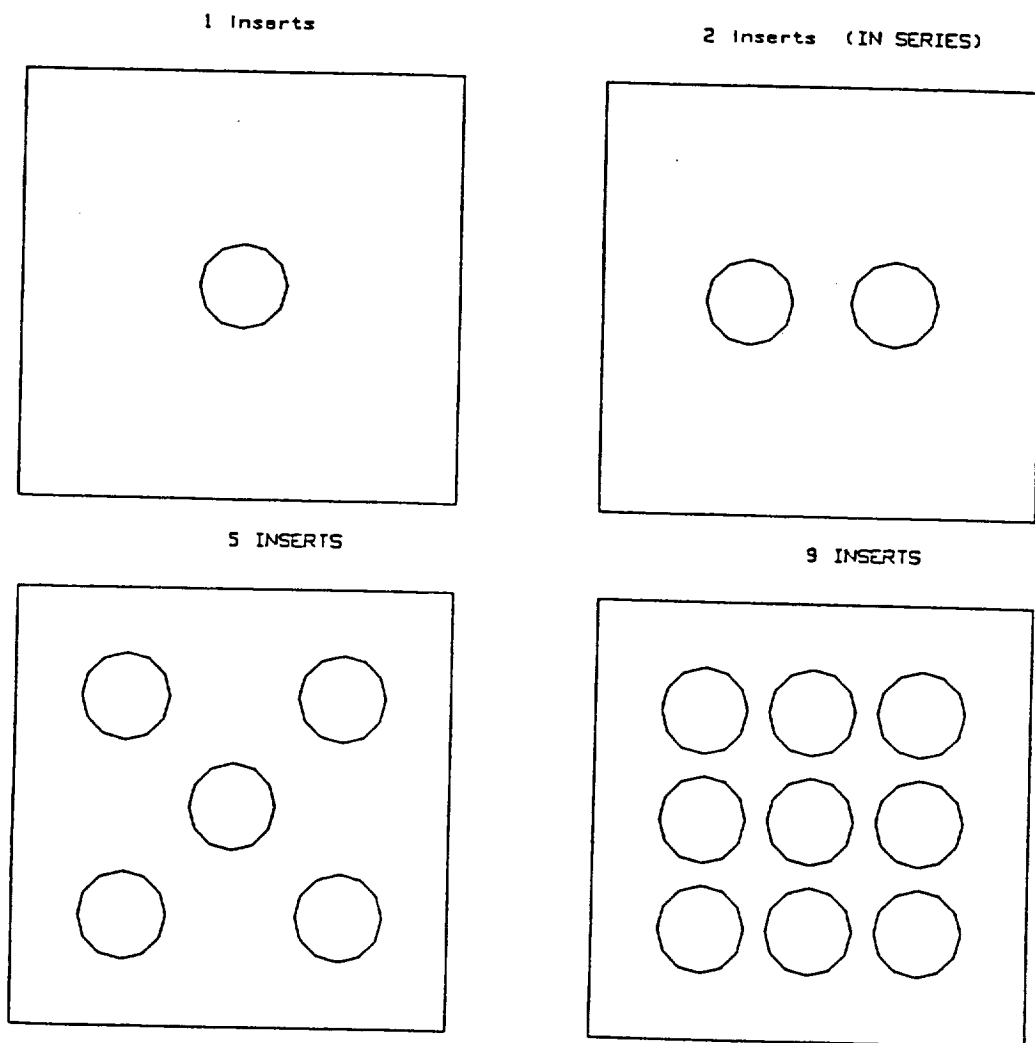
Fig. 7.3.1 Arrangement of Multiple Inserts in a Unit Cube
Subjected to Lateral Tension
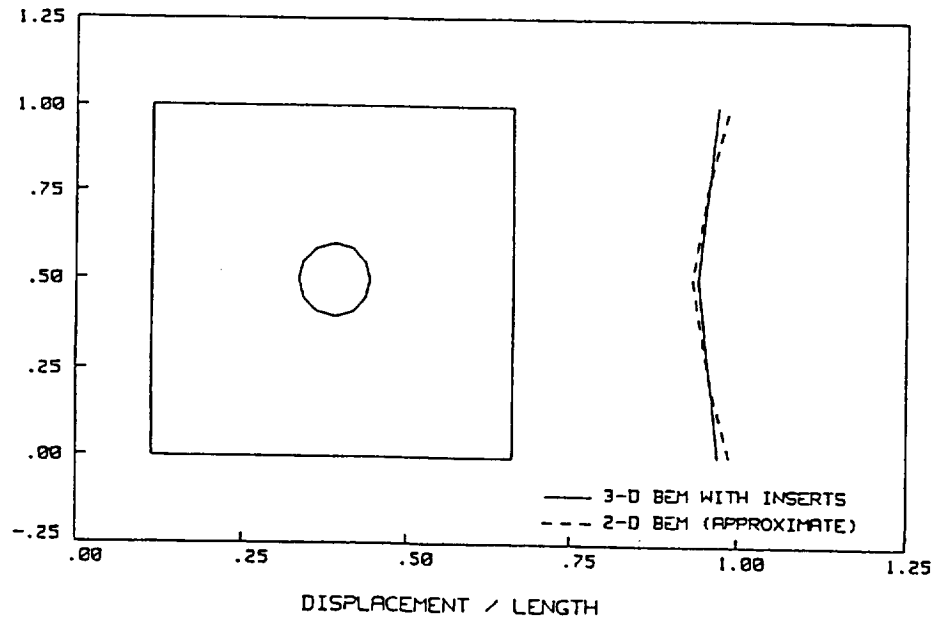
Fig. 7.3.2 Displacement Profile of a Cube with a Single Insert Under Lateral Tension
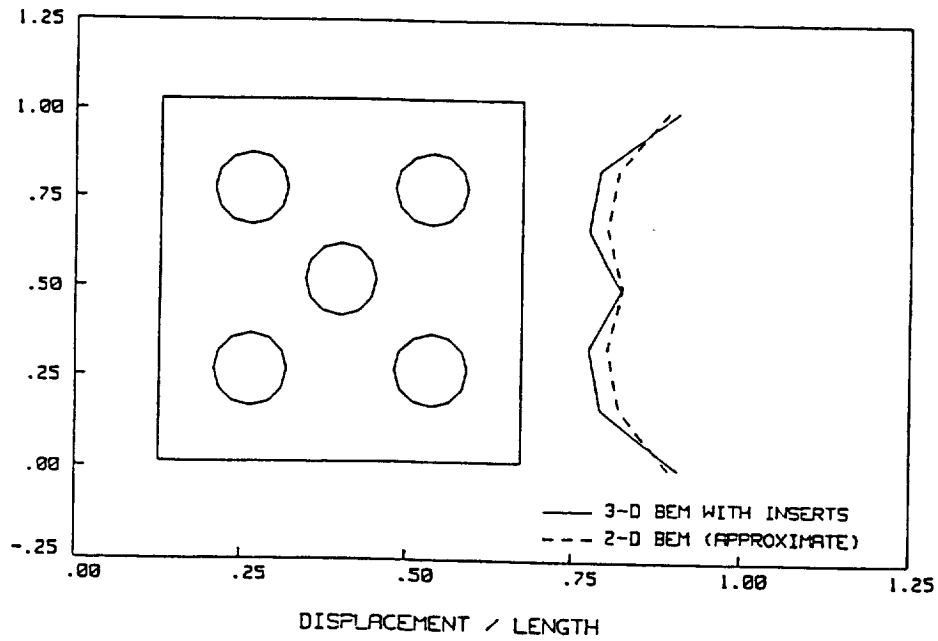
Fig. 7.3.3 Displacement Profile of a Cube with Five Inserts Under Lateral Tension
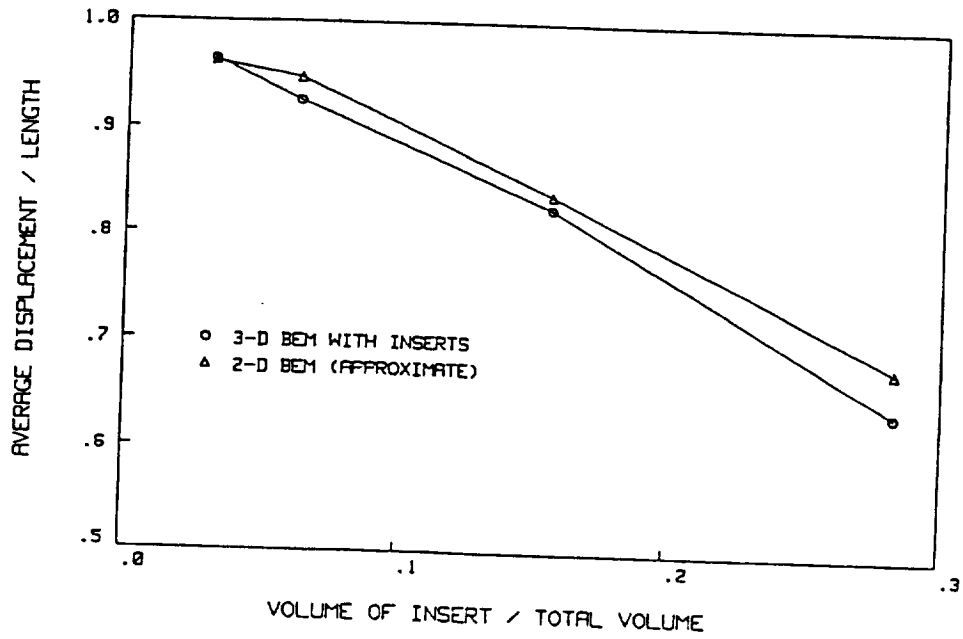
Fig. 7.3.4 Average End Displacement of a Cube Under Tension vs. the Volume of Insert to Total Volume Ratio
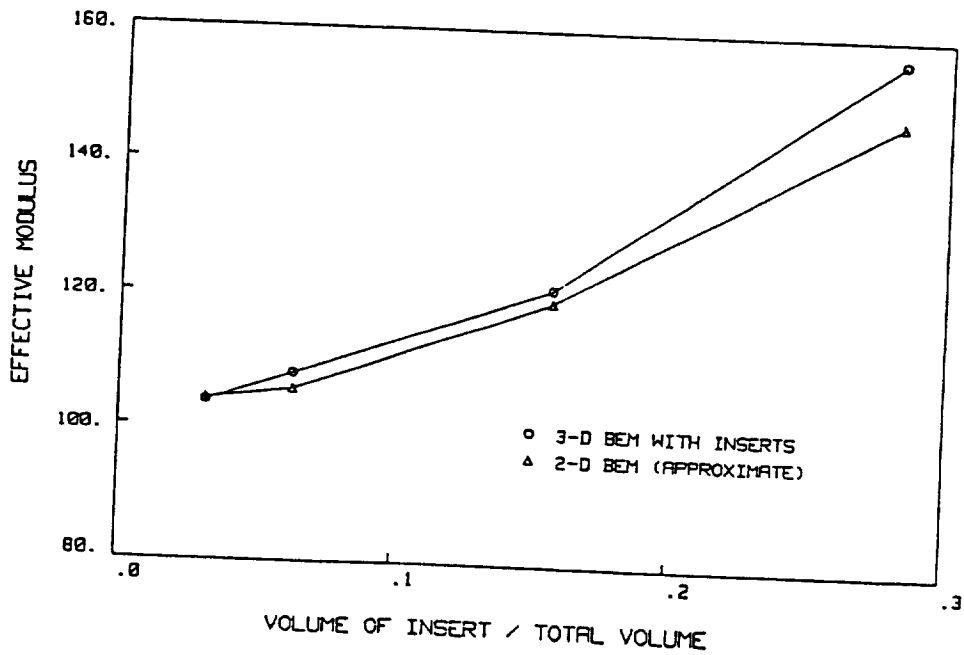
Fig. 7.3.5 Effective Transverse Modulus of a Cube as a Function of Insert Volume to Total Volume
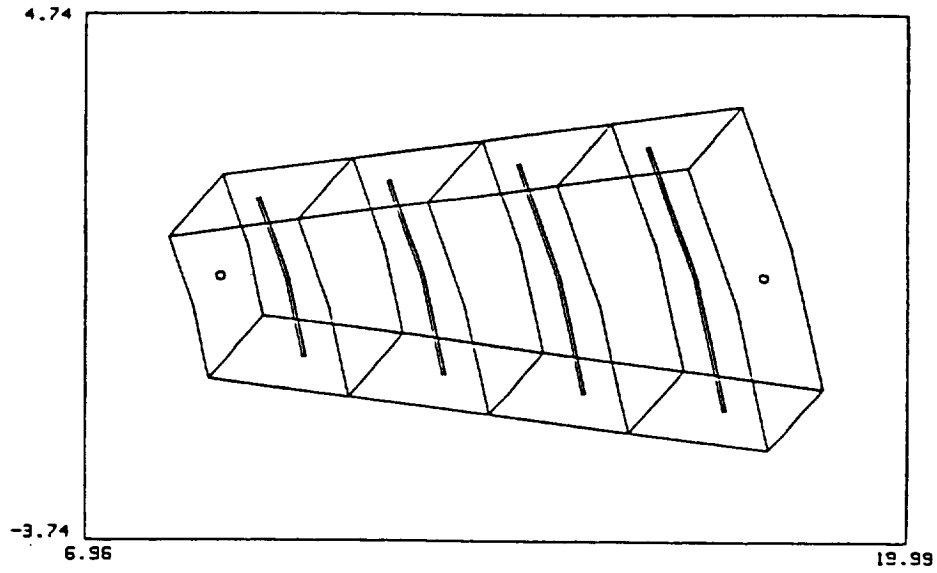
Fig. 7.4.1   Discretization of a Thick Cylinder with Four Inserts
             Utilizing Insert Elements

+ Element edge
o Mid-point
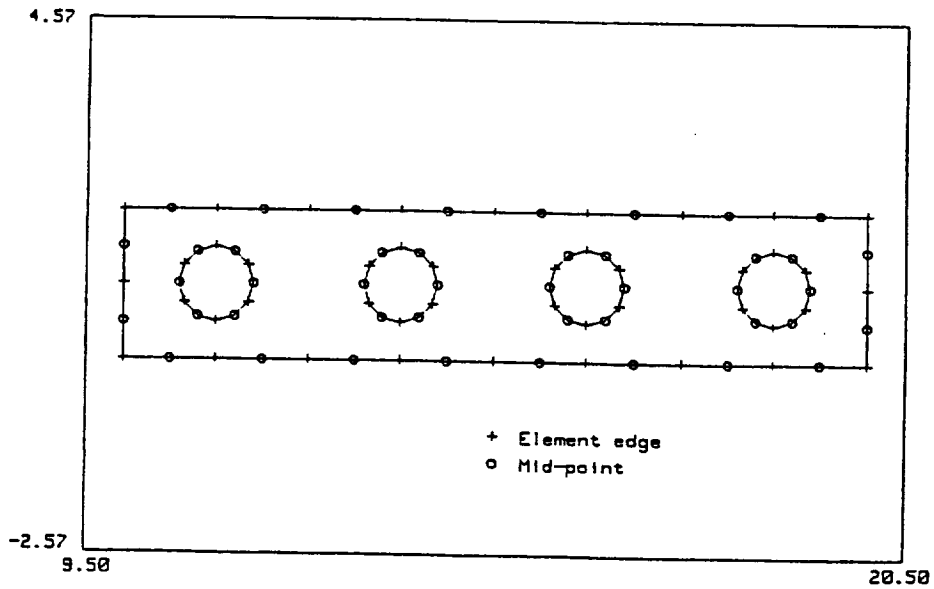
Fig. 7.4.2   Axisymmetric Multi-region, Discretization of a
             Thick Cylinder with Four Inserts
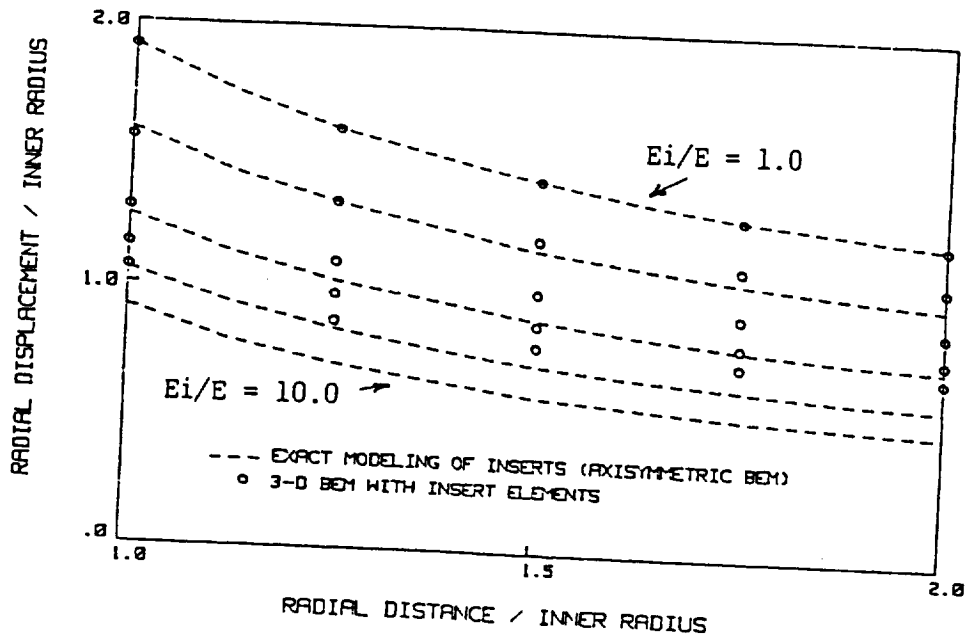
Fig. 7.4.3 Radial Displacement Through a Pressurized Thick
Cylinder with Circumferential Inserts for Ei/E = 1.0,
2.5, 5.0, 7.5, 10.0
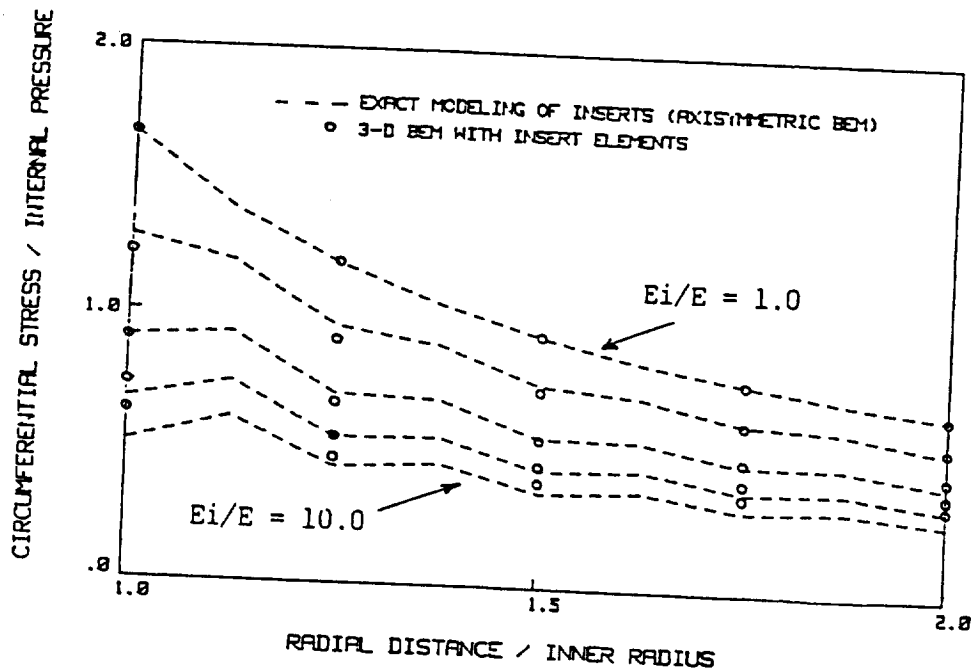
Fig. 7.4.4 Circumferential Stress Through a Pressurized Thick
Cylinder with Circumferential Inserts for Ei/E = 1.0,
2.5, 5.0, 7.5, 10.0
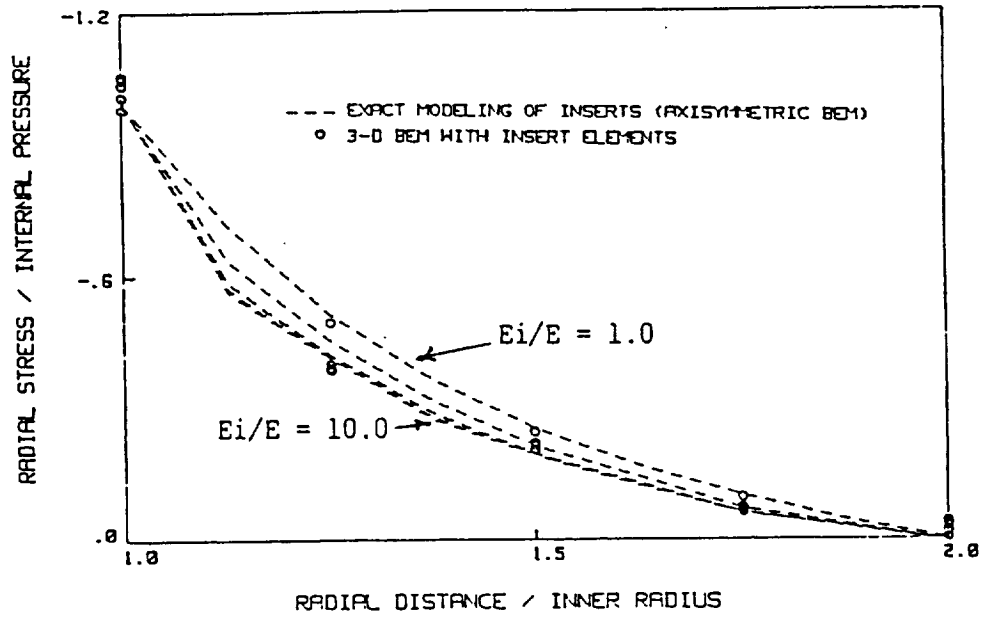
Fig. 7.4.5   Radial Stress Through a Pressurized Thick Cylinder
with Circumferential Inserts for Ei/E = 1.0, 2.5,
5.0, 7.5, 10.0
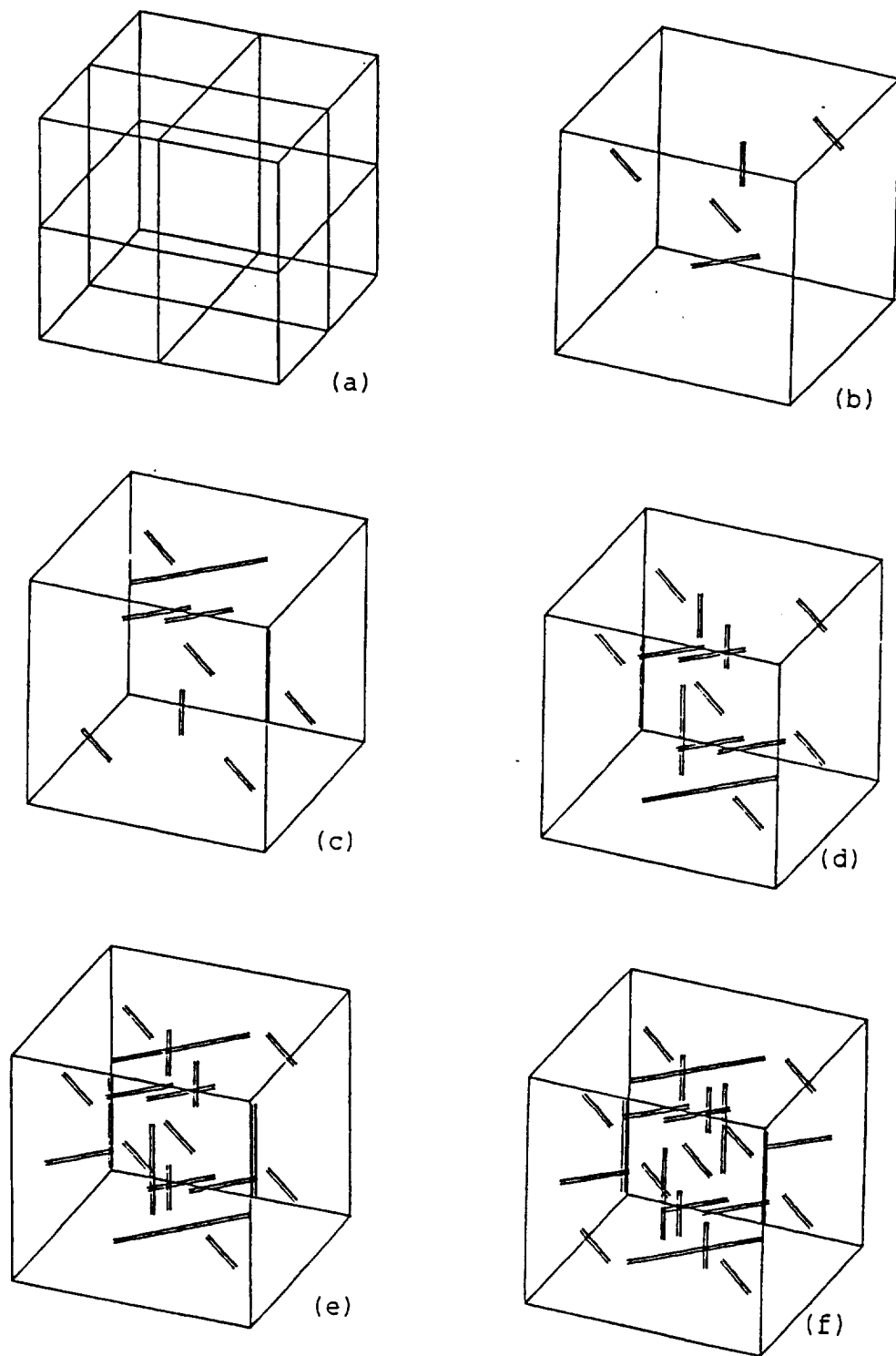
Fig. 7.5.1 (a) Surface Discretization of a Unit Cube Used in the Study of Random Oriented Inserts, (b-f) Orientation of Variable Length Inserts Within Unit Cubes Containing 5, 10, 15, 20, and 25 Inserts, respectively
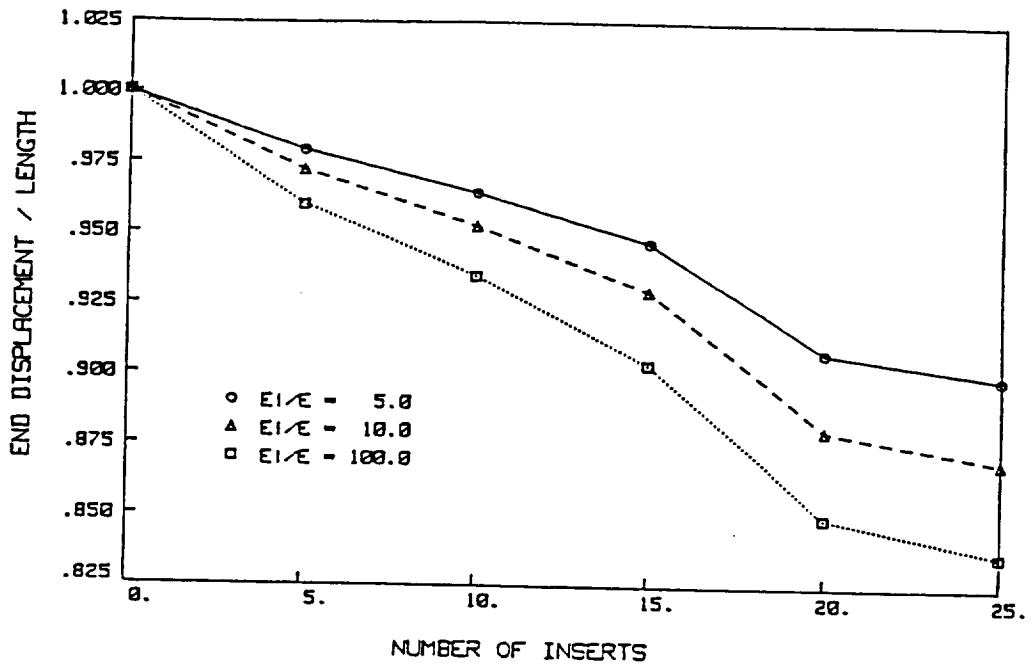
Fig. 7.5.2 End Displacement of a Unit Cube With Random Oriented
Inserts of Ei/E = 5.0, 10.0 and 100.0
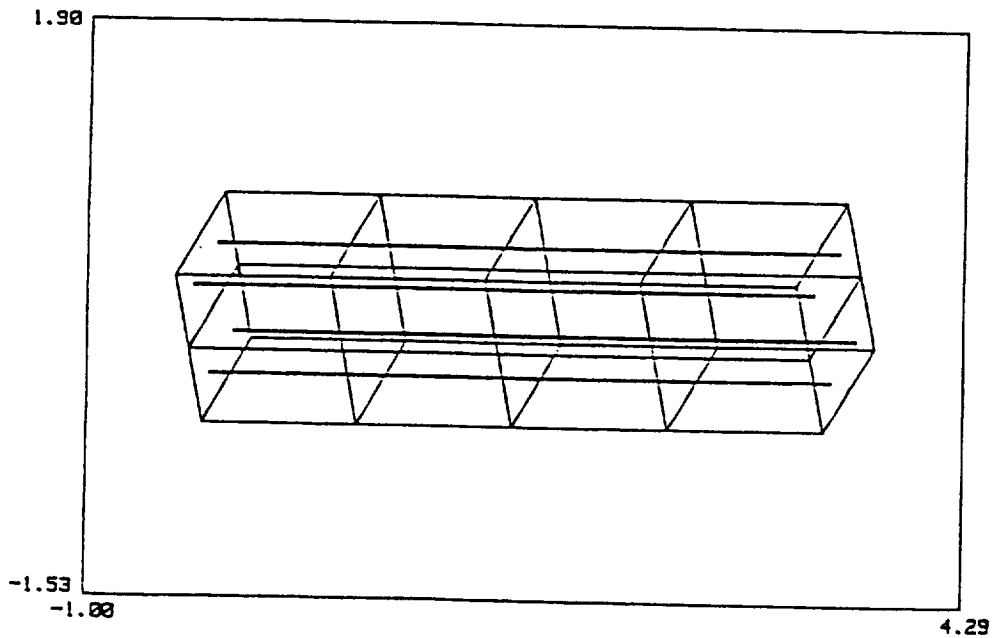
Fig. 7.6.1 Discretization of a Reinforced Beam Utilizing
Quadratic Insert Elements to Model the Four Inserts
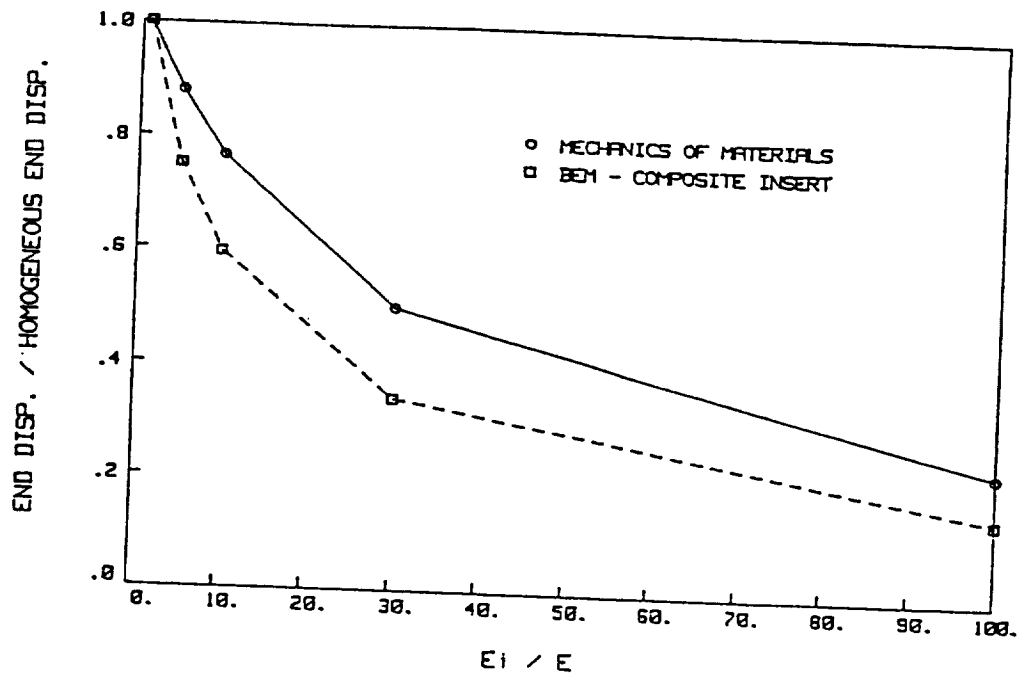
Fig. 7.6.2 Non-dimensionalized Vertical End Displacement of a
Reinforced Beam in Bending vs. the Modulus of the
Insert Over Modulus of the Beam
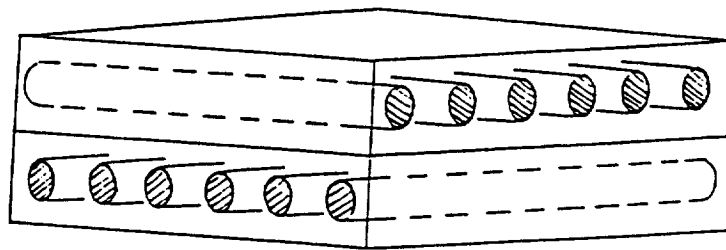
Fig. 7.7.1 Laminate-Fiber Composite
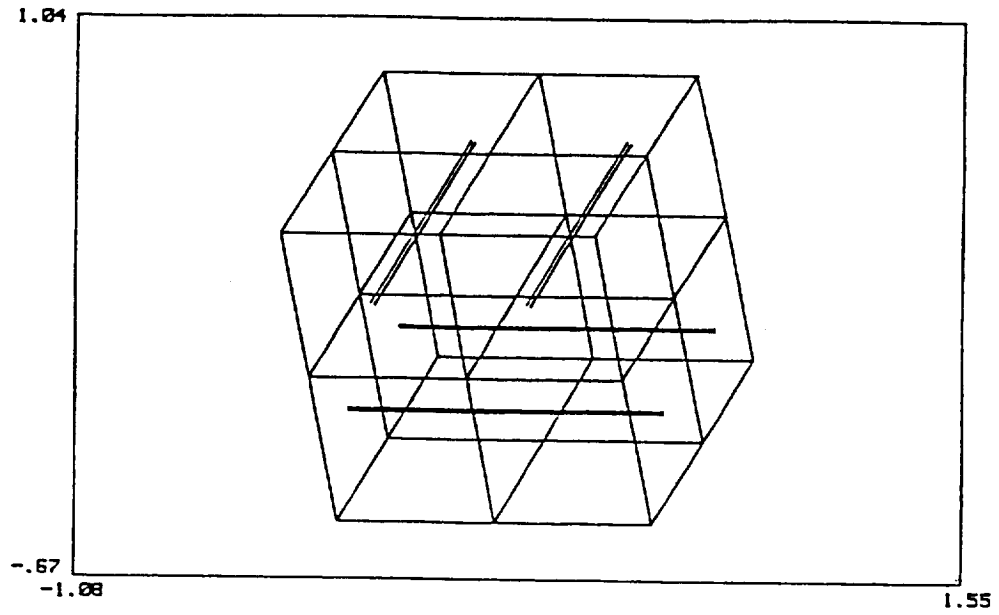
Fig. 7.7.2 BEM Discretization of a Laminate-Fiber Composite
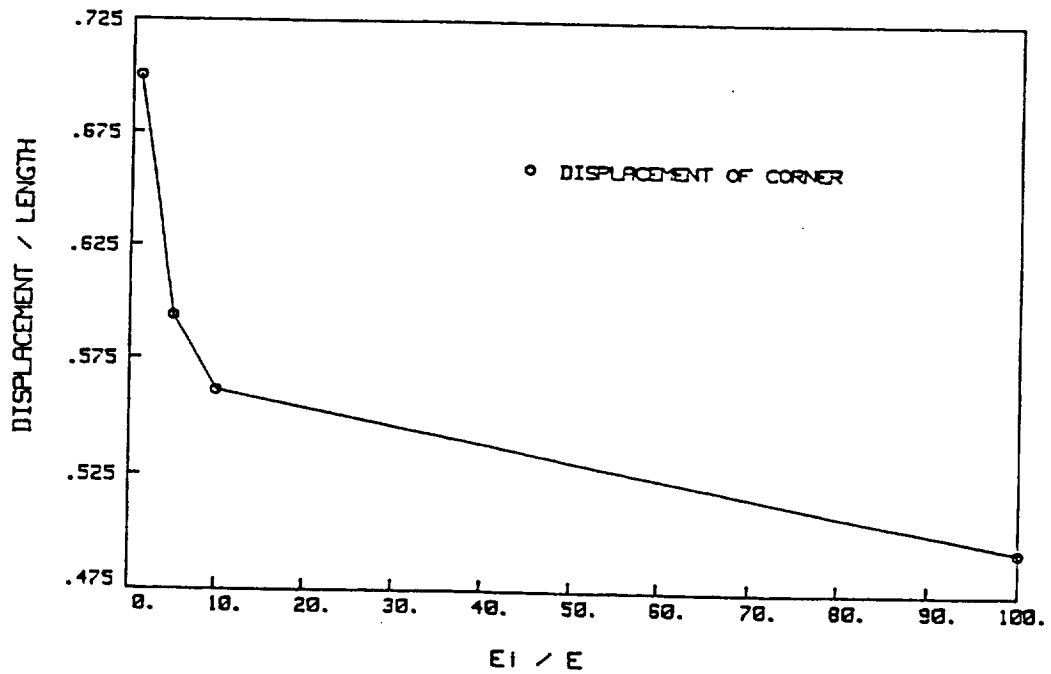
Fig. 7.7.3 Lateral Displacement of a Point at the Corner of
the Interface of a Laminate-Fiber Composite Under
Bi-axial Tension
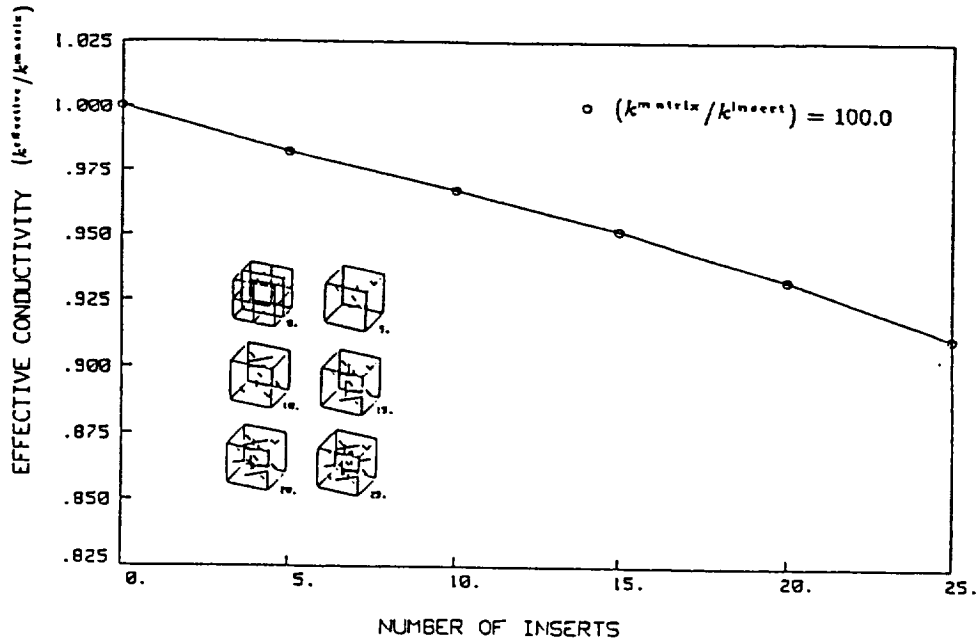
Fig. 7.8.1. Effective Conductivity of a Unit Cube with
Random Oriented Inserts
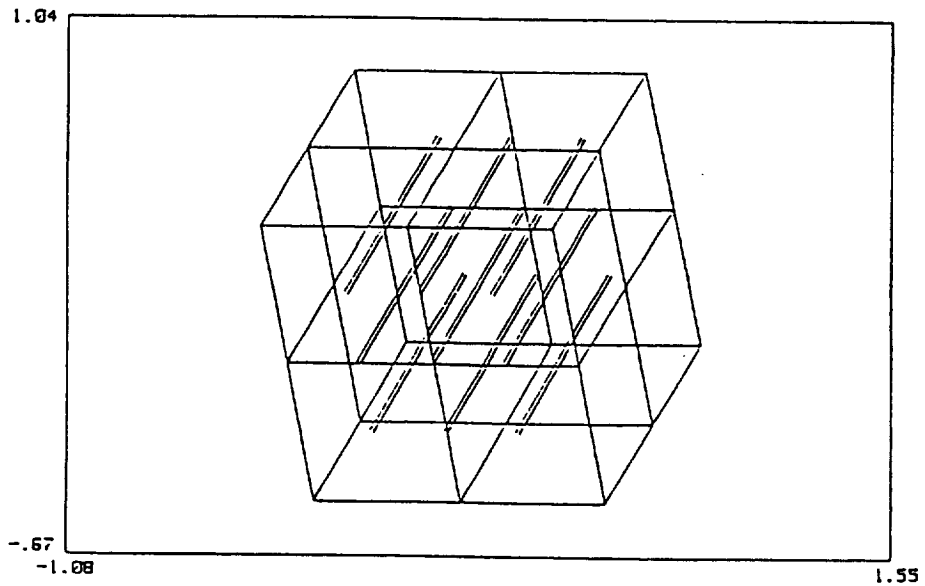
Fig. 7.9.1. Discretization of a Cube with Nine Quadratic
Insert Elements
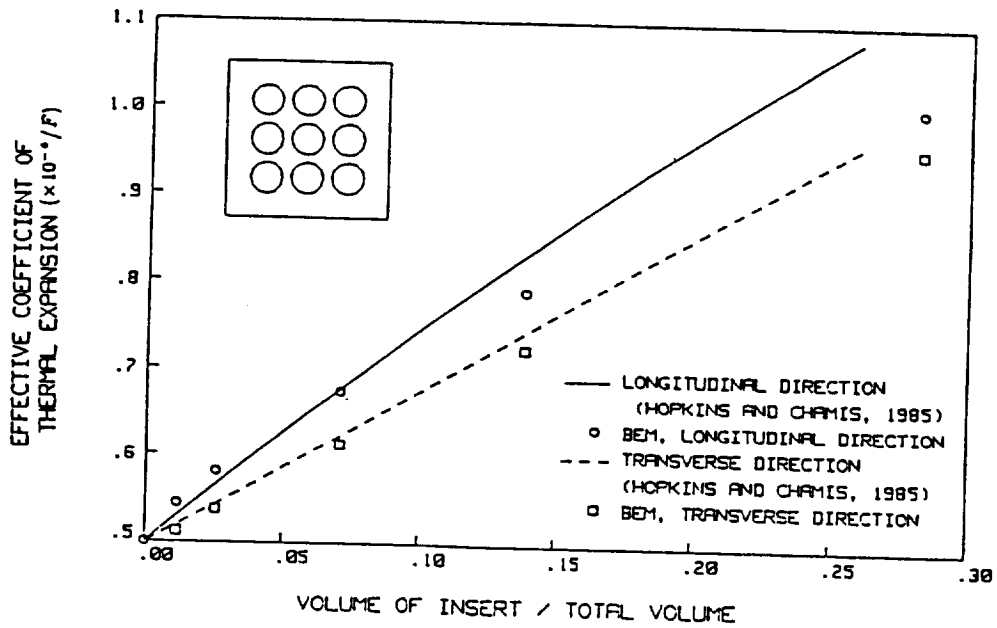
Fig. 7.9.2   Effective Coefficient of Thermal Expansion
in a Cube with Nine Perfectly-bonded Insert Fibers
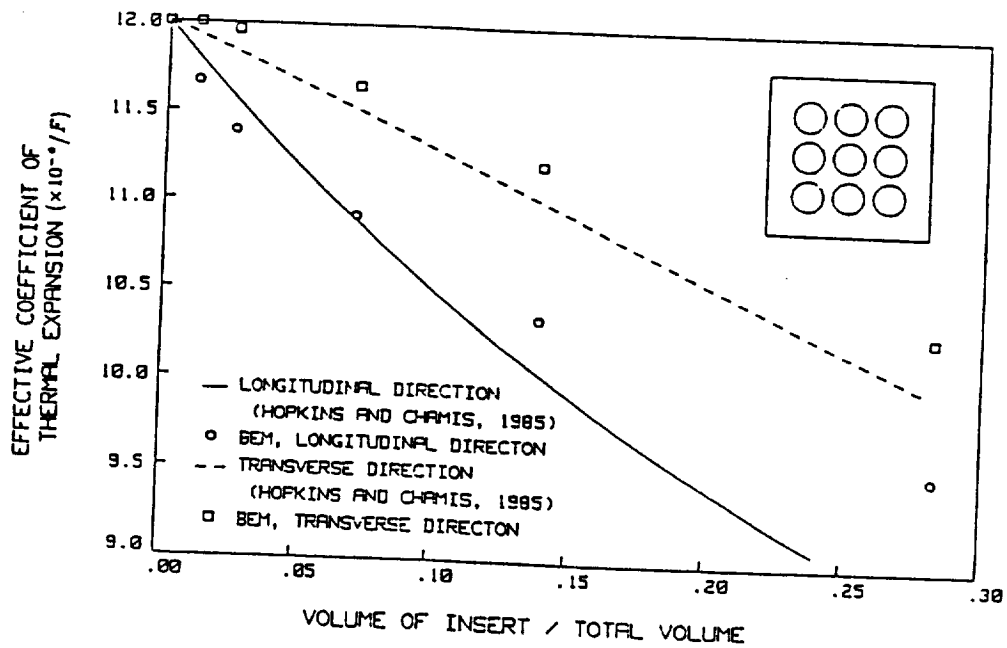
Fig. 7.9.3   Effective Coefficient of Thermal Expansion
in a Model of an Aluminum Cube with Nine
Perfectly-bonded Steel Inserts
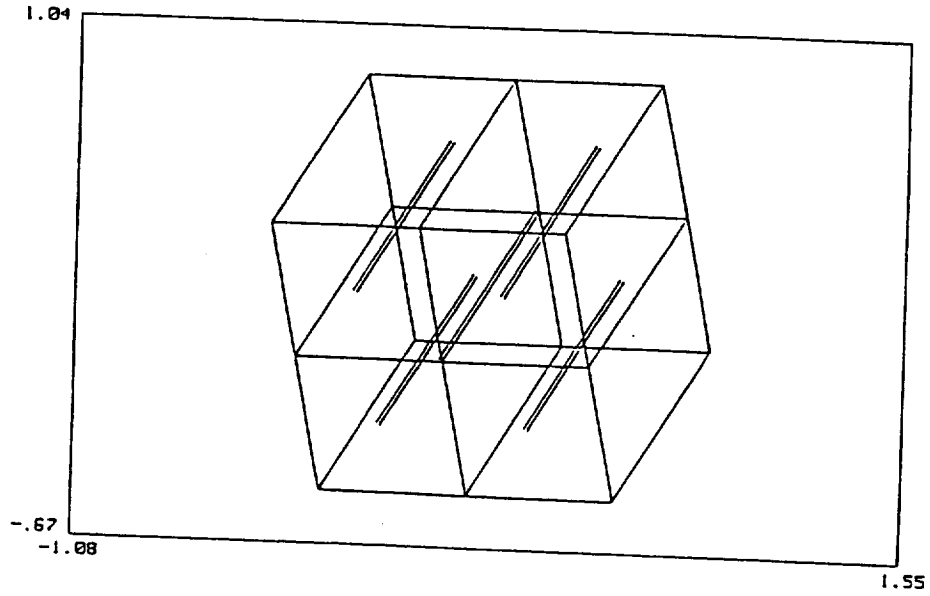
Fig. 7 .10.1. Discretization of a Cube with Five Inserts
Connected to the Matrix Through a Linear
Constitutive Model
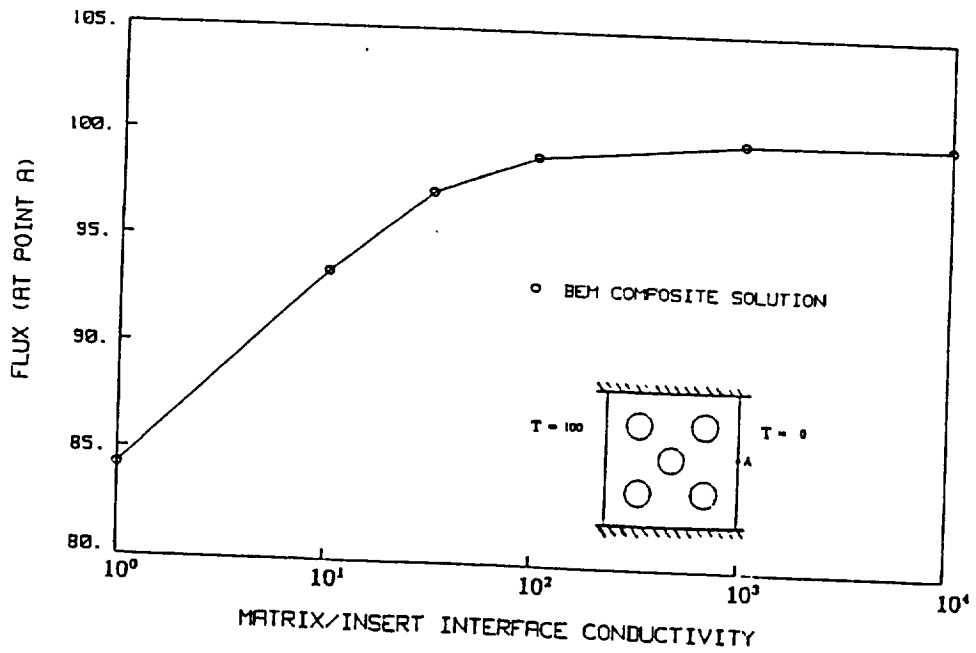
Fig. 7.10.2   Effect of a Thermal Resistant Matrix/Insert
Interface on the Flux Through a Fiber
Composite
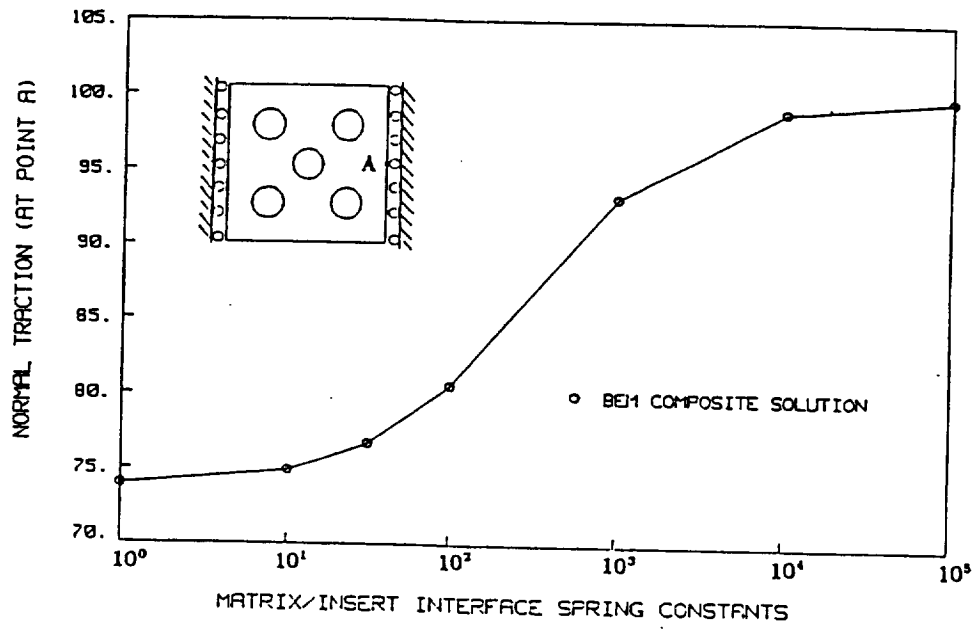
Fig. 7.11.1   Effect of a Spring Interface Between the
Matrix and Insert in a Cube Subjected to a
Uniform Temperature Decrease
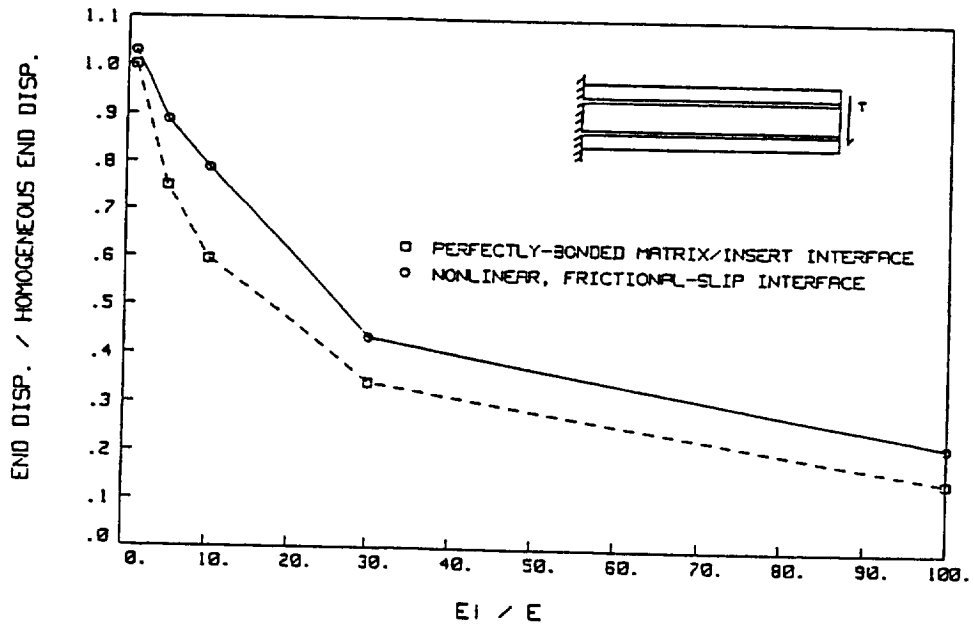
Fig. 7.12.1   Vertical End Displacement of a Beam in
Bending with a Nonlinear, Frictional
Interface Between Four Insert Fibers and
the Matrix
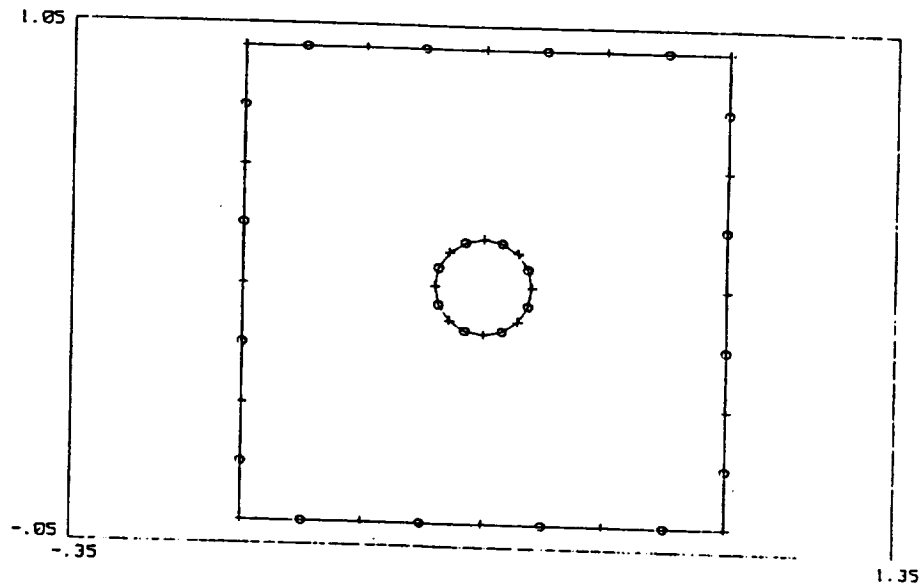
Fig. 7.13.1. Two-dimensional, Multi-region BEM Model of
a Cube with a Single Insert.
(Insert Volume/Total Volume = 0.031)

POISSON RATIO OF INSERT / POISSON RATIO OF MATRIX
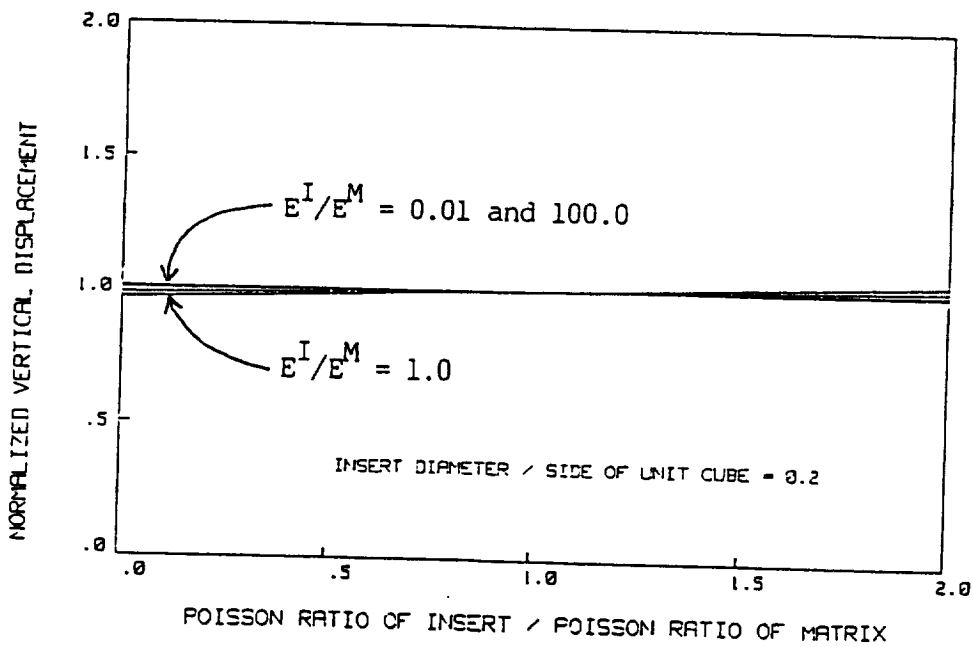
Fig. 7.13.2. Effect of Poisson Ratio on the Lateral
Displacement of a Single-fiber Composite in
Tension for Values of E^I/E^M
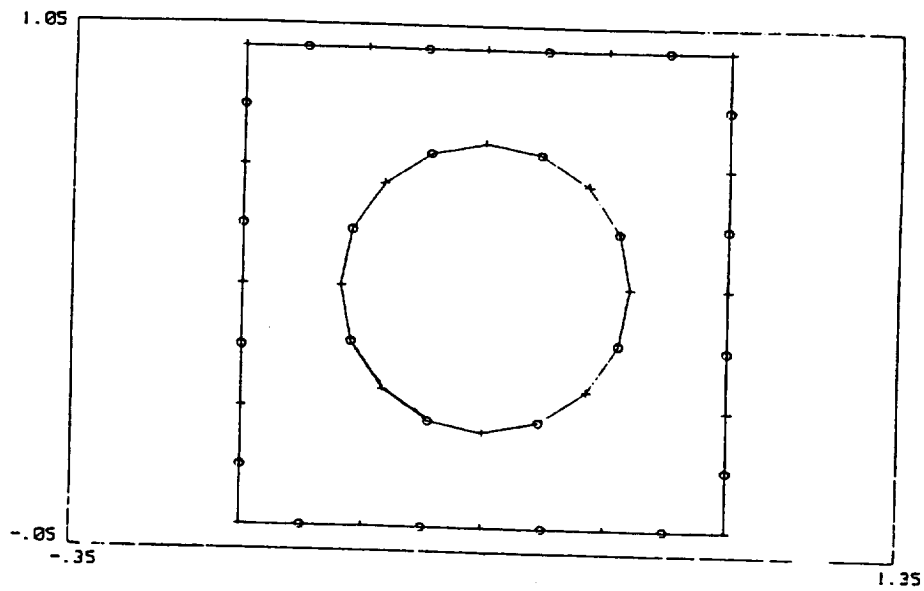
Fig. 7.13.3. Two-dimensional, Multi-region BEM Model of
a Cube with a Single Insert
(Insert Volume/Total Volume = 0.283)
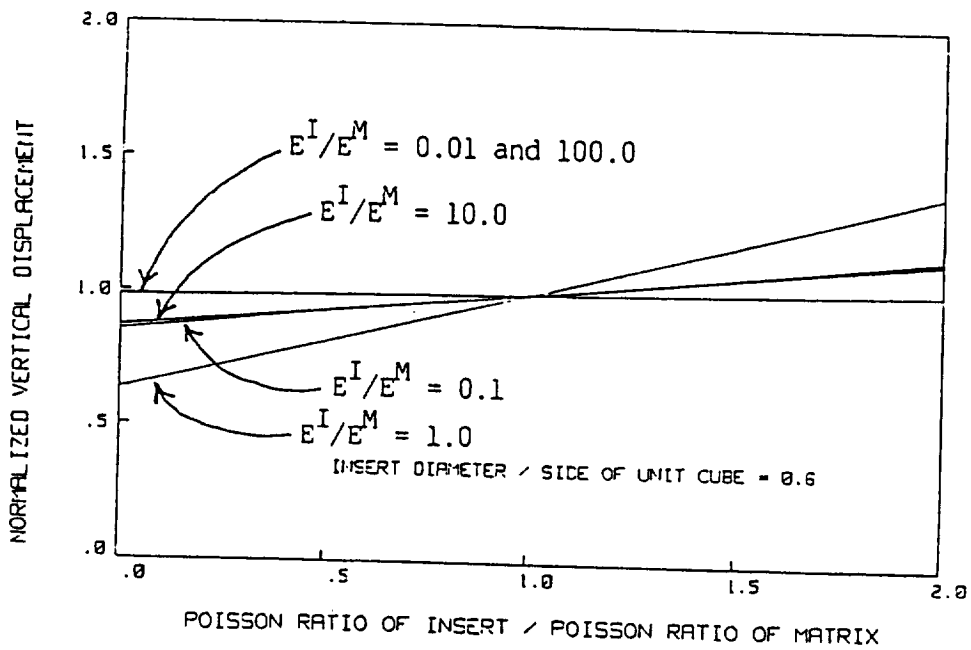
Fig. 7.13.4. Effect of Poisson Ratio on the Lateral
Displacement of a Single-fiber Composite in
Tension for values of E^I/E^M of 0.01, 0.1, 1.0,
10.0, and 100.0
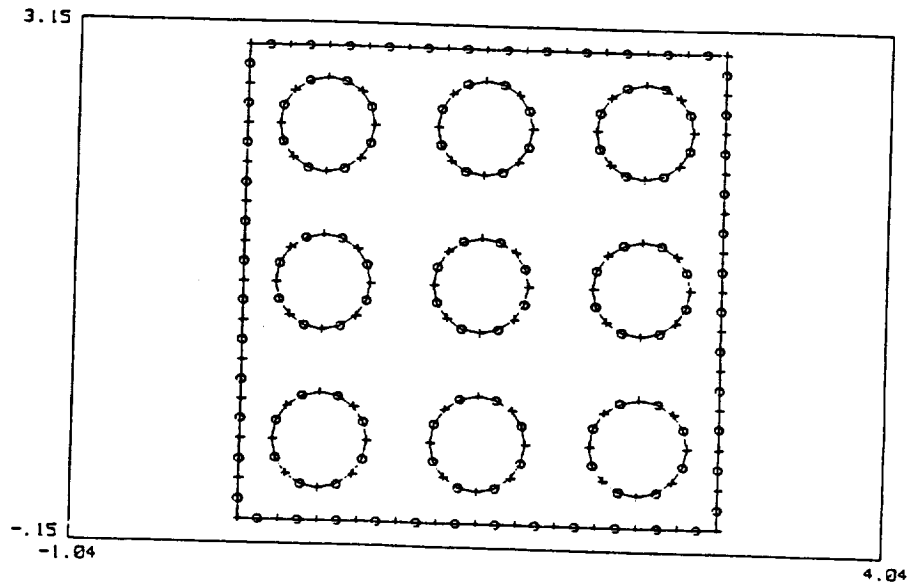
Fig. 7.13.5. Two-dimensional, Ten Region BEM Model of a
Cube with Nine Inserts
(Insert Volume/Total Volume = 0.283)

$E^I/E^M = 1.0$

NORMALIZED VERTICAL DISPLACEMENT

- - - - - SINGLE FIBER
——— MULTIPLE FIBERS

INSERT DIAMETER / SIDE OF UNIT CUBE = 0.6

POISSON RATIO OF INSERT / POISSON RATIO OF MATRIX
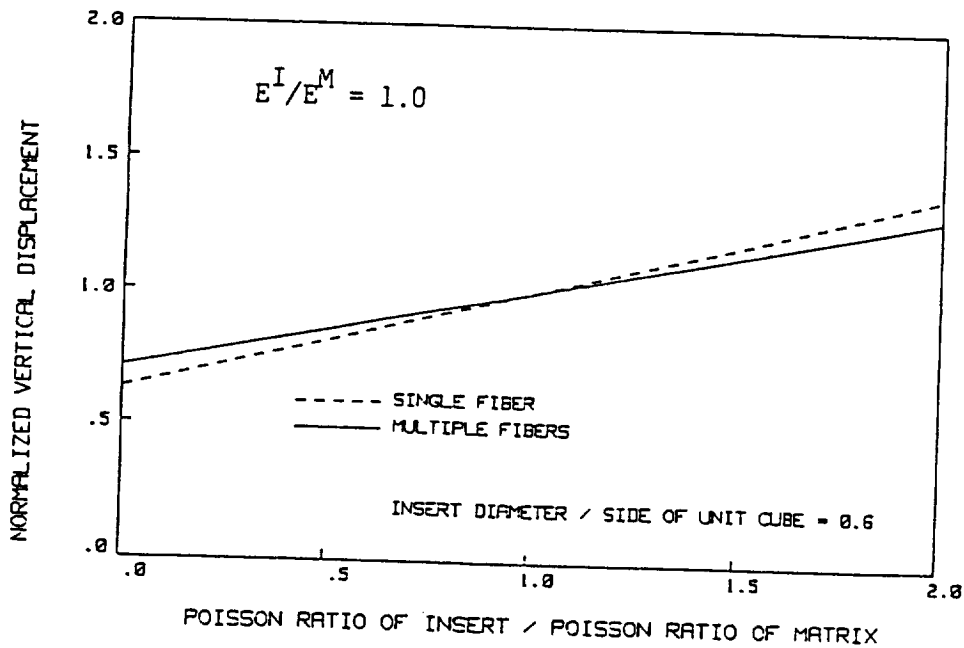
Fig. 7.13.6. Effect of Poisson Ratio on the Lateral
Displacement of a Single-fiber Versus a
Multi-fiber Composite in Tension with
$E^I/E^M = 1.0$
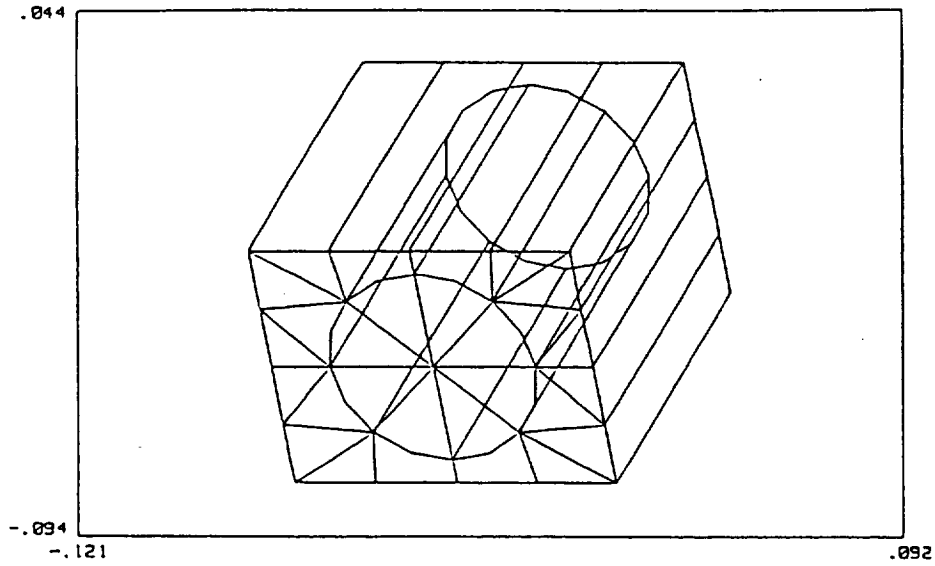
Fig. 7.14.1   Discretization of the Outer Surface of a
                Two-region Model Containing 50 Small Insert Fibers
                (Small Insert Fibers not shown)
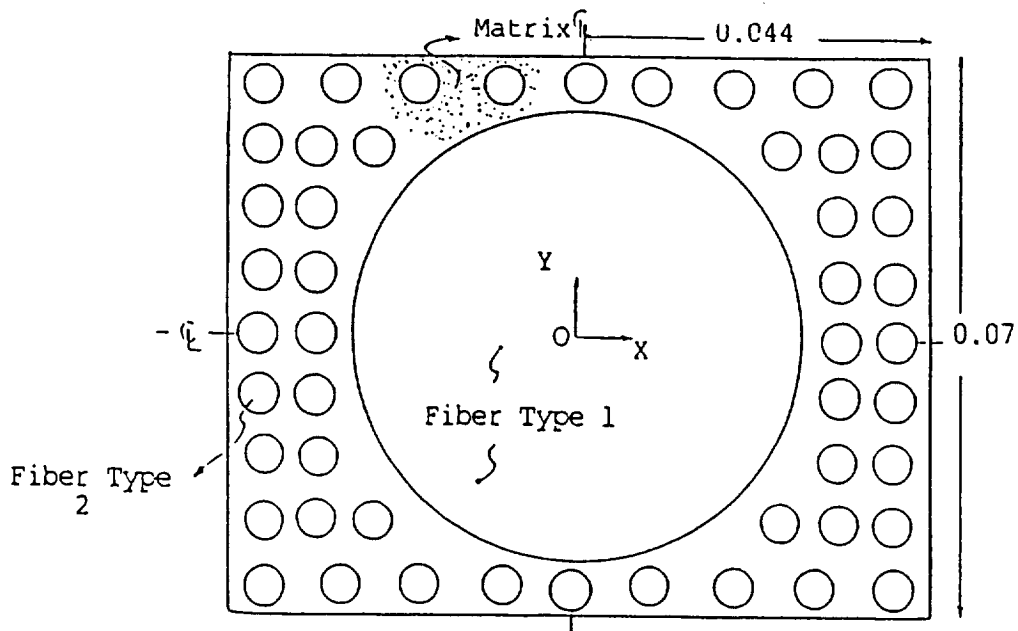
Fig. 7.14.2   Cross-section   of the Composite Displaying
                the Arrangement of One Large (Type 1) Fiber
                and Fifty Small (Type 2) Fibers
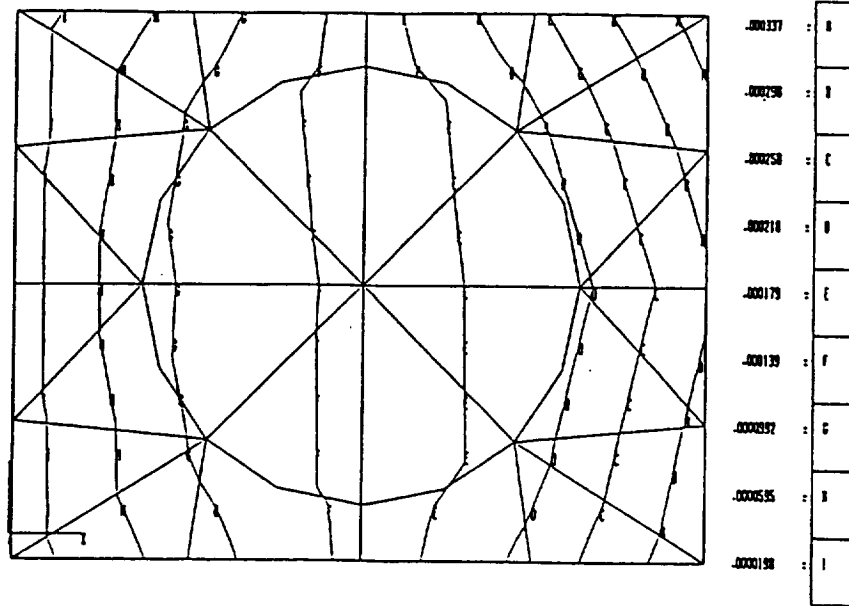
Fig. 7.14.3. Contours of Displacement in the Horizontal
Direction Relative to the Page Due to a
(Horizontal) Traction Loading
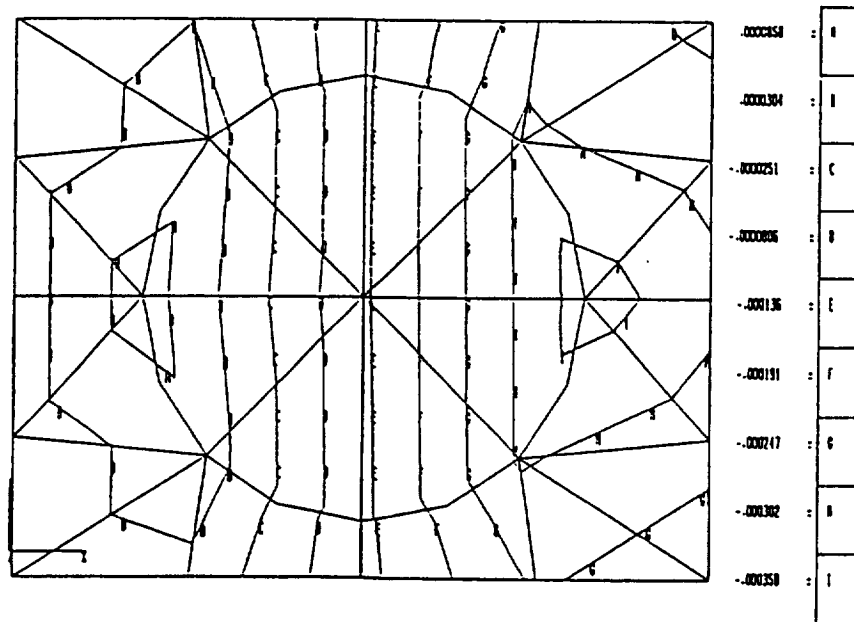
Fig. 7.14.4 Contours of Displacement in the Horizontal
Direction Relative to the Page Due to a
(Horizontal) Traction and Temperature Loading
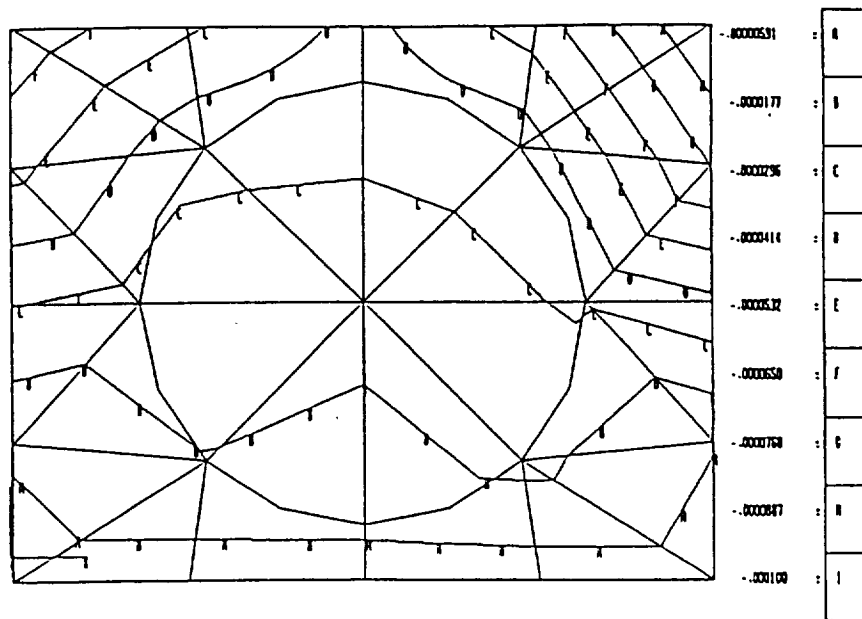
Fig. 7.14.5. Contours of Displacement in the Vertical
Direction Relative to the Page Due to a
(Horizontal) Traction Loading
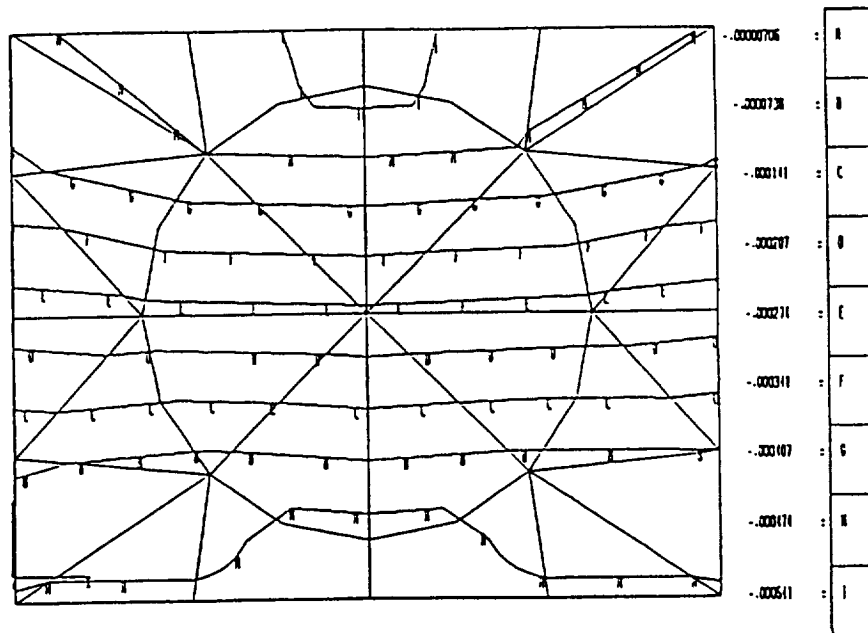
Fig. 7.14.6. Contours of Displacement in the Vertical
Direction Relative to the Page Due to a
(Horizontal) Traction and Temperature Loading
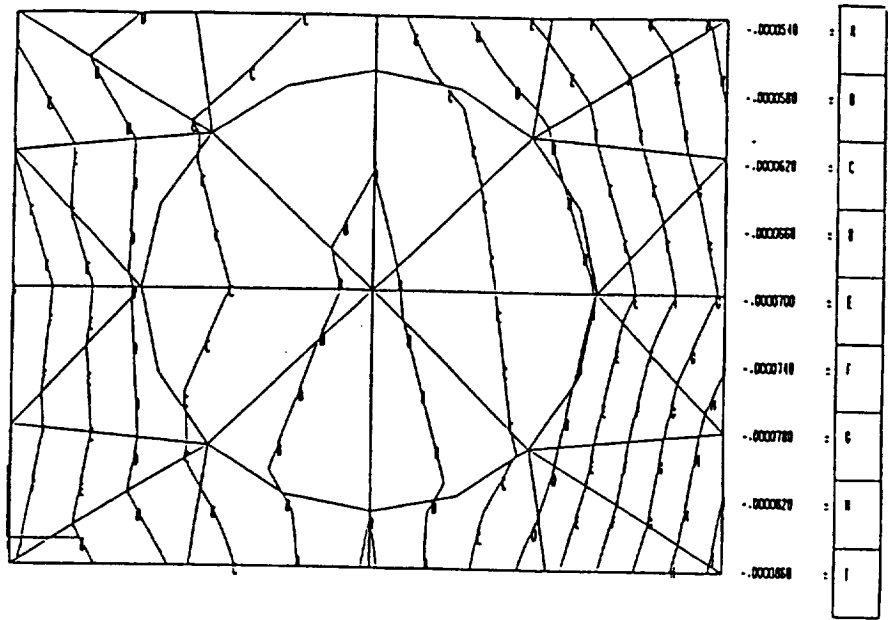
Fig. 7.14.7.   Contours of Displacement in the Out of Plane
Direction Due to a (Horizontal) Traction Loading
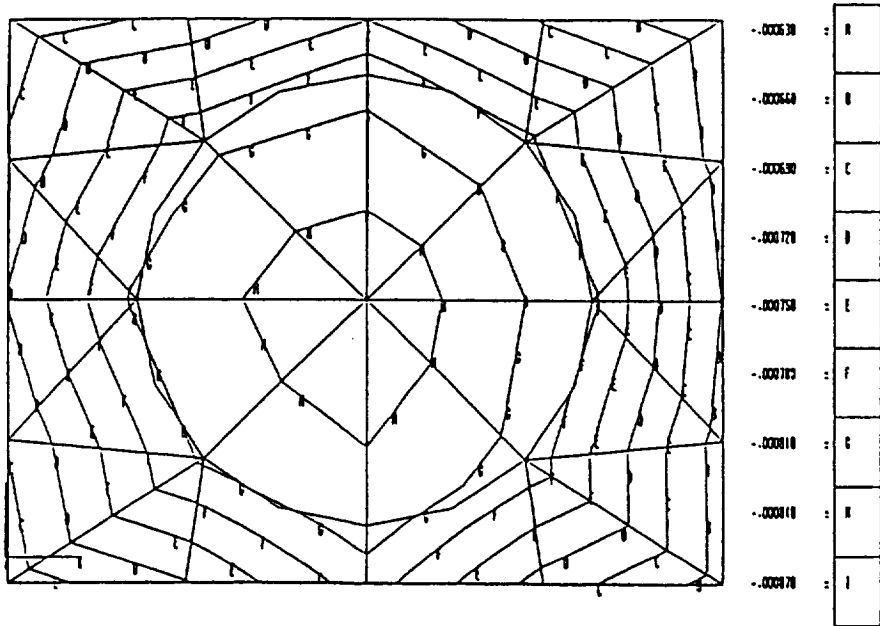
Fig. 7.14.8.   Contours of Displacement in the Out of Plane
Direction Due to a (Horizontal) Traction and
Temperature Loading
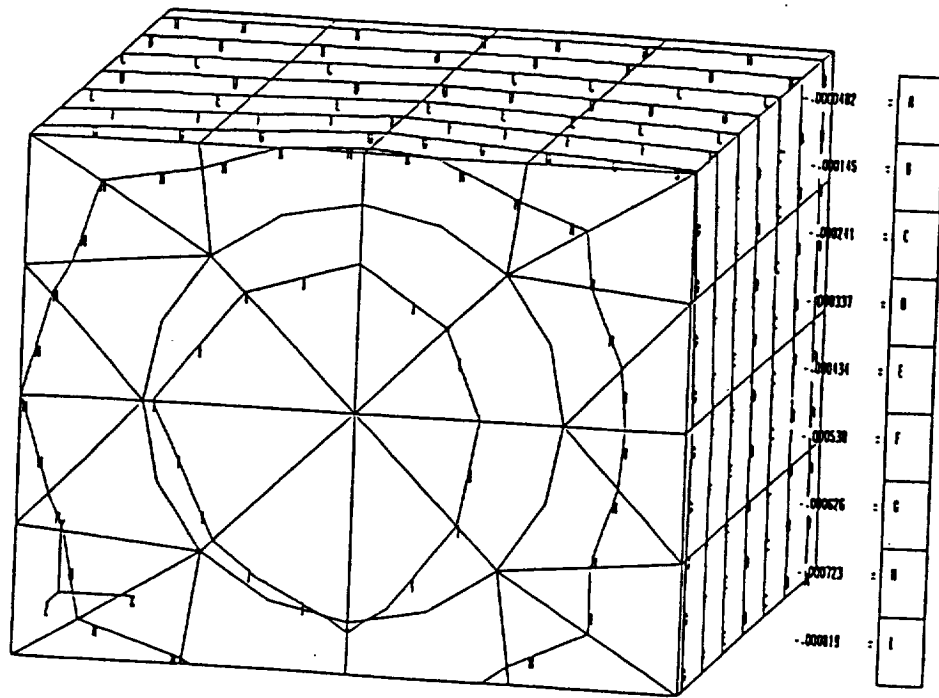
Fig. 7.14.9   Three-dimensional View: Contours of Displacement in
the Out-of-Plane (z) Direction Due to a Traction
(x-direction) and Temperature Loading
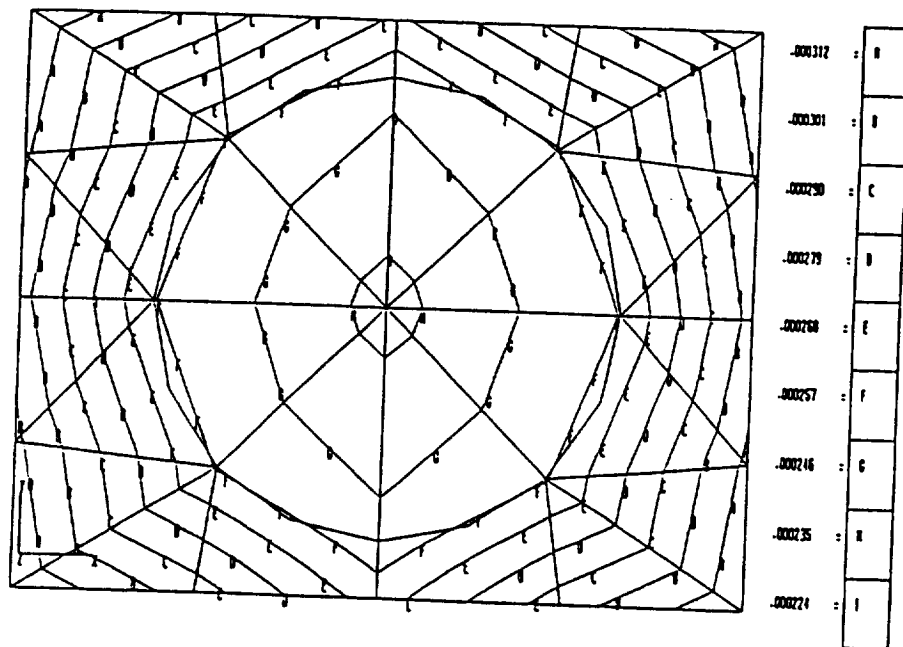
Fig. 7.14.10. Contours of Displacement in the Out-of-Plane
Direction Due to a Traction in the Out-of-Plane
Direction (Symmetric Boundary Conditions)
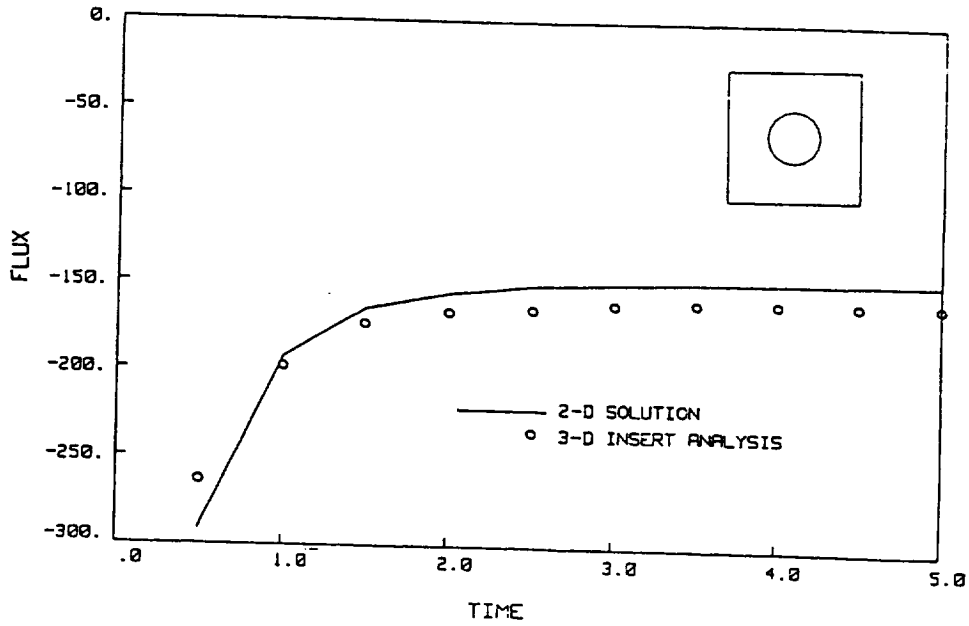
Fig. 7.15.1   Flux vs. Time: Transient Thermoelastic Analysis
of A Cube with One Insert
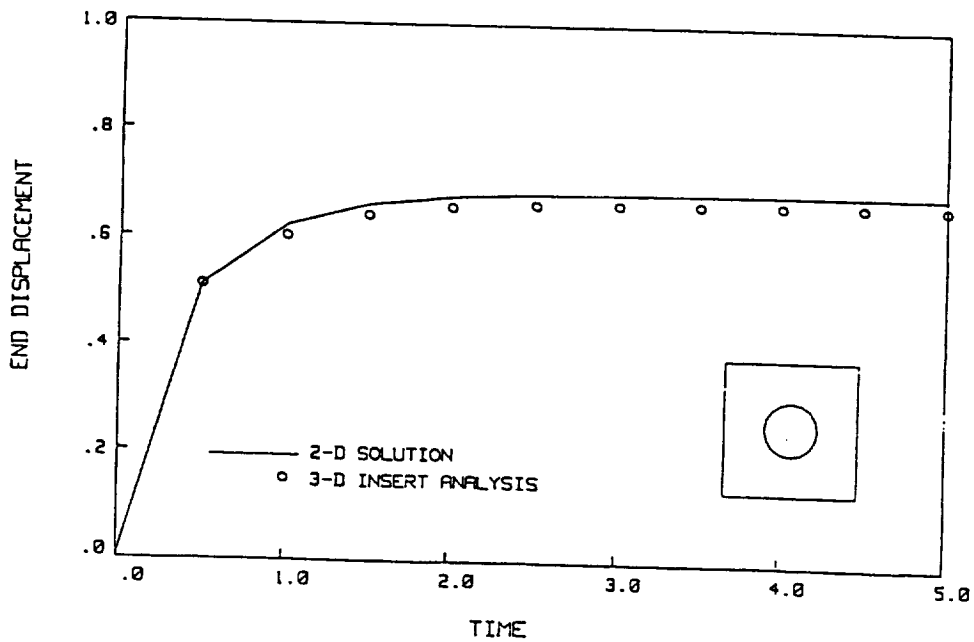
Fig. 7.15.2   End Displacement vs. Time: Transient
Thermoelastic Analysis of a Cube with
One Insert
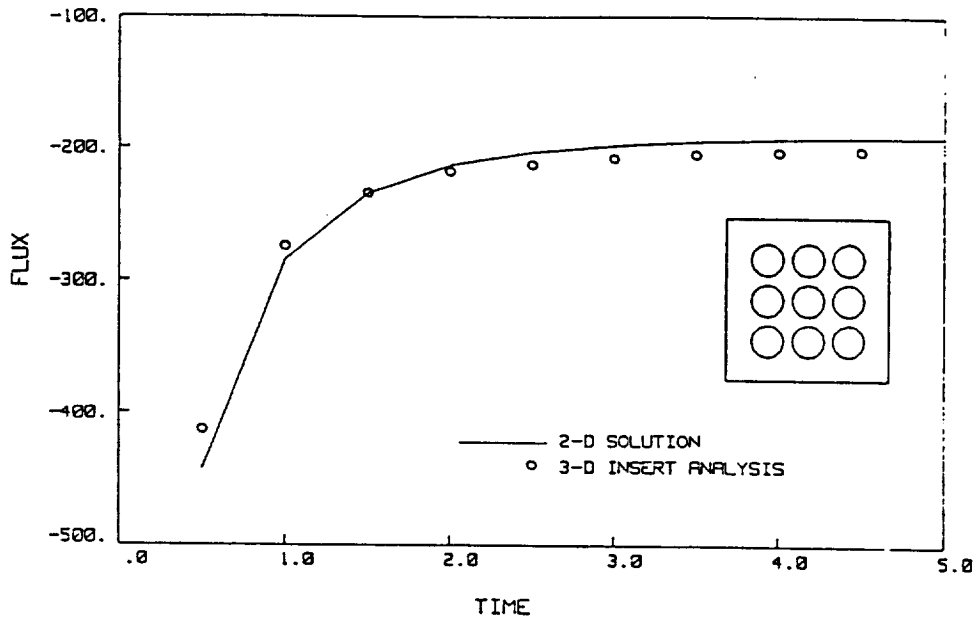
Fig. 7.15.3 Flux vs. Time: Transient Thermoelastic Analysis
of a Cube with Nine Inserts
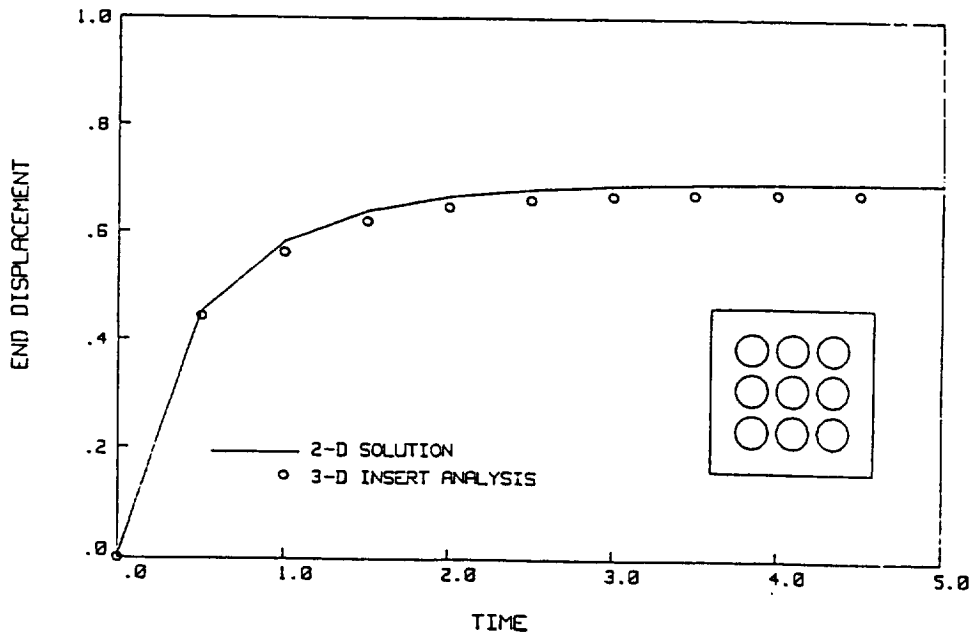
Fig. 7.15.4 End Displacement vs. Time: Transient Thermoelastic
Analysis of a Cube with Nine Inserts
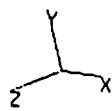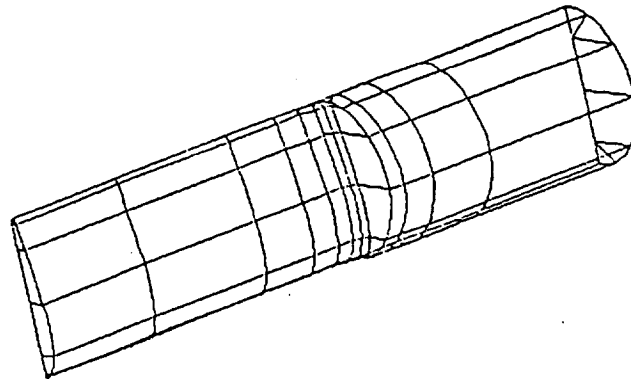
DISCRETIZATION OF A TURBINE BLADE

(HALF SYMMETRY)

Fig. 7.16.1   Discretization of a Turbine Blade
(Insert Elements Not Shown)
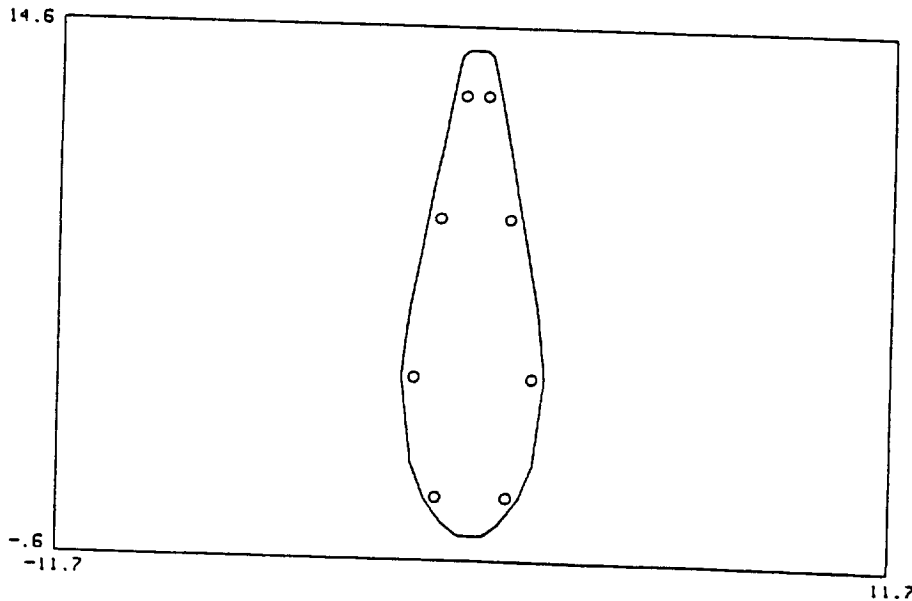
Fig. 7.16.2   Cross-Sectional View of the Turbine Blade
Displaying the Location of the Eight Fibers
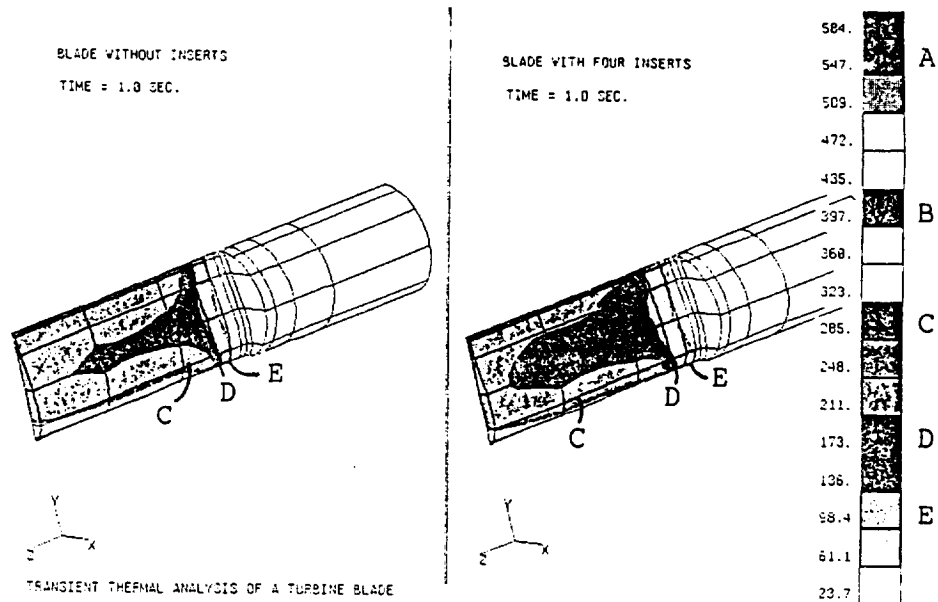
Fig. 7.16.3 Transient Heat Conduction Analysis of a
            Turbine Blade at 1 Second
            (Contour Plots of Temperature)
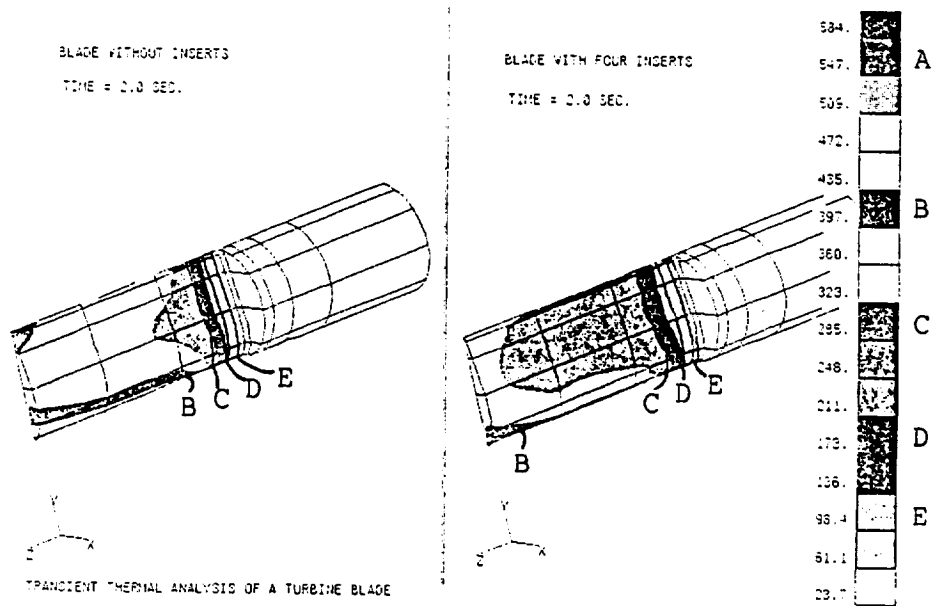
Fig. 7.16.4 Transient Heat Conduction Analysis of a Turbine
            Blade at 2 Seconds
            (Contour Plots of Temperature)
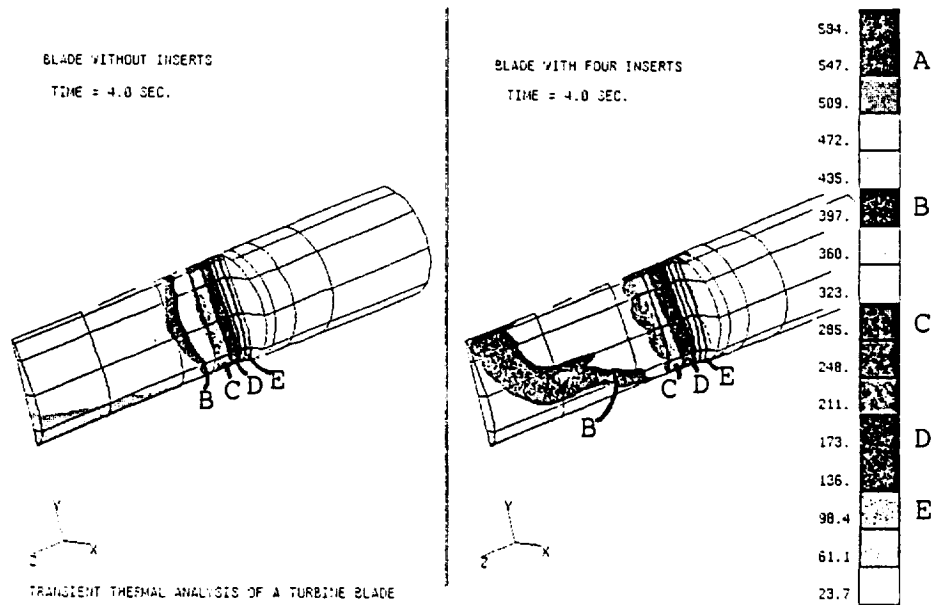
Fig. 7.16.5   Transient Heat Conduction Analysis of a
Turbine Blade at 4 Seconds
(Contour Plots of Temperature)
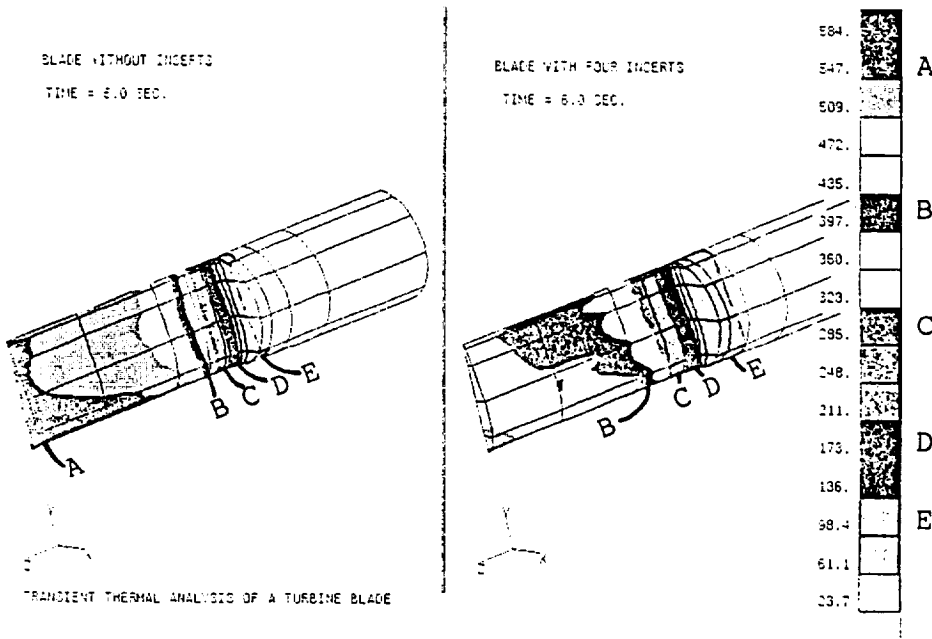
Fig. 7.16.6   Transient Heat Conduction Analysis of a Turbine
Blade at 6 Seconds
(Contour Plots of Temperature)

BLADE WITHOUT INSERTS

TIME = 8.0 SEC.

BLADE WITH FOUR INSERTS

TIME = 8.0 SEC.

TRANSIENT THERMAL ANALYSIS OF A TURBINE BLADE

584.
547.   A
509.
472.
435.
397.   B
360.
323.
285.   C
248.
211.
173.   D
136.
98.4   E
61.1
23.7
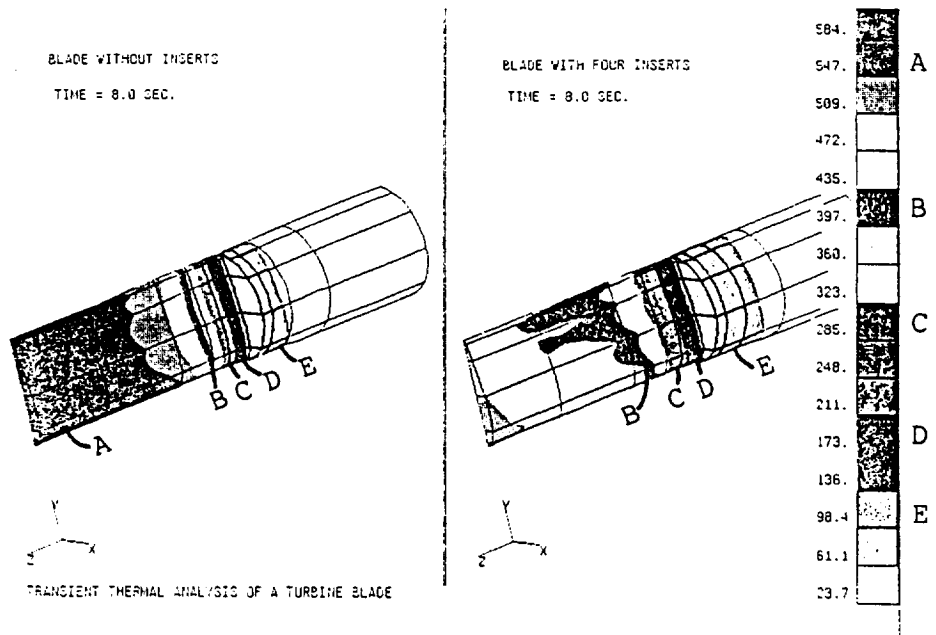
Fig. 7.16.7   Transient Heat Conduction Analysis of a Turbine
Blade at 8 Seconds
(Contour Plots of Temperature)
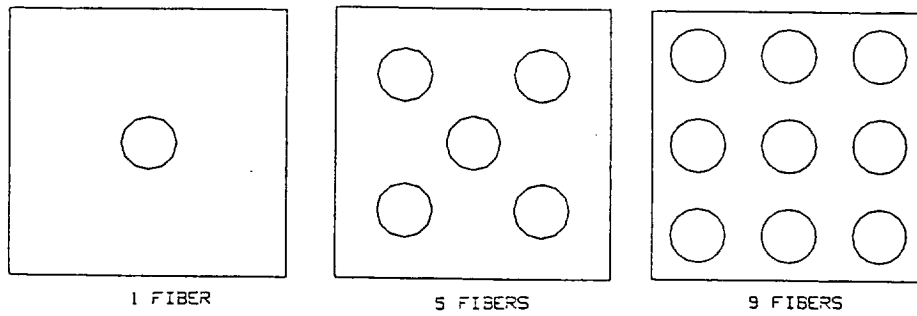
Fig. 7.17.1   Arrangement of Multiple Fibers in a Unit Cube
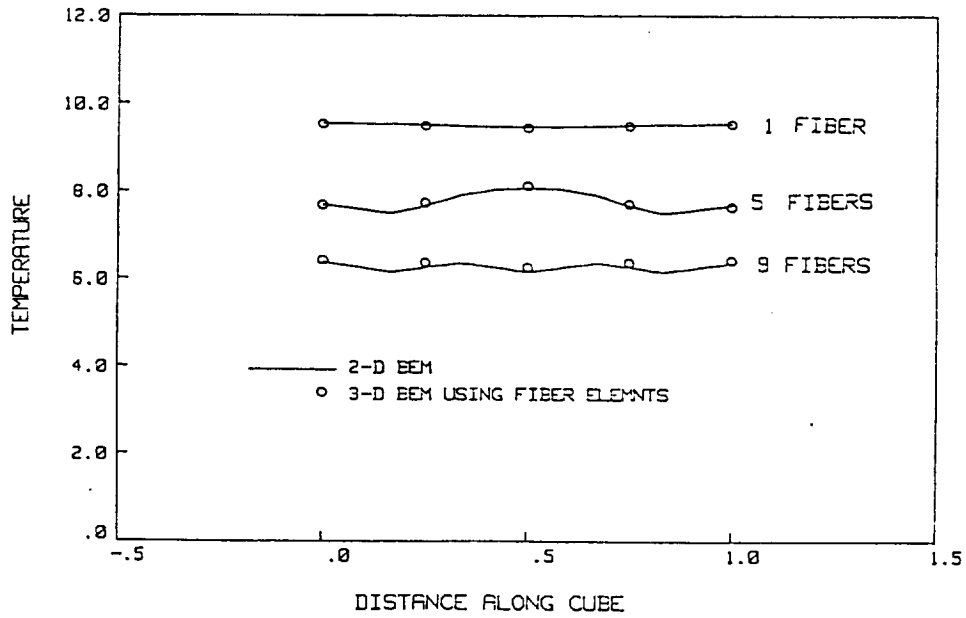
Fig. 7.17.2   Temperature Profile in a Cube for Multiple
Fiber Arrangements
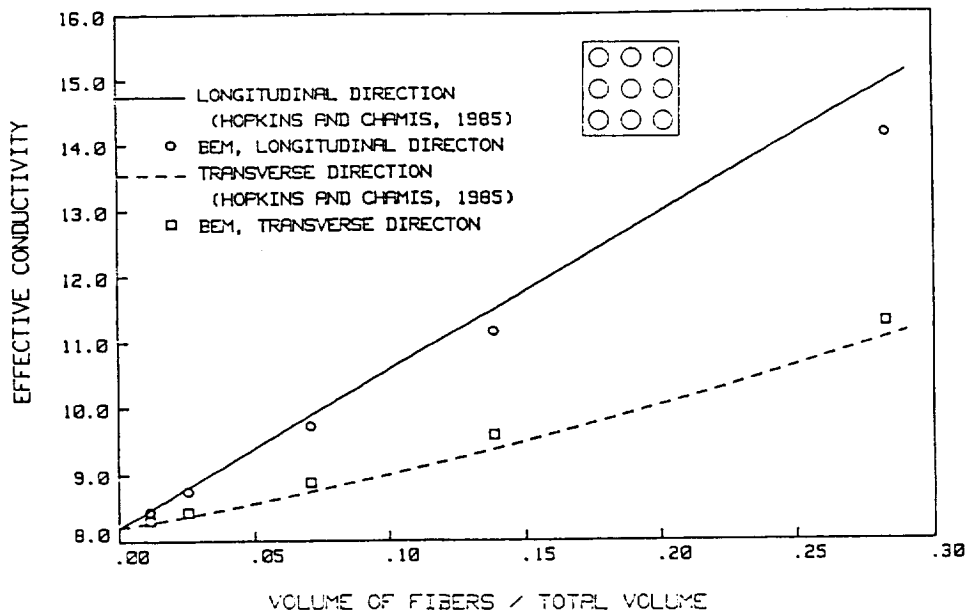
Fig. 7.18.1   Effective Conductivity in a Cube for Various
Fiber Diameters
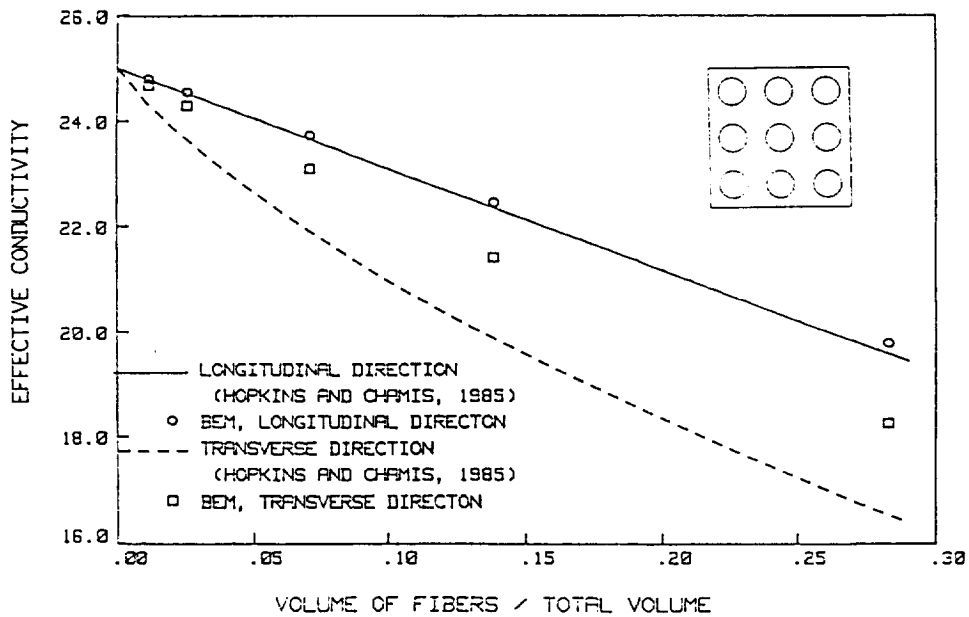
Fig. 7.18.2   Effective Conductivity in a Cube for Various
Fiber Diameters
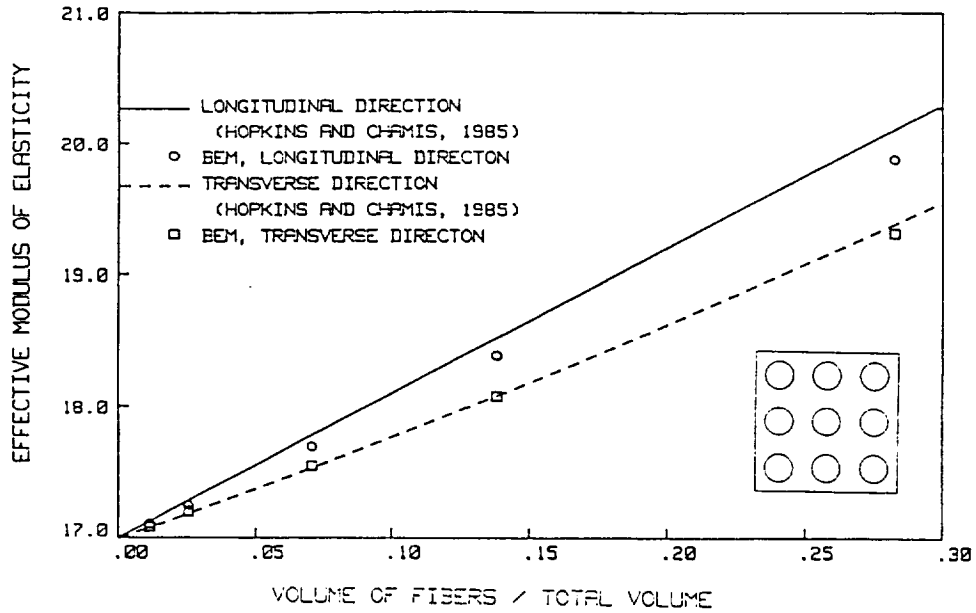
Fig. 7.19.1 Effective Modulus of Elasticity in a Cube for Various Fiber Diameters
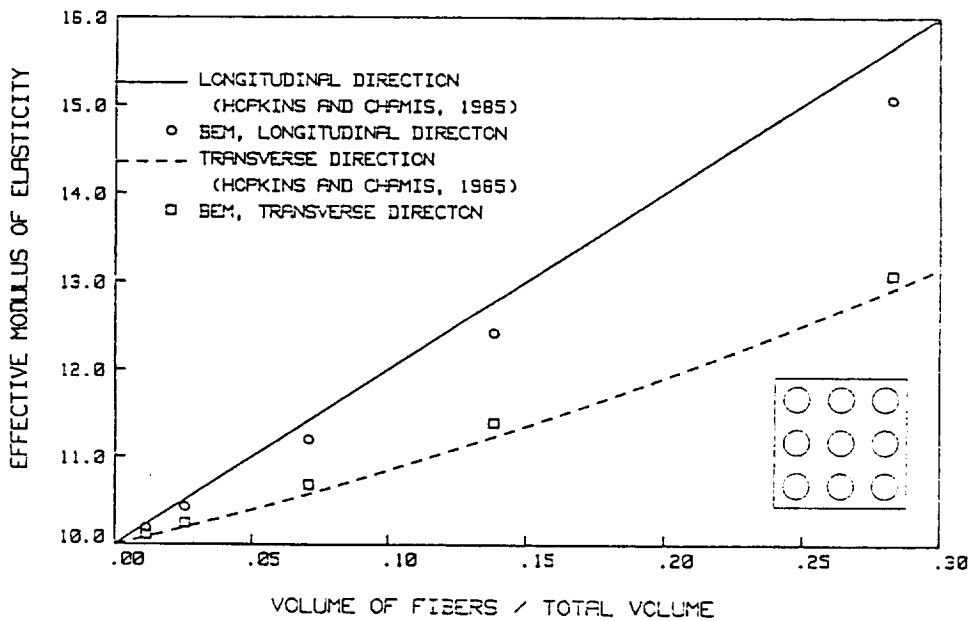
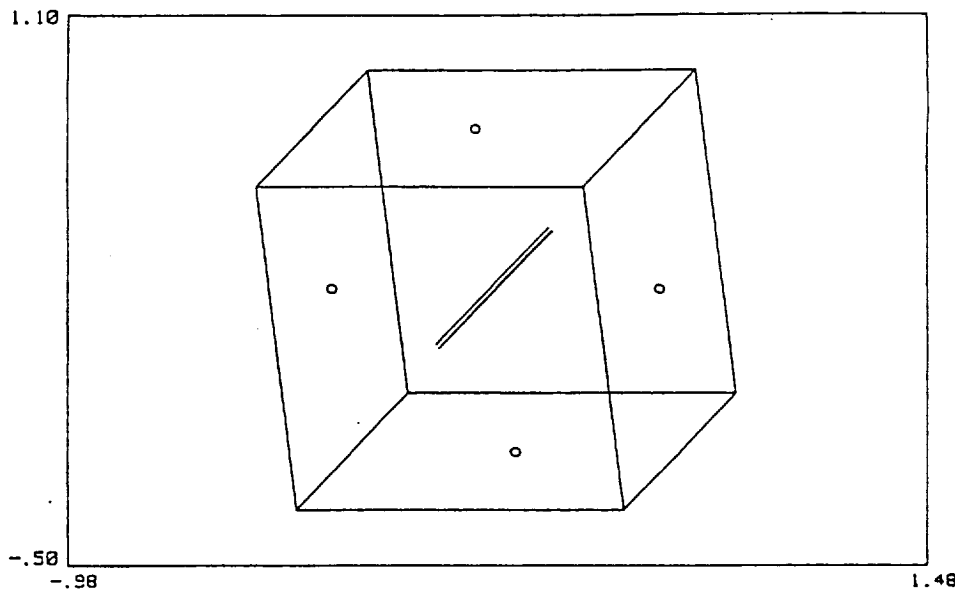Fig. 7.19.2 Effective Modulus of Elasticity in a Cube for Various Fiber Diameters

Fig. 7.20.1 Discretization of a Fiber in a Unit Cube Using
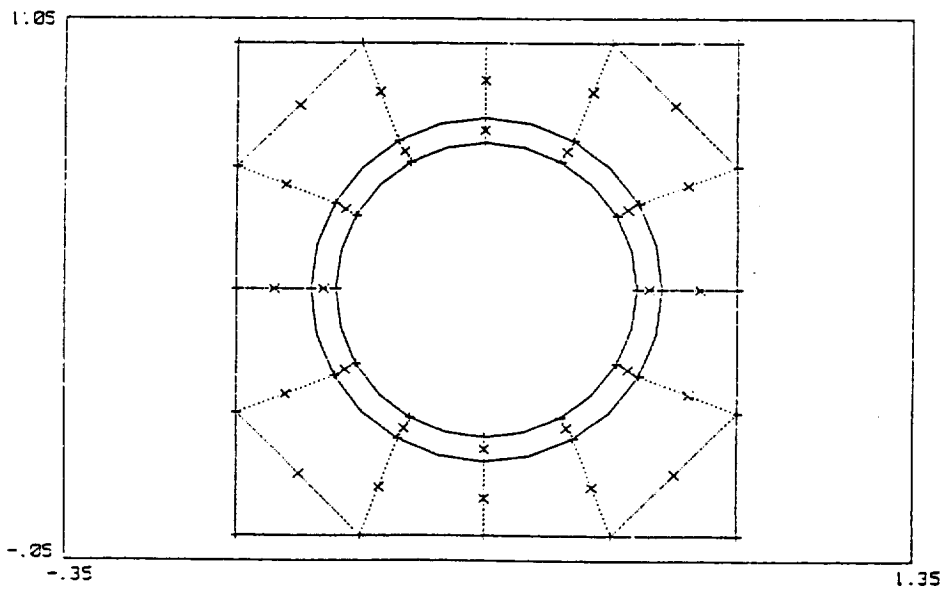Fiber Elements (includes a cylindrical volume
cell not shown)

Fig. 7.20.2 Two-dimensional Discretization of a Cube with a
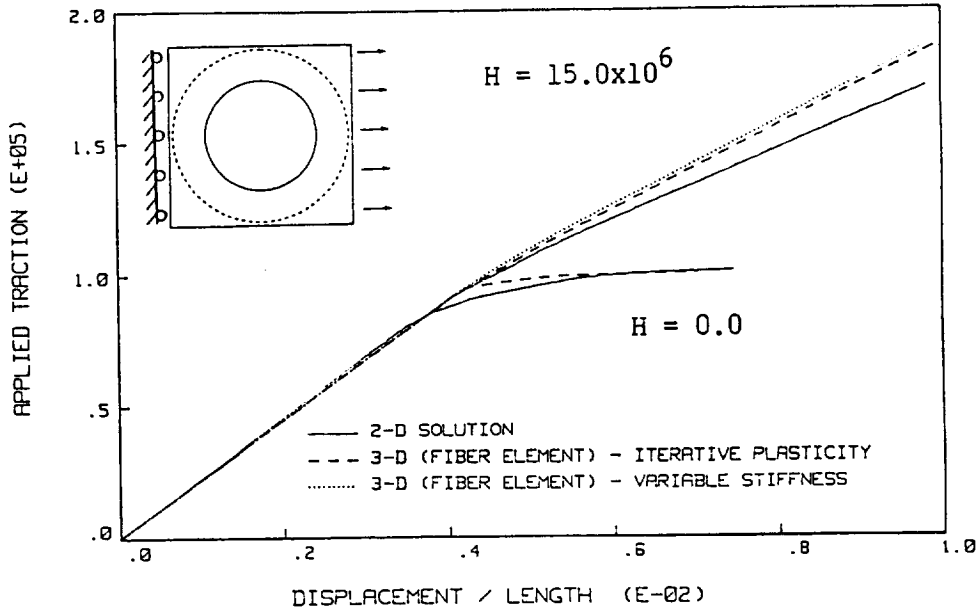Fiber; Two-dimensional Volume Cells are Shown

Fig. 7.20.3 Load vs. Displacement Response in a Fiber Composite
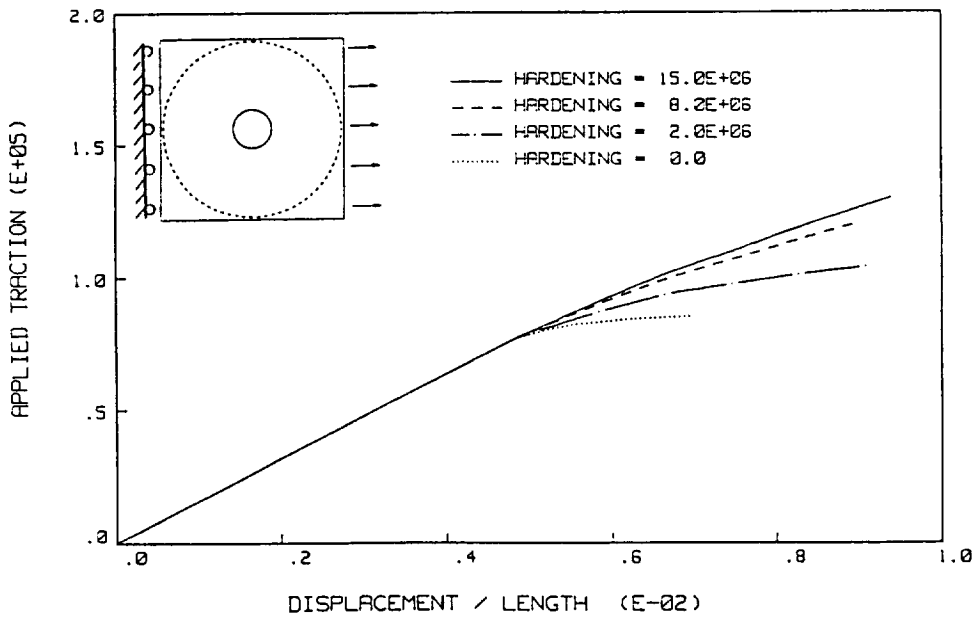With and Without Strain Hardening

Fig. 7.21.1 Load vs. Displacement Response in a Fiber Composite
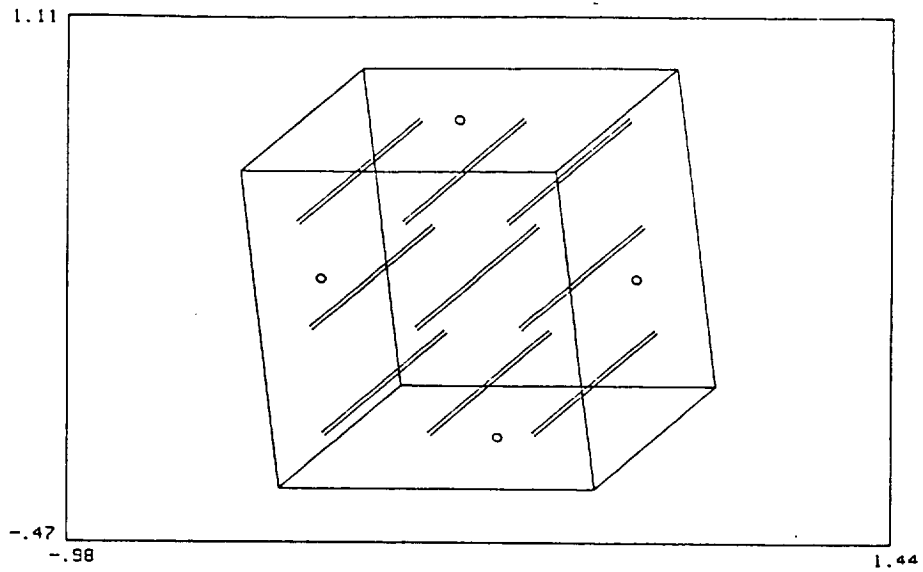for Various Strain Hardening Parameters

Fig. 7.22.1 Discretization of Nine Fibers in a Unit Cube
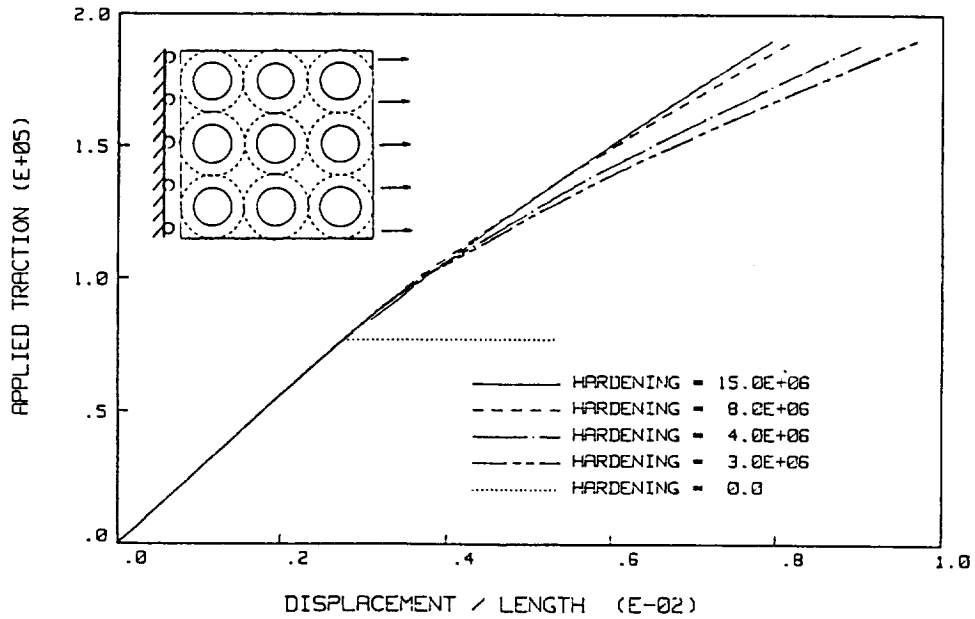using Fiber Elements (includes cylindrical volume
cells not shown)

Fig. 7.22.2 Load vs. Displacement Response in a Fiber
Composite for Various Strain Hardening
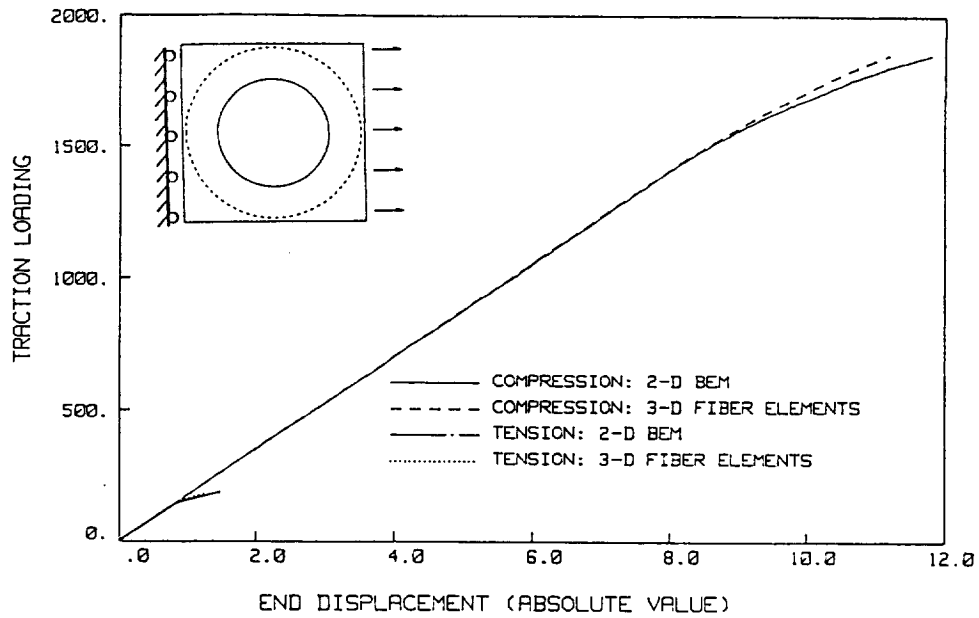Parameters

Fig. 7.23.1 Elastoplastic-Fracture Load vs. Displacement Behavior in a Fiber Composite. Response under a Tensile Load and a Compression Load are both shown.
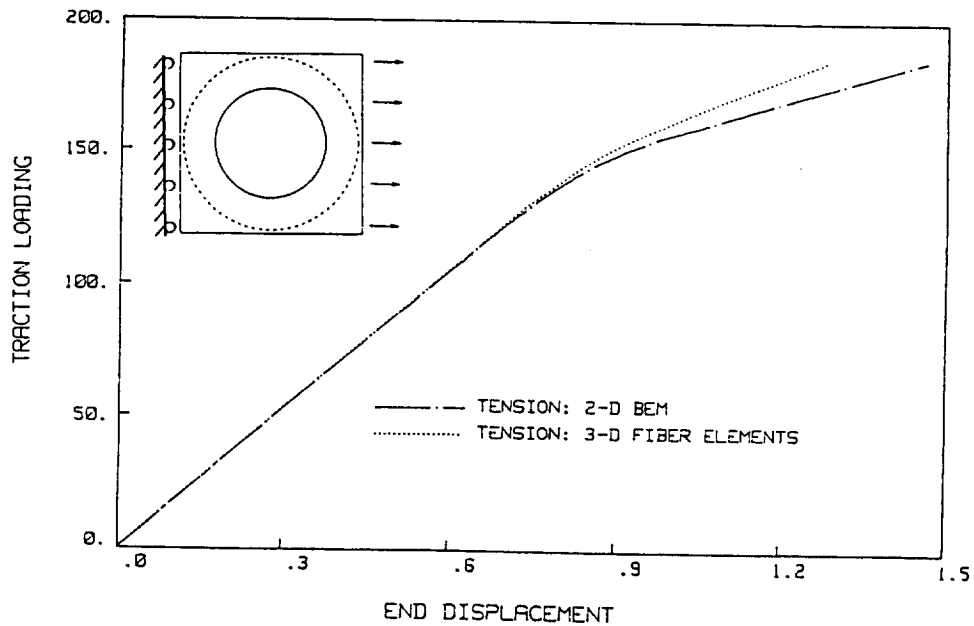
Fig. 7.23.2 Close-up of the Elastoplastic-Fracture Load vs. Displacement Response of the Tensile Load of Fig. 8.23.1
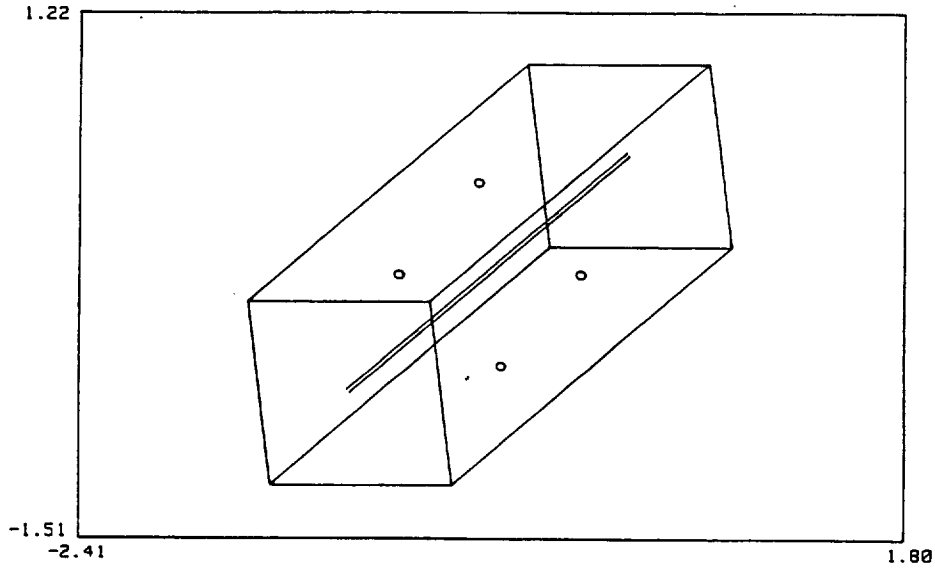
Fig. 7.24.1 Discretization of a Fiber in a Rectangular Block using Fiber Elements (includes a cylindrical volume cell not shown)
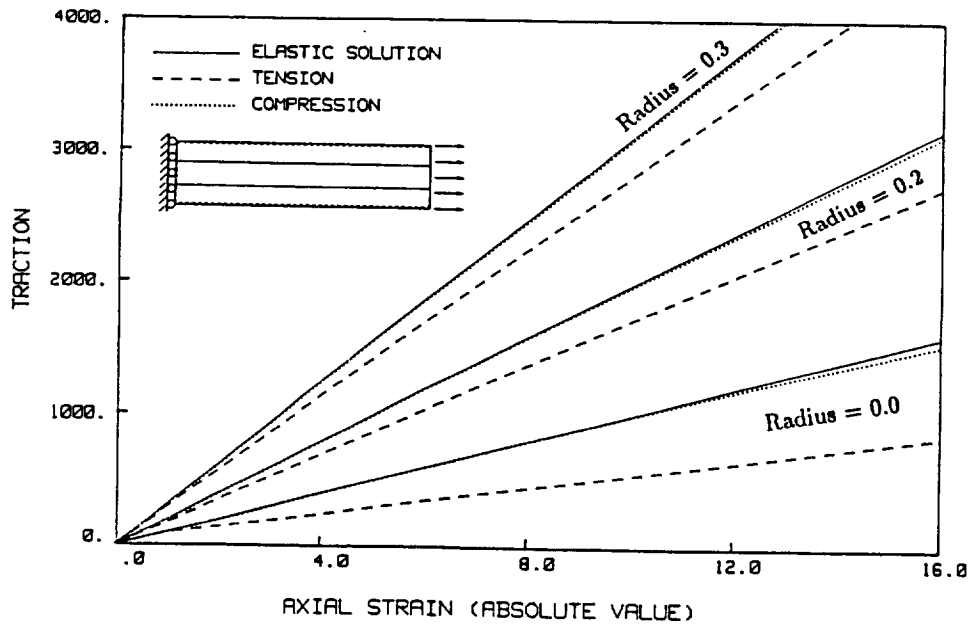
Fig. 7.24.2 Traction vs. Axial Strain (in the Fiber). Elastoplastic-Fracture with Strain Hardening Behavior in Fiber Composites for Various Fiber Radii
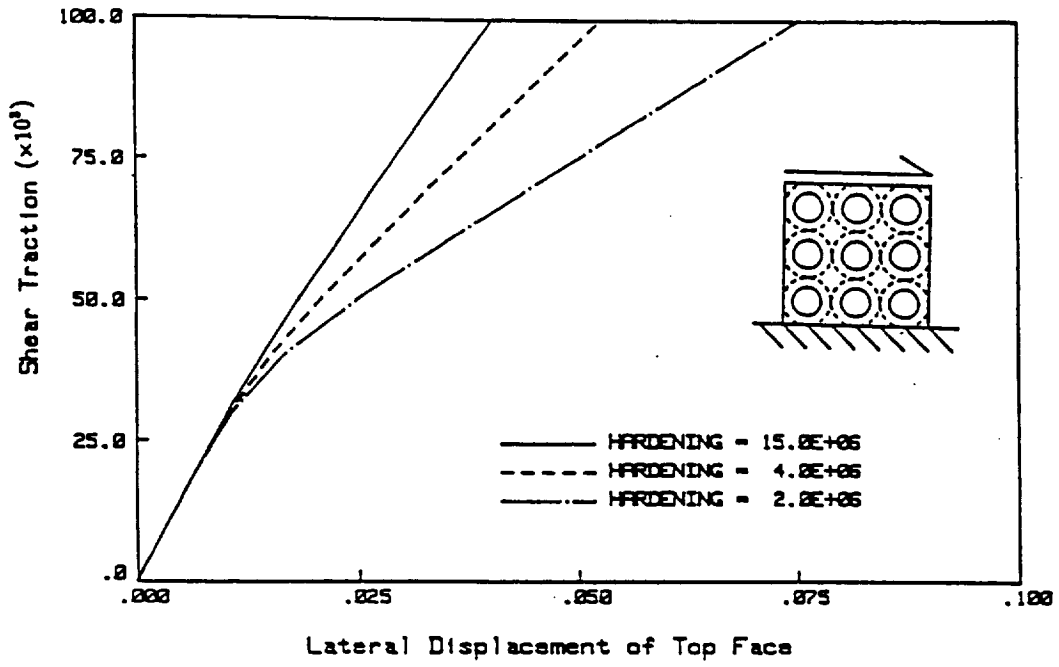
Fig. 7.25.1 Block with Nine Fibers (radius = 0.1) Subjected to Shear Loading Across Fibers
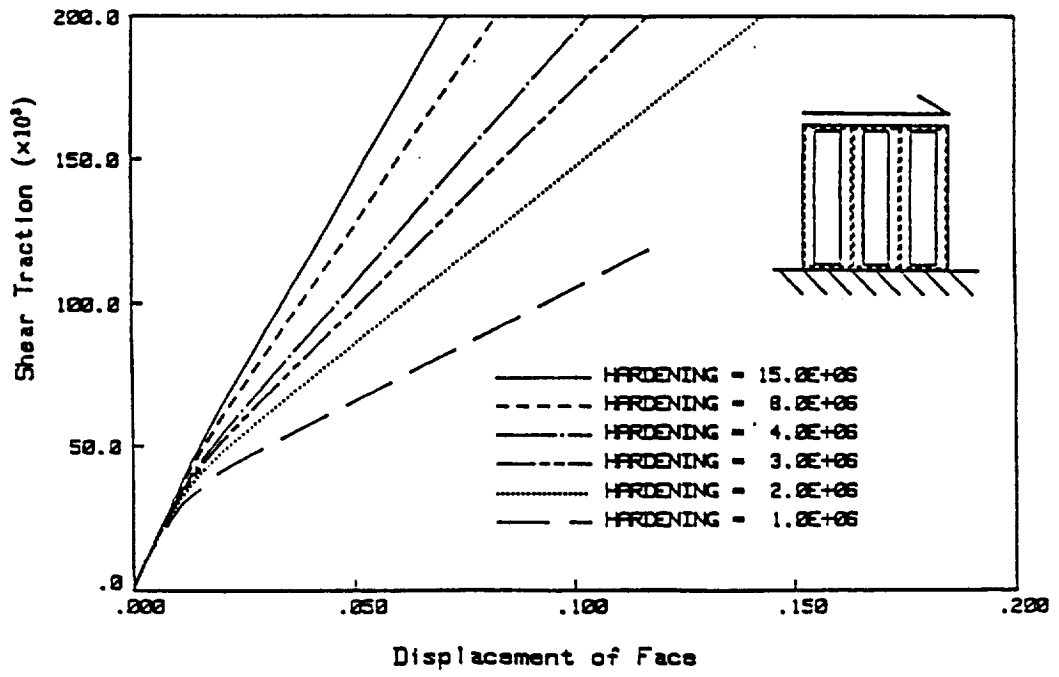
Fig. 7.25.2 Block with Nine Fibers (radius = 0.1) Subjected to Shear Loading Perpendicular to Fibers
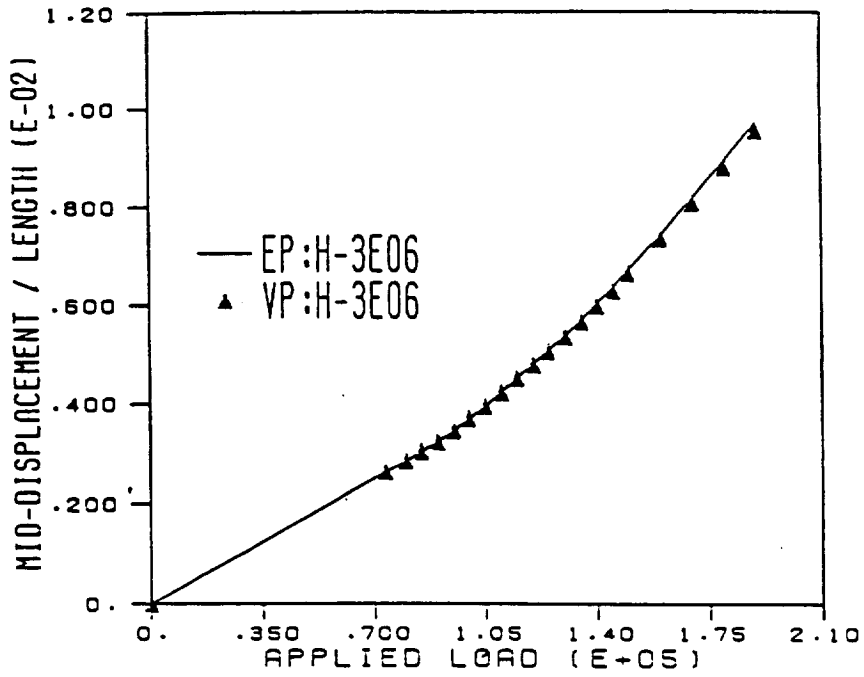
Fig. 7.25.3 Comparison of   Elastoplastic and Viscoplastic
Solutions for Cube with Nine Fibers
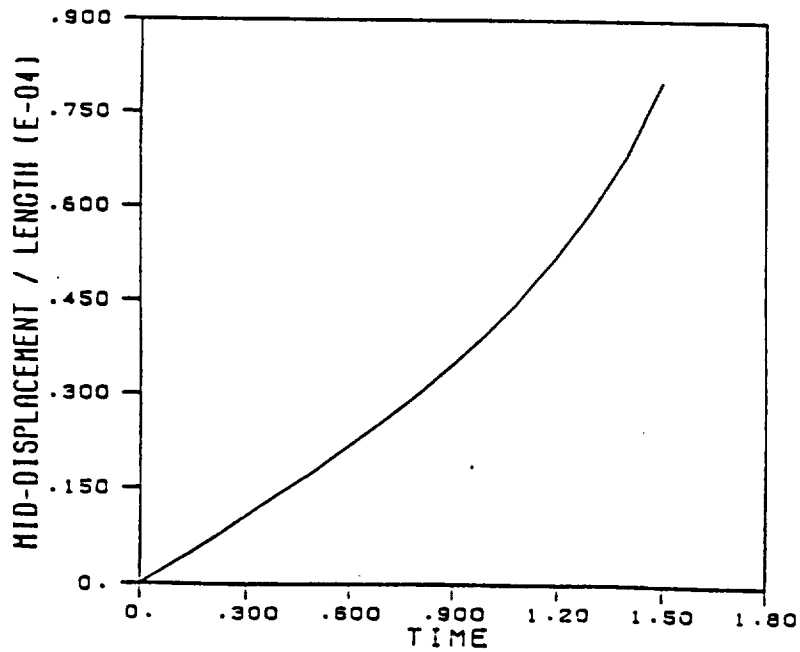
Fig. 7.25.4 Creep Solution: Strain-time Response for Cube
with Nine Fibers (Gamma = 1.596E-16, N = 4)
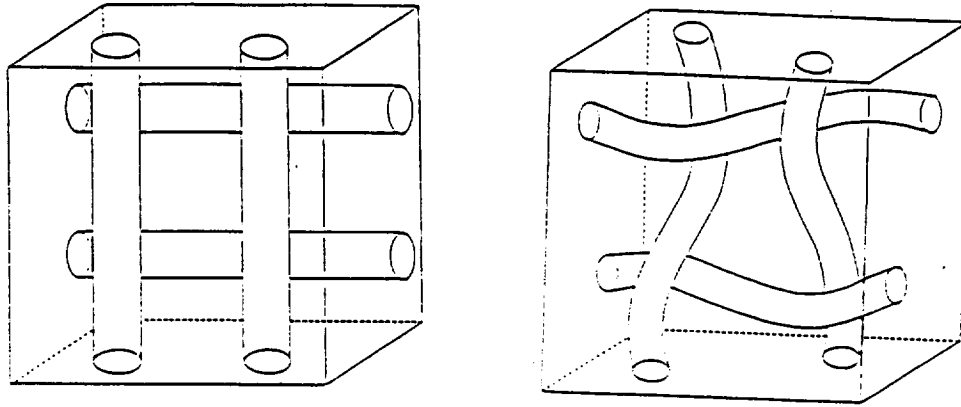
Fig. 7.26.1   SiC/RBSN Composite - Crossed and Woven Fibers
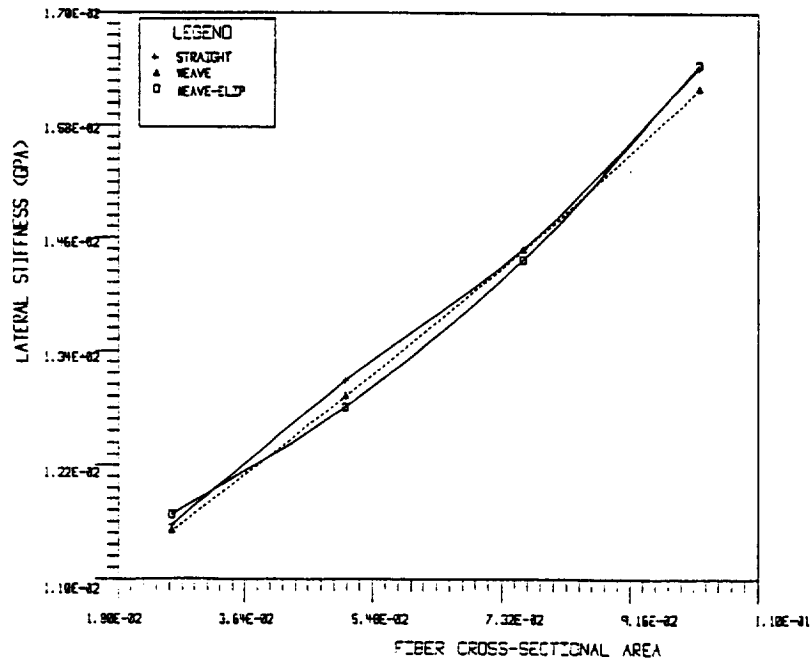
Fig. 7.26.2   Lateral Stiffness in a SiC/RBSN Composite -
Crossed, Weave and Weave with Elliptic Cross-Section
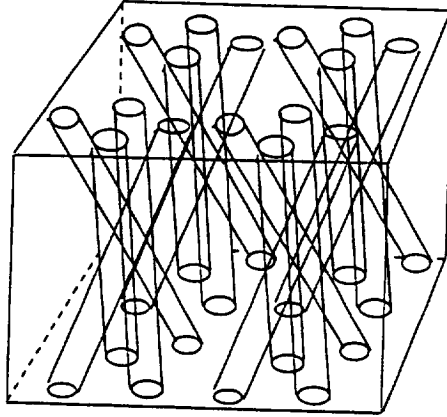
Fig. 7.27.1 SiC/RBSN Composite with Oblique Fiber Bundles
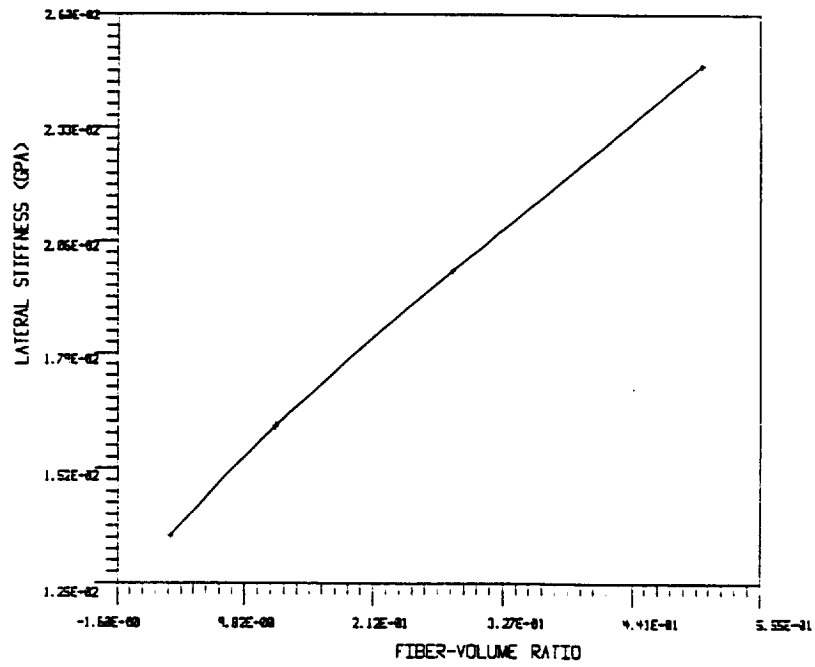
Fig. 7.27.2 Axial Stiffness in a SiC/RBSN Composite with Oblique
Fibers
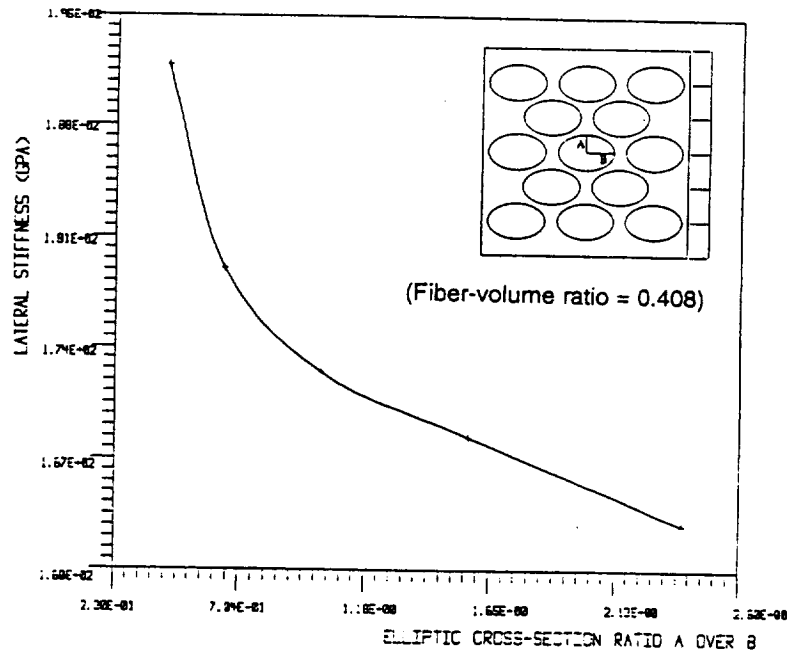
Fig. 7.28.1   Lateral Stiffness in a SiC/RBSN Fiber Composite
for Various Elliptic Cross-sections
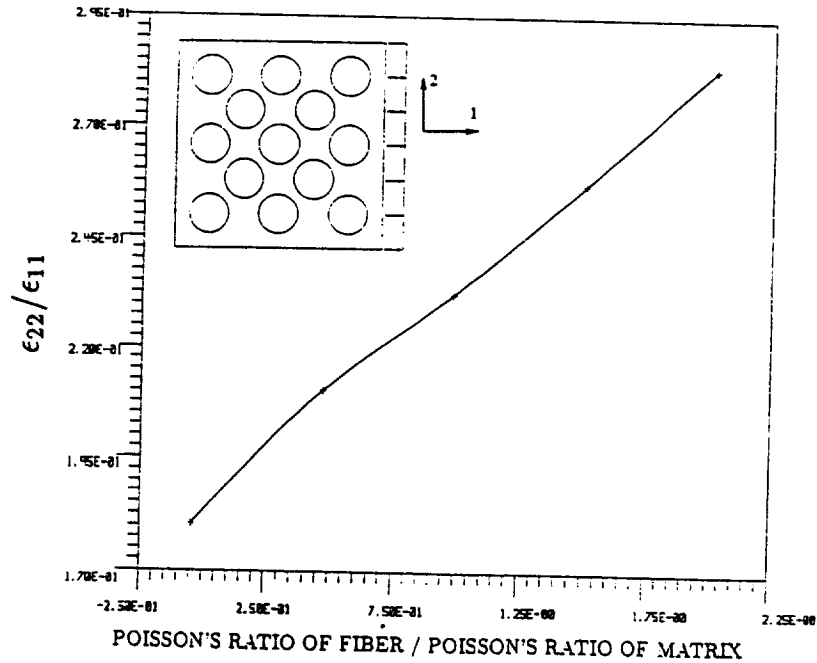
Fig. 7.29.1   Lateral Deformation for Different Fiber Poisson's
Ratios
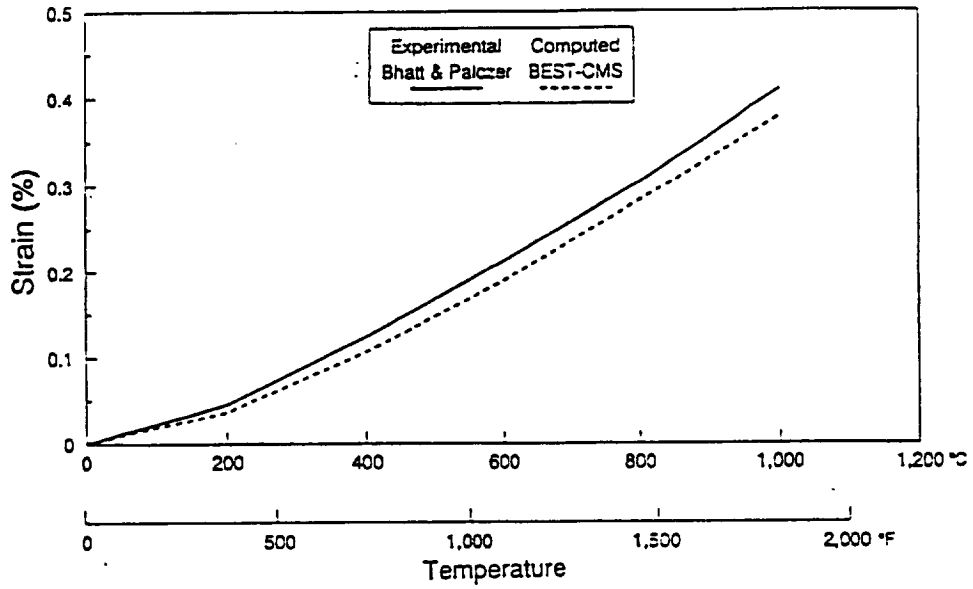
Fig. 7.30.1   Longitudinal Thermal Expansion of Unidirectional
SiC/RBSN with Strong Interface
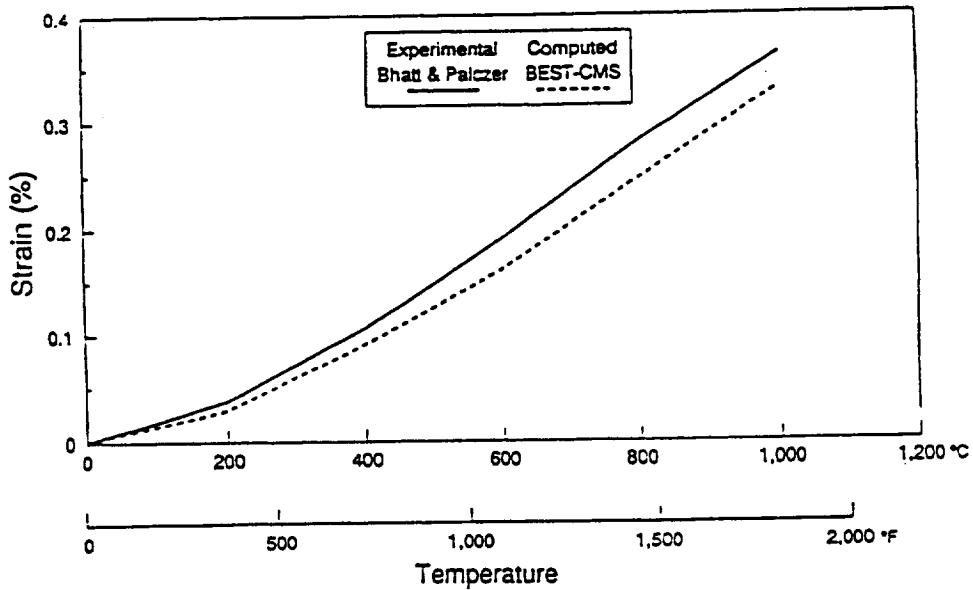
Fig. 7.30.2   Longitudinal Thermal Expansion of Unidirectional
SiC/RBSN with Weak Interface
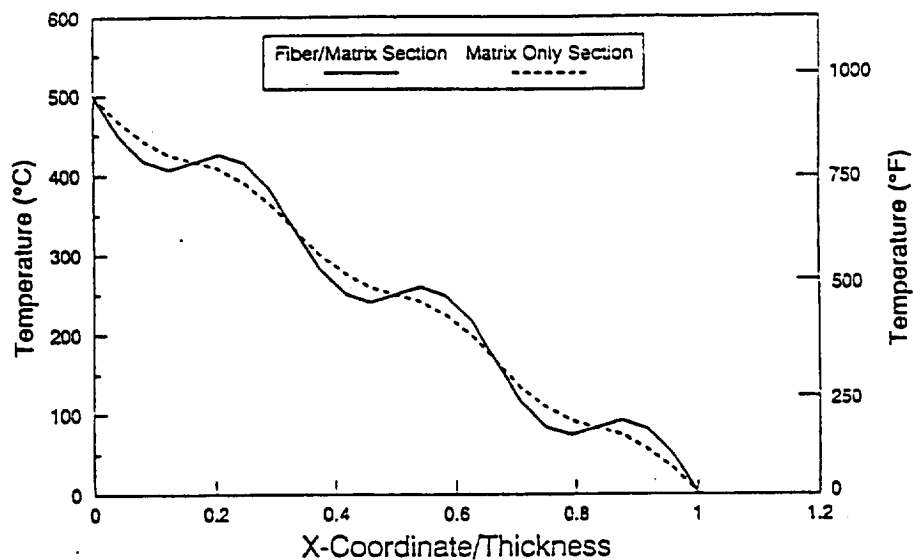
Fig. 7.30.3   Through-the Thickness Temperature Profile of
               SiC/Ti-15-3; Fiber/Matrix Conductivity Ratio = 50;
               Three Fibers Thick, Constant Fiber Spacing
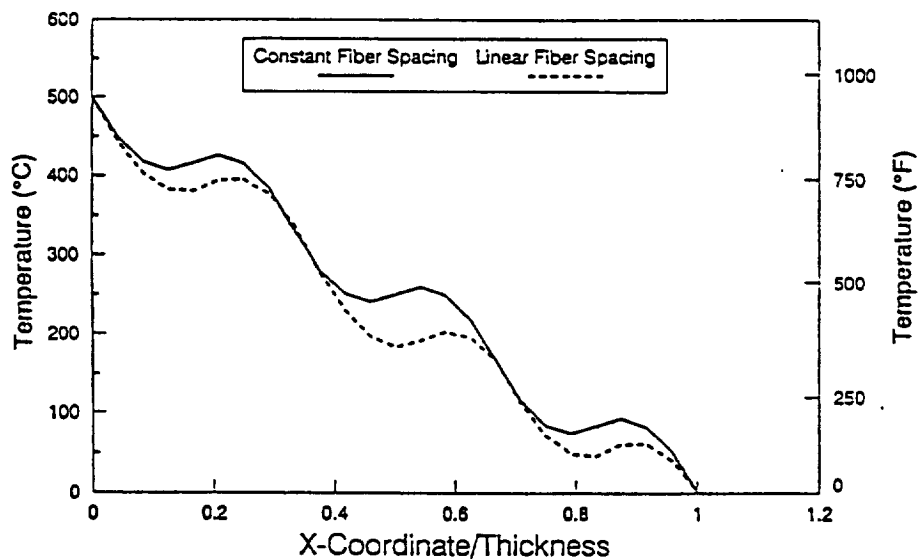
Fig. 7.30.4   Effect of Varying Fiber Spacing on Temperature
               Profile for Fiber/Matrix Cross Section; Three
               Fibers Thick, Conductivity Ratio = 50

Fiber 2

Fiber 1

(a)

(b)
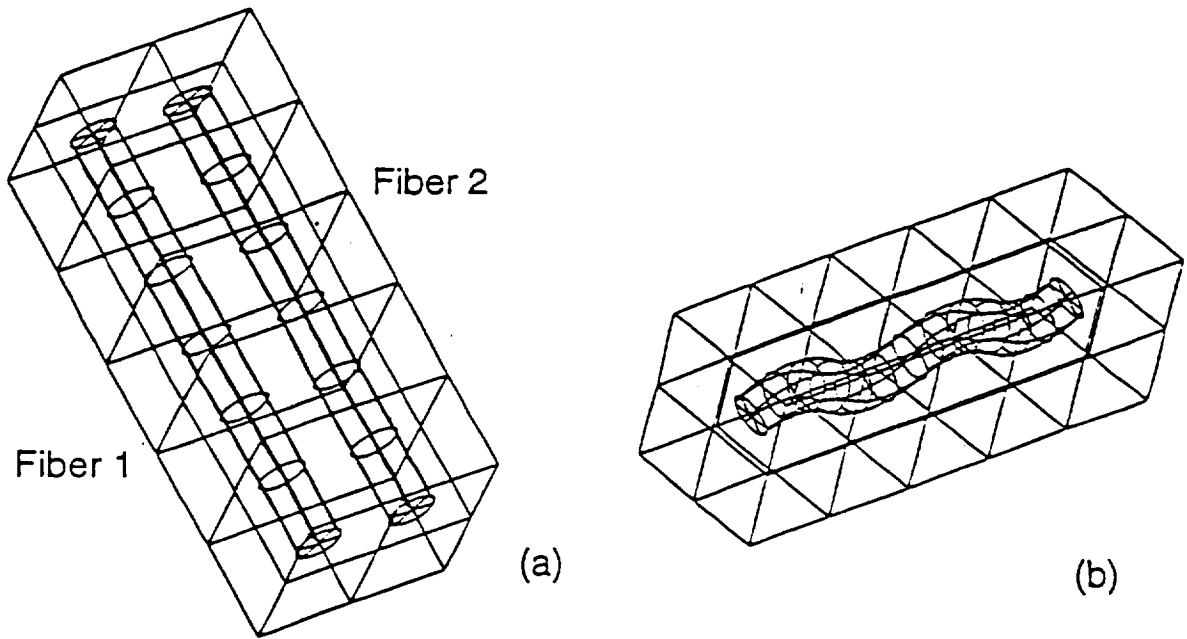
Fig. 7.31.1   Microsurface Models of Straight Fibers and a
Single Curved Fiber
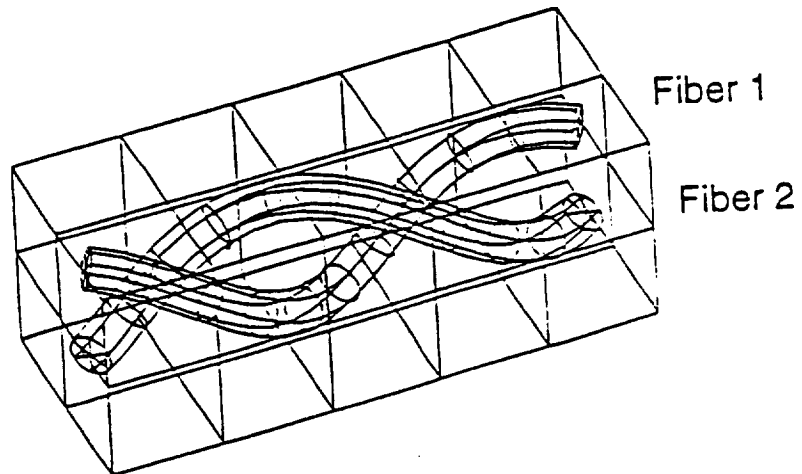
Fiber 1

Fiber 2

Fig. 7.31.2   Microsurface Models of Two Curved Fibers

Bonded

Sliding
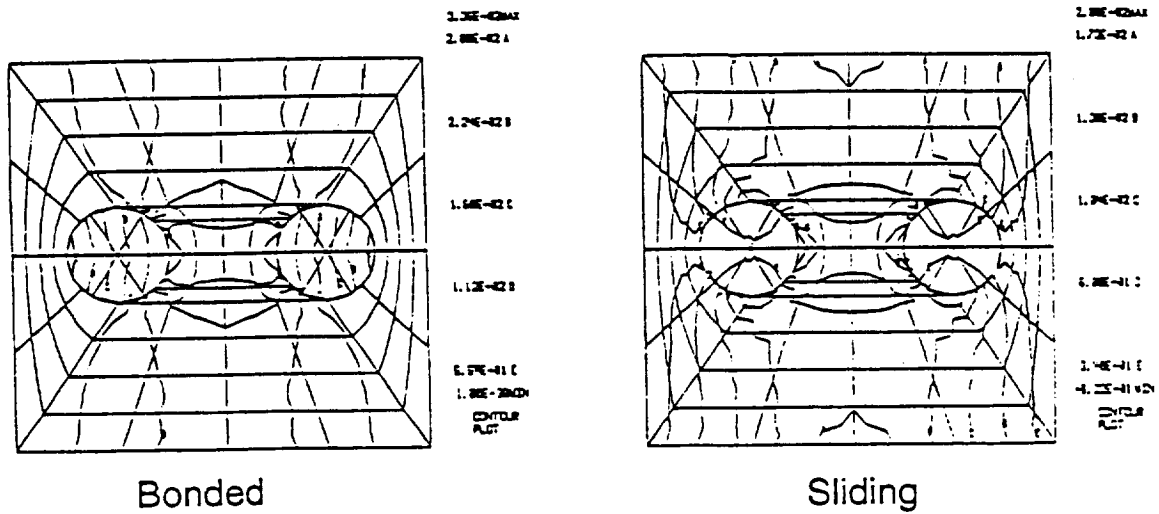
Fig. 7.31.3   Sampling Surface at L = 15 Cutting Plane Showing
Shear Stress Distribution of the Model Shown in
Fig. 7.31.1a under Linear Bending
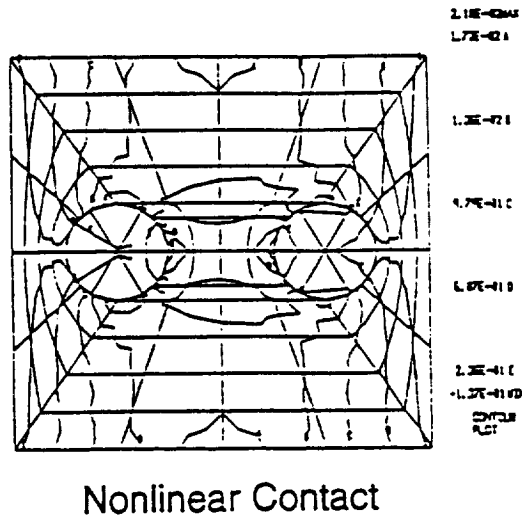
Nonlinear Contact

Fig. 7.31.4   Sampling Surface at L = 15 Cutting Plane Showing
Shear Stress Distribution of the Model Shown in
Fig. 7.31.1a for a Nonlinear Contact Analysis
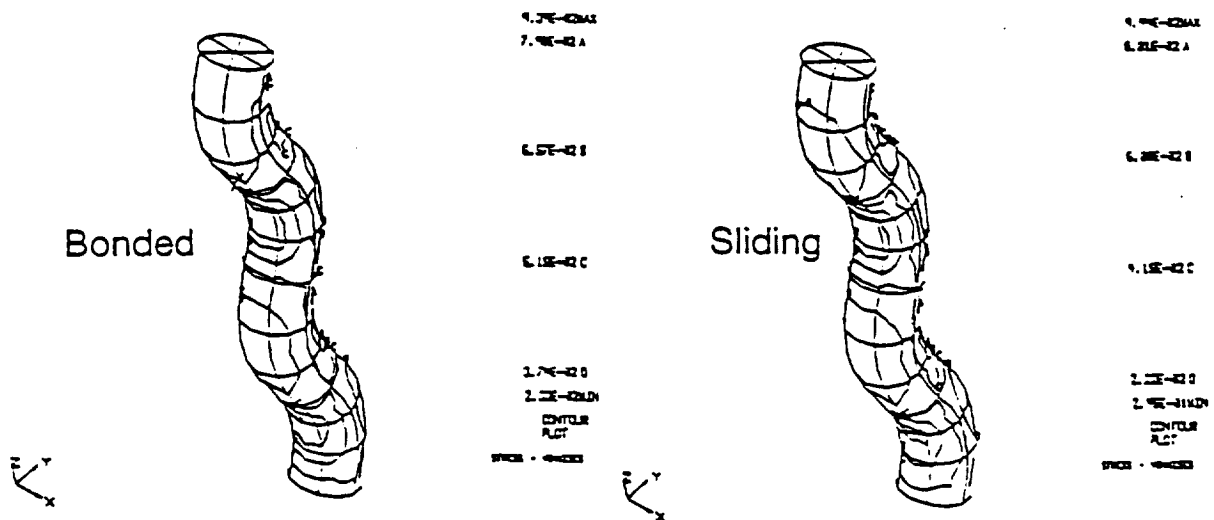
Fig. 7.31.5   Von Mises Stresses on the Surface of a Fiber Embedded
in a Block (shown in Fig. 7.31.1b) under Axial Tension
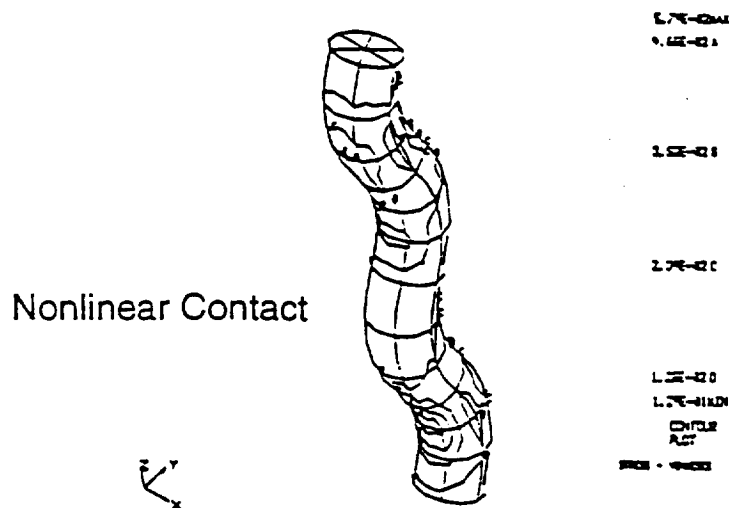
Fig. 7.31.6   Von Mises Stresses on the Surface of a Fiber Embedded
in a Block (shown in Fig. 7.31.1b) under Uniaxial Tension
in a Nonlinear Analysis with Gaps and Contact
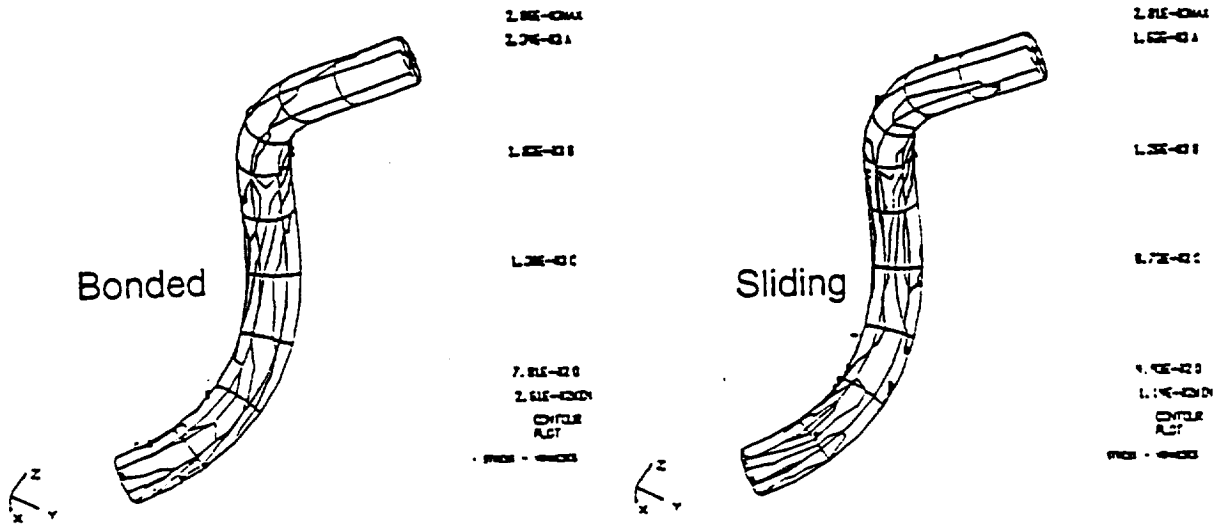
Fig. 7.31.7   Von Mises Stresses on Microsurfaces of Fiber 1 of the
              Two Fiber Problem (Fig. 7.31.2) under Bending with Bonded
              and Sliding Interfaces
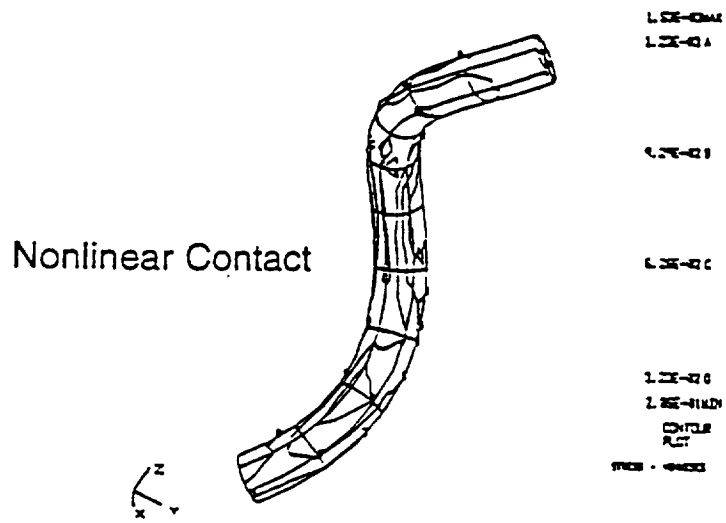
Fig. 7.31.8   Von Mises Stresses on Microsurfaces of Fiber 1 of the
              Two Fiber Problem (Fig. 7.31.2) under Bending with a
              Nonlinear Interface

| 8.0 | BEST-CMS NEUTRAL FILE FORMAT |

The BEST-CMS neutral file is designed as a general interface for post-processing of BEST-CMS result. Included in the neutral file are material and geometrical properties of the model as well as the result at each time step.

The name of the neutral file is NEUTRAL.OUT. It is a formatted, direct access file and each record is 80 characters in length. The number of records is based on the size and type of analysis.

On the next page a description of the contents of the BEST-CMS neutral file is given. This is followed by an outline of the order in which the sections appear in the neutral file along with indexing information for retrieving the appropriate records.

CONTENTS OF NEUTRAL FILE

OUTLINE OF NEUTRAL FILE
(RECORD ORDER AND INDEXING INFORMATION)

Section	Record Number of this record appears on
MAIN HEADER (always record 1)	- - -
MATERIAL HEADER (GMR 1)	main header
Material data for GMR 1	material 1 header

(Material header and data is recorded once for each GMR)

MATERIAL HEADER (GMR N)	material N-1 header
Material data for GMR N	material N header
GMR HEADER (GMR 1)	main header
GMR data for GMR 1	GMR 1 header

(GMR header and data is recorded for each GMR)

GMR Header (GMR N)	GMR N-1 header
GMR data for GMR N	GMR N header
RESULT HEADER (time 1)	main header
Results for Time 1	result header for time 1

(Result header and results are recorded for each time step)

RESULT HEADER (time N)	result header for time N-1
Results for Time N	result header for time N

NOTE: Indexing information for the data within a section is found on the header card of that section.

8.1 | PROBLEM (CASE INPUT) DESCRIPTION

8.1.1 | MAIN HEADER

The main header is always the 1st record of the neutral file. It describes the basic information about the problem of indexing information. The main header has the following format:

<u>Column Fields</u>

1) Problem type: (1 - 4)
   - = CTHE if Concurrent Thermal analysis
   - = ELAS if Elastic analysis
   - = HEAT if Heat Transfer analysis
   - = PLAS if Plasticity analysis

2) Dimensionality of the problem: (6 - 7)
   - = 3D if Problem is a 3-D

3) Symmetry of Problem : (8 - 9)
   - = blank if Problem does not specify symmetry
   - = HF if Problem has symmetry about the Y-Z plane
     ($R - \theta$ symmetry in axisymmetry)
   - = QU if Problem has symmetry about the X-Z & Y-Z planes
   - = OC if Problem has symmetry about the X-Z, Y-Z & X-Y planes

4) Restart condition: (11 - 14)
   - = WRIT   Write save all intergration file generated
   - = READ   Read bypasses intergration phase for run

5)   Problem precision: (16 - 19)

       = HIGH   Uses a 4 x 4 integration on each element

       = MEDI   Uses a 3 x 3 integration on each element

       = LOW   Uses a 2 x 2 integration on each element

6)   Total number of user defined material sets   (42 - 43)

7)   Total number of GMR's (geometric modeling regions)   (45 - 46)

8)   Total number of time steps   (48 - 50)

9)   Total number of records in this BEST-CMS neutral file   (52 - 57)

10)   Record number of the header of the first material   (59 - 64)

      (=0 if no material data is present)

11)   Geometric data flag   (66 - 66)

      (=0 header info only ; =1 complete geometric data)

12)   Record number of the header of the first GMR   (67 - 72)

13)   Record number of the header of the results of   (74 - 79)

      the first time step

14)   Continuation flag   (80)

BEST-CMS neutral file main header data format (1st record)

```
 1  2 3   4    5                    6  7  8    9     10  11   12      13 14
ANAL SYMM RE-5 PREC                NO. NO. NO.  TOTAL MAT 1   GMR 1   RES.1
                                  MAT GMR TS   NOR   HEADER  HEADER  HEADER
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCC CCCC CCCC CCCC                XX XX XXX XXXXXX XXXXXX XXXXXXX XXXXXX*
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)

    x - any valid number (0 - 9)

    * - indicates that the main header is continued on the next record

BESTCMS neutral file main header data format CARD 2

The second main heder card hs the following format:

1)   Title   (1 - 72)


```
                                 1
                               TITLE
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCCC
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```


where:

    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)

8.2 | MATERIAL DESCRIPTION

A material description is defined once for every GMR in the analysis, regardless of the number of user defined material sets in the BEST-CMS input data file of the problem. The reason for this being that certain material properties may be temperature dependent (and/or a function of velocity in fluids analysis) and even if the same material set is used for two or more GMR's the material properties assumed in the analysis may be different for each GMR since the temperature (and/or velocity) assumed in the integration (see TINT card in **GMR input) may be different for each GMR.

The material header consists of two subsequent records. Each material will have a material header that contains material data and indexing information.

MATERIAL HEADER (CARD 1)

The first material header card has the following format:

			Column Fields
1)	GMR ID		(1-7)
2)	Material ID		(10 - 17)
3)	Number of temperature values at which temperature dependent material properties are defined (= 0 for temperature independent)		(19-20)
4)	Starting record for temperature values at which temperature dependent properties are defined		(22 - 27)
5)	Number of temperature dependent moduli of elasticity		(29-30)
6)	Starting record for temperature dependent elastic modulus		(32 - 37)
7)	Number of temperature dependent thermal coefficient		(39 - 40)
8)	Starting record for temperature dependent thermal coefficient of expansion		(42 - 47)
9)	Starting record of compliance matrix (= 0 if not applicable to analysis)		(53-58)
10)	Starting record of stiffness matrix (= 0 if not applicable to analysis)		(60-65)
11)	Record number of thermal coefficient of expansion tensor (= 0 if not applicable to analysis)		(67-72)
12)	Record number of header for next material		(76 - 79)
13)	Continuation flag:		(80)

```
   1        2      3   4     5    6     7    8          9     10     11     12   13
  GMR    MATERIAL No. 1st.  No. 1st.  No.  1st.      COMP.  STIFF.  ALPHA  NEXT
   ID       ID    TEMP TEMP EMOD EMOD ALPH ALPHA     MATRIX MATRIX TENSOR HEADER C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC CCCCCCCC xx xxxxxx xx xxxxxx xx xxxxxx      xxxxxx xxxxxx xxxxxx xxxxxx*
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    C - any valid character (A-Z or a-z or 0-9 or space)

    x - any valid number (0 -= 9)

    * - indicates the material header is continued on the next record

MATERIAL HEADER (CARD 2)

The second material header card has the following format:

Column Fields

1)   Integration temperature   (2 - 13)
      (see TINT card in **GMR)
2)   Modulus of elasticity assumed for integration   (15 - 26)
3)   Thermal coefficient of expansion assumed for   (29 - 39)
      integration
4)   Poisson's ratio   (41 - 52)

```
     1              2              3              4
 INTEGRATION    INTEGRATION    INTEGRATION    POISSON'S
 TEMPERATURE      EMOD           ALPHA          RATIO
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0-9)
    s - is the sign of the mantisa and exponent
    . - is the decimal point and is required
    E - is an exponent sign and is required
    * - either * or blank

8.2.2	TEMPERATURE INPUT VALUES

FOR TEMPERATURE DEPENDENT MATERIAL

The temperature values at which temperature dependent material properties are defined are stored in the following format:

		Column Fields
1)	Temperature value one	(2 - 13)
2)	Temperature value two	(15 - 26)
3)	Temperature value three	(28 - 39)
4)	Temperature value four	(41 - 52)
5)	Temperature value five	(54 - 65)
6)	Temperature value six	(67 - 78)
7)	Continuation flag	(80)

```
         1              2              3              4              5              6        7
   TEMPERATURE1   TEMPERATURE2   TEMPERATURE3   TEMPERATURE4   TEMPERATURE5   TEMPERATURE8  C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
  x - any valid number (0-9)
  s - is the sign of the mantisa and exponent
  . - is the decimal point and is required
  E - is an exponent sign and is required
  * - either * or blank

For more than six values, the continuation flag '*' is used and data is continued on to the next record. Repeat with as many records as necessary until all values are accounted for.

8.2.3	ELASTIC MODULUS INPUT VALUES

FOR TEMPERATURE DEPENDENT MATERIAL

The elastic modulus values for which temperature values are defined and are stored in the following format:

<u>Column Fields</u>

1)	Elastic modulus for temperature value one	(2 - 13)
2)	Elastic modulus for temperature value two	(15 - 26)
3)	Elastic modulus for temperature value three	(28 - 39)
4)	Elastic modulus for temperature value four	(41 - 52)
5)	Elastic modulus for temperature value five	(54 - 65)
6)	Elastic modulus for temperature value six	(67 - 78)
7)	Continuation flag:	(80)

```
         1             2             3             4             5             6          7
    E-MODULUS1    E-MODULUS2    E-MODULUS3    E-MODULUS4    E-MODULUS5    E-MODULUS6
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

x - any valid number (0-9)

s - is the sign of the mantisa and exponent

. - is the decimal point and is required

E - is an exponent sign and is required

* - either * or blank

For more than six values, the continuation flag '*' is used and the data continued on to the next record. Repeat with as many records as necessary until all values are accounted for.

The values of the thermal coefficient of expansion corresponding to the temperature input values of a temperature dependent material are defined and are stored in the following format:

<u>Column Fields</u>

1)	Elastic modulus for temperature value one	(2 - 13)
2)	Elastic modulus for temperature value two	(15 - 26)
3)	Elastic modulus for temperature value three	(28 - 39)
4)	Elastic modulus for temperature value four	(41 - 52)
5)	Elastic modulus for temperature value five	(54 - 65)
6)	Elastic modulus for temperature value six	(67 - 78)
7)	Continuation flag:	(80)

```
         1              2              3              4              5              6            7
     ALPHA1         ALPHA2         ALPHA3         ALPHA4         ALPHA5         ALPHA6
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

   x - any valid number (0-9)

   s - is the sign of the mantisa and exponent

   . - is the decimal point and is required

   E - is an exponent sign and is required

   * - either * or blank

For more than six values, the continuation flag '*' is used and the data is continued on to the next record. Repeat with as many records as necessary until all values are accounted for.

| 8.2.5 | COMPLIANCE MATRIX |

The compliance matrix corresponding to the modulus of elasticity used during integration relates the stress and strain (defined in global coordintes) in the following manner:

$$
\begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \epsilon_{xy} \\ \epsilon_{xz} \\ \epsilon_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}
$$

where the σ_{ij} are stresses, and ϵ_{ij} are tensor strains.

Each row of the compliance matrix is stored on a record for a total of six records.

The format of the records are:

<u>Column Fields</u>

1) Value corresponding to column one   (2 - 13)
2) Value corresponding to column two   (15 - 26)
3) Value corresponding to column three   (28 - 39)
4) Value corresponding to column four   (41 - 52)
5) Value corresponding to column five   (54 - 65)
6) Value corresponding to column six   (67 - 78)
7) Continuation flag: (80)

```
        1               2               3               4               5               6       7
    COLUMN 1        COLUMN 2        COLUMN 3        COLUMN 4        COLUMN 5        COLUMN 6
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0-9)
    s - is the sign of the mantisa and exponent
    . - is the decimal point and is required
    E - is an exponent sign and is required
    * - either * or blank

The continuation flag '*' is used to indicate the data is continued on the next card.

The stiffness matrix corresponding to the modulus of elasticity used during integration relates the stress and strain (defined in global coordinates) in the following manner:

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{13} & S_{14} & S_{15} & S_{16} \\ S_{21} & S_{22} & S_{23} & S_{24} & S_{25} & S_{26} \\ S_{31} & S_{32} & S_{33} & S_{34} & S_{35} & S_{36} \\ S_{41} & S_{42} & S_{43} & S_{44} & S_{45} & S_{46} \\ S_{51} & S_{52} & S_{53} & S_{54} & S_{55} & S_{56} \\ S_{61} & S_{62} & S_{63} & S_{64} & S_{65} & S_{66} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \epsilon_{xy} \\ \epsilon_{xz} \\ \epsilon_{yz} \end{pmatrix}$$

where the σ_{ij} are stresses, and ϵ_{ij} are tensor strains.

Each row of the stiffness matrix is stored on a record for a total of six records.

The format of the records are:

		Column Fields
1)	Value corresponding to column one	(2 - 13)
2)	Value corresponding to column two	(15 - 26)
3)	Value corresponding to column three	(28 - 39)
4)	Value corresponding to column four	(41 - 52)
5)	Value corresponding to column five	(54 - 65)
6)	Value corresponding to column six	(67 - 78)
7)	Continuation flag:	(80)

```
        1             2             3             4             5            6       7
    COLUMN%%[ PrinterError: Operator Call - paper entry misfeed ]%% 1       COLUMN 2       COLUMN 3
    ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
    s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  *
    ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    x - any valid number (0-9)

    s - is the sign of the mantisa and exponent

    . - is the decimal point and is required

    E - is an exponent sign and is required

    * - either * or blank

The continuation flag '*' is used to indicate the data is continued on the next card.

The thermal coefficient of expansion tensor given in global coordinates relates temperature to thermal strains.

$$
\begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \epsilon_{xy} \\ \epsilon_{xz} \\ \epsilon_{yz} \end{pmatrix}
=
\begin{pmatrix} \alpha_{11} \\ \alpha_{22} \\ \alpha_{33} \\ \alpha_{12} \\ \alpha_{13} \\ \alpha_{23} \end{pmatrix}
* \text{ Temperature}
$$

in which thermal strains are defined as tensor strains. For isotropic material $\alpha_{11} = \alpha_{22} = \alpha_{33}$ and $\alpha_{12} = \alpha_{13} = \alpha_{23} = 0$.

The following format is used.

		Column Fields
1)	Value of α_{11}	(2 - 13)
2)	Value of α_{22}	(15 - 26)
3)	Value of α_{33}	(28 - 39)
4)	Value of α_{12}	(41 - 52)
5)	Value of α_{13}	(54 - 65)
6)	Value of α_{23}	(67 - 78)

```
     1            2            3            4            5            6
  ALPHA-11     ALPHA-22     ALPHA-33     ALPHA-12     ALPHA-13     ALPHA-23
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    x - any valid number (0-9)

    s - is the sign of the mantisa and exponent

    . - is the decimal point and is required

    E - is an exponent sign and is required

    * - either * or blank

| 8.3 | GMR DESCRIPTION |

| 8.3.1 | GMR HEADER |

The GMR header consists of three subsequent records. Each GMR will have a header that contains GMR data and indexing information.

GMR HEADER (CARD 1)

The first GMR header card has the following format:

		Column Fields
1)	GMR's ID:	(1 - 8)
2)	Material ID for region:	(10 - 17)
3)	Total number of surfaces in this GMR:	(22 - 25)
4)	Total number of points in this GMR:	(27 - 30)
5)	Total number of boundary elements in this GMR:	(32 - 35)
6)	Total number of boundary source points in this GMR:	(37 - 40)
7)	Total number of hole elements in this GMR:	(42 - 45)
8)	Total number of hole source points in this GMR:	(47 - 50)
9)	Total number of volume cells in this GMR:	(52 - 55)
10)	Total number of volume source points in GMR:	(57 - 60)
11)	Total number of sampling points in this GMR:	(62 - 65)
12)	Total number of sampling surfaces in this GMR:	(67 - 69)
13)	Record number of header for next GMR: (=0 for last GMR)	(74 - 79)
14)	Continuation flag:	(80 - 80)

```
     1         2         3     4    5    6    7    8    9   10   11   12        13 14
   GMR      MATERIAL     NO.  NO.  NO.  NO.  NO.  NO.  NO.  NO.  NO.  NO.      NEXT
   ID         ID         SURF PTS  B-EL SP   H-EL HSP  CELL VSP  SAMP S.S.     HEADER
 ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 CCCCCCCC CCCCCCCC       XXXX XXXX XXXX XXXX XXXX XXXX XXXX XXXX XXXX XXX      XXXXXX*
 ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where   .
   C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
   x - any valid number (0 - 9)
   * - indicates the GMR header is continued on the next record

GMR HEADER (CARD 2)

The second GMR header card which contains indexing information has the following format:

		Column Fields
1)	Starting record number of coordinate data	(2 - 7)
2)	Number of records for coordinate data	(9 - 12)
3)	Starting record number of boundary element connectivity	(14 - 19)
4)	Number of records for boundary element connectivity data	(26 - 31)
5)	Starting record number of hole/insert element connectivity	(26 - 31)
6)	Number of records for hole/insert element connectivity data	(33 - 36)
7)	Starting record number of volume cell connectivity	(38 - 43)
8)	Number of records for volume cell connectivity data	(45 - 48)
9)	Starting record number of sampling surface-element connectivity	(50 - 55)
10)	Number of sampling surface elements (since sampling surface elements require only one record per element, this is also equal to the number of records for sampling surface elements data)	(57 - 60)
11)	Indicates the GMR header is continued on the next record	(80)

```
     1       2       3       4       5       6       7       8       9       10                      9
   1st.            1st.            1st.            1st.            1st     NELE
   COORD.  NOR   B-ELEM  NOR   H-ELEM  NOR    CELL   NOR   SS-ELE  NOR                                 C
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
   xxxxxx xxxx xxxxxx xxxx xxxxxx xxxx xxxxxx xxxx xxxxxx xxxx                                         *
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    x - any valid number (0-9)

GMR HEADER (CARD 3)

The third GMR header card whic contains the reference (initial) temperature of the region and the temperature used at integration is given in the following format:

<u>Column Fields</u>

1)   Reference (initial) temperature of GMR   (2 - 13)
         (see TREF card in **GMR)

2)   Temperature used at integration   (15 - 26)

```
     1              2
  REFERENCE    INTEGRATION
 TEMPERATURE   TEMPERATURE
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx  s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0-9)
    s - is the sign of the mantisa and exponent
    . - is the decimal point and is required
    E - is an exponent sign and is required

| 8.3.2 | NODAL COORDINATES |

The nodal coordinate records have the following format:

<u>Column Fields</u>

1)   GMR ID   (1 - 8)

2)   Node type 1   (10)

            = U if the node is user defined

            = P if the node is generated by the program

3)   Node type 2   (12)

            = R regular coordinate

            = H hole or insert coordinate

            = S sampling point coordinate

4)   Node ID   (16 - 21)

5)   x coordinate   (23 - 34)

6)   y coordinate   (36 - 47)

7)   z coordinate   (49 - 60)

```
   1     2 3    4       5           6           7
  GMR   NODE  NODE      X           Y           Z
  ID    TYPE   ID    COORDINATE  COORDINATE  COORDINATE
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC C C    xxxxxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)

    x - any valid number (0 - 9)

    s - is the sign of the mantisa and the exponent

    . - is the decimal point and it is required

    E - is an exponent sign and it is required

BOUNDARY CONNECTIVITY (CARD 1)

The first boundary element connectivity card has the following format:

		Column Fields
1)	Surface ID	(1 - 8)
2)	Number of geometric points in element	(9)
3)	Element type (special element indicator)	(10)
	= R for regular element	
4)	Element type (functional variation)	(11)
	= B for linear	
	= C for quadratic Serendipity	
	= D for quadratic Lagrangian	
	= G for quartic Serendipity	
	= H for quartic Lagrangian	
5)	Boundary element ID number	(13 - 17)
6)	ID number of first node of boundary element	(19 - 24)
7)	ID number of second node of boundary element	(26 - 31)
8)	ID number of third node of boundary element	(33 - 38)
9)	ID number of fourth node of boundary element	(40 - 45)
10)	ID number of fifth node of boundary element	(47 - 52)
11)	ID number of sixth node of boundary element	(54 - 59)
12)	ID number of seventh node of boundary element	(61 - 66)
13)	ID number of eighth node of boundary element	(68 - 73)
14)	Continuation flag	(80)

```
    1       2  3 4    5      6       7       8       9       10      11      12      13   14
   SURF.   NO ELEM   ELEM   NODE    NODE    NODE    NODE    NODE    NODE    NODE    NODE
    ID     PT TYP     ID     1       2       3       4       5       6       7       8    C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC xx C C    xxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx*
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
   x - any valid number (0-9)
   * - either * or blank

BOUNDARY CONNECTIVITY (CARD 2)

The second boundary element connectivity card has the following format:

```
   1      2      3      4      5      6      7      8      9     10     11 12
  NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE
   9      10     11     12     13     14     15     16     17     18     19  C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx*
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where C - any valid character (A - Z or a - z or 0 - 9 or SPACE)

    x - any valid number (0-9)

    * - either * or blank

Note: May be continued on third card if necessary using the format of card 2.

FIBER INSERT ELEMENT CONNECTIVITY (CARD 1)

The first hole/insert connectivity card has the following format:

		Column Fields
1)	Hole/insert indicator (= HOLE)	(1 - 4)
2)	Number of user defined geometric point in the insert element	(6 - 7)
3)	Element type (number of functional variational nodes in the circumferential direction)	(9)
4)	Element type (functional variation in axial direction) = L for linear = Q for quadratic	(11)
5)	Hole element ID number	(15 - 19)
6)	ID number of first node of hole element	(21 - 26)
7)	ID number of second node of hole element	(28 - 33)
8)	ID number of third node of hole element	(35 - 40)
9)	Radius of hole/insert	(42 - 53)
10)	Continuation flag (= * if the number of functional variational nodes in the circumferential direction (item #3) is greater than 1)	(80)

```
1     2   3 4    5      6      7      8       9                                                    10
HOLE  NO  ELEM   ELEM   NODE   NODE   NODE    RADIUS
DEF.  PT  TYPE   ID     1      2      3       OF HOLE                                              c
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
HOLE xx C C    xxxxx xxxxxx xxxxxx xxxxxx  s.xxxxxxEsxx                                            *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
   C - any valid character (A - Z or a - z or O - 9 or SPACE)
   x - any valid number (0 - 9)
   s - is the sign of the mantisa and the exponent
   . - is the decimal point and it is required
   E - is an exponent sign and it is required
   * - either * or blank

FIBER INSERT CONNECTIVITY (CARD 2)

If the hole/insert has a trigonometric functional variation in the circumferential direction, the connectivity (generated by the program) is defined in the following format:

		Column Fields
1)	Number of total functional variational nodes in this element	(1 - 2)
2)	ID number of first node	(4 - 9)
3)	ID number of second node	(11 - 16)
4)	ID number of third node	(18 - 23)
5)	ID number of fourth node	(25 - 30)
6)	ID number of fifth node	(32 - 37)
7)	ID number of sixth node	(39 - 44)
8)	ID number of seventh node	(46 - 51)
9)	ID number of eighth node	(53 - 58)
10)	ID number of ninth node	(60 - 65)

```
1    2      3      4      5      6      7      8      9      10
NO   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE
PT   1      2      3      4      5      6      7      8      9
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
XX XXXXXX XXXXXX XXXXXX XXXXXX XXXXXX XXXXXX XXXXXX XXXXXX XXXXXX
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

    x - any valid number (0-9)

The sampling surface element connectivity card has the following format:

<u>Column Fields</u>

1)   Sampling surface ID   (1 - 8)
2)   Number of geometric points in the element   (10)
3)   Sampling surface element ID   (12 - 17)
4)   ID number of first node of element   (19 - 24)
5)   ID number of second node of element   (26 - 31)
6)   ID number of third node of element   (33 - 38)
7)   ID number of fourth node of element   (40 - 45)
8)   ID number of fifth node of element   (47 - 52)
9)   ID number of sixth node of element   (54 - 59)
10) ID number of seventh node of element   (61 - 66)
11) ID number of eighth node of element   (68 - 73)
12) ID number of ninth node of element   (75 - 80)

```
    1     2   3      4      5      6      7      8      9      10     11     12
S-SURF.  NO. ELEM.  NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE   NODE
   ID.   PT   ID     1      2      3      4      5      6      7      8      9
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC x xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx xxxxxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where:

    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
    x - any valid number (0-9)

8.4 | RESULT DESCRIPTION

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

8.4.1 | RESULT HEADER

The result header consists of two subsequent records. Each time step has a result reader which contains result indexing information.

RESULT HEADER (CARD 1)

The first result header card has the following format:

<u>Column Fields</u>

1) Time step number: (1 - 4)
2) Time: (6 - 17)
3) Starting record number for boundary results: (22 - 27)
(=0 if not present)
4) Starting record number for element load results: (29 - 34)
(=0 if not present)
5) Starting record number for nodal stress results: (36 - 41)
(=0 if not present)
6) Record number of result header for next time step (74 - 79)
(= 0 for last time step)
7) Continuation flag: (80 - 80)

```
     1         2              3      4     5                                           6    7
   TIME                     1ST    1ST   1ST                                         NEXT
   STEP      TIME           BOUND   LOAD  NODAL                                       HEADER
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
   xxxx  s.xxxxxxEsxx      xxxxxx xxxxxx xxxxxx                                      xxxxxx*
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
x - any valid number (0 - 9)
s - is the sign of the mantisa and the exponent
. - is the decimal point and it is required
E - is an exponent sign and it is required
* - indicates the result header is continued on the next record

RESULT HEADER (CARD 2)

The second card of the result header has the following format:

		Column Fields
1)	Starting record number for boundary element results same as #3, card 1: (=0 if not present)	(2 - 7)
2)	Number of records for boundary element results:	(9 - 13)
3)	Starting record number for hole element results same as: (=0 if not present)	(15 - 20)
4)	Number of records for hole element results:	(22 - 26)
5)	Starting record number for boundary element load results same as #4, card 1: (=0 if not present)	(41 - 46)
6)	Number of records for boundary element load results:	(48 - 52)
7)	Starting record number for hole element load results: (=0 if not present)	(54 - 59)
8)	Number of records for hole element load results:	(61 - 65)
9)	Starting record number for nodal based results Same as #5, card 1. (= 0 if not present)	(67 - 72)
10)	Number of records for nodal based results:	(79 - 83)

```
   1ST              1ST                   1ST          1ST         1ST
  BOUND   NOR     HOLE   NOR            B-LOAD  NOR   H-LOAD  NOR  NODAL     NOR
    1       2       3      4               5      6      7      8     9       10
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
XXXXXX XXXXX XXXXXX XXXXX             XXXXXX XXXXX XXXXXX XXXXX XXXXXX XXXXX
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0 - 9)

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

The boundary element scalar results of temperature, flux, and/or normal displacement have the following format:

		Column Fields
1)	GMR ID:	(1 - 8)
2)	Surface ID (Not available):	(10 - 17)
3)	Element ID:	(20 - 24)
4)	Node ID:	(26 - 30)
5)	Temperature:	(32 - 43)
6)	Flux:	(45 - 56)
7)	Normal displacement:	(58 - 69)
8)	Link the next record identifier:	(80 - 80)

```
    1         2        3     4       5              6              7                  8
   GMR      SURFACE   ELEM  NODE  TEMPERATURE     FLUX          NORMAL             LINK
   ID        ID       ID    ID                                  DISPLACEMENT       ID
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC CCCCCCCC  xxxxx xxxxx s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx           *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
    x - any valid number (0 - 9)
    s - is the sign of the mantisa and the exponent.
    . - is the decimal point and it is required.
    E - is an exponent sign and it is required.
    * - either * or blank

NOTE: The '*' character at the 80th column position will identify if the next record of data is 'LINKED' to the current record. This occurs when the X - Y - Z vectors for displacement & traction are present.

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

NOTE: If a link '*' was present on the previous record, then this record contains vector data. Based on the dimensionality of the problem the vector may or may not be present. If the vector doesn't exist, due to the nature of the problem, then the field will be blank.

The boundary element vector results of displacement and tractions have the following format:

<u>Column Fields</u>

1)	X-vector displacement:	(2 - 13)
2)	Y-vector displacement:	(15 - 26)
3)	Z-vector displacement:	(28 - 39)
4)	X-vector traction:	(41 - 52)
5)	Y-vector traction:	(54 - 65)
6)	Z-vector traction:	(67 - 78)

```
         1             2             3            4            5           6
     X-VECTOR      Y-VECTOR      Z-VECTOR     X-VECTOR     Y-VECTOR    Z-VECTOR
   DISPLACEMENT  DISPLACEMENT  DISPLACEMENT   TRACTION     TRACTION    TRACTION
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
   s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx  s.xxxxxxEsxx
   ----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where

C - any valid character (A - Z or a - z or 0 - 9 or SPACE)

x - any valid number (0 - 9)

s - is the sign of the mantisa and the exponent.

. - is the decimal point and it is required.

E - is an exponent sign and it is required.

* - either * or blank

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

The hole containing the fiber insert element scalar results of temperature, flux and/or normal displacement have the following format:

		Column Fields
1)	GMR ID:	(1 - 8)
2)	Hole indicator (= HOLE):	(10 - 17)
3)	Element ID:	(20 - 24)
4)	Node ID:	(26 - 30)
5)	Temperature:	(32 - 43)
6)	Flux:	(45 - 56)
7)	Link the next record identifier:	(80 - 80)

```
    1        2      3     4      5              6                                                    7
  GMR      HOLE    ELEM  NODE  TEMPERATURE     FLUX                                                LINK
   ID     (TUBE)    ID    ID                                                                         ID
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC HOLE          xxxxx xxxxx s.xxxxxxEsxx  s.xxxxxxEsxx                                         *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
    x - any valid number (0 - 9)
    s - is the sign of the mantisa and the exponent.
    . - is the decimal point and it is required.
    E - is an exponent sign and it is required.
    * - either * or blank

NOTE: The '*' character at the 80th column position will identify if the next record of data is 'LINKED' to the current record. This occurs when the X - Y - Z vectors for displacement & traction are present.

| 8.4.5 | FIBER INSERT ELEMENT VECTOR RESULTS |

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

> NOTE: If a link '*' was present, then the next record of data is is the vector data. Based on the dimensionality of the problem the vector may or may not be present. If the vector doesn't exist, due to the nature of the problem, then the field will be blank.

The hole containing the fiber insert element vector results of displacement and traction have the following format:

<u>Column Fields</u>

1)	X-vector displacement:	(2 - 13)
2)	Y-vector displacement:	(15 - 26)
3)	Z-vector displacement:	(28 - 39)
4)	X-vector traction:	(41 - 52)
5)	Y-vector traction:	(54 - 65)
6)	Z-vector traction:	(67 - 78)

```
        1               2               3               4               5               6
    X-VECTOR        Y-VECTOR        Z-VECTOR        X-VECTOR        Y-VECTOR        Z-VECTOR
  DISPLACEMENT    DISPLACEMENT    DISPLACEMENT      TRACTION        TRACTION        TRACTION
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx   s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
> C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
> x - any valid number (0 - 9)
> s - is the sign of the mantisa and the exponent.
> . - is the decimal point and it is required.
> E - is an exponent sign and it is required.
> * - either * or blank

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

The boundary element energy and load balance calculation has the following format:

		Column Fields
1)	GMR ID:	(1 - 8)
2)	Surface ID (Not available):	(10 - 17)
3)	Element ID:	(20 - 24)
4)	Number of nodes per this element:	(26 - 27)
5)	Energy rate:	(29 - 40)
6)	X-vector load rate:	(42 - 53)
7)	Y-vector load rate:	(55 - 66)
8)	Z-vector load rate:	(68 - 79)

```
    1           2        3    4      5            6              7              8
   GMR       SURFACE   ELEM # OF   ENERGY      X-VECTOR       Y-VECTOR       Z-VECTOR
    ID         ID       ID NODES    RATE         LOAD           LOAD           LOAD
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC CCCCCCCC  xxxxx xx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxE.xx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
  C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
  x - any valid number (0 - 9)
  s - is the sign of the mantisa and the exponent.
  . - is the decimal point and it is required.
  E - is an exponent sign and it is required.

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

Nodal based results are available at all boundary element nodes and at all sampling points.

The format for this card is the following:

		Column Fields
1)	GMR ID:	(1 - 8)
2)	Node ID number:	(10 - 15)
3)	Temperature:	(29 - 40)
4)	Continuation flag (= * if displacements are provided):	(80)

```
    1        2                         3                                                                    4
  GMR      NODE
   ID       ID                     TEMPERATURE                                                              C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
CCCCCCCC xxxxxx              s.xxxxxxEsxx                                                                   *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    C - any valid character (A - Z or a - z or 0 - 9 or SPACE)
    x - any valid number (0 - 9)
    s - is the sign of the mantisa and the exponent.
    . - is the decimal point and it is required.
    E - is an exponent sign and it is required.
    * - either * or blank

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

Nodal based results are available at all boundary element nodes and at all sampling points.

If nodal based displacements are provided, the card will have the following format:

<u>Column Fields</u>

1)	Displacement in x direction:	(2 - 13)
2)	Displacement in y direction:	(15 - 26)
3)	Displacement in z direction:	(28 - 39)
4)	Continuation flag (= * if stresses are provided):	(80)

```
        1             2             3                                                                      4
DISPLACEMENT DISPLACEMENT DISPLACEMENT
        X             Y             Z                                                                      C
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx                                                                     *
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0 - 9)
    s - is the sign of the mantisa and the exponent.
    . - is the decimal point and it is required.
    E - is an exponent sign and it is required.

The PRINT and CALC cards in **CASE INPUT determine which type of results will be provided in the BEST-CMS neutral file.

Nodal based results are available at all boundary element nodes and at all sampling points.

The format of the nodal based stress results is given below.

NOTE: Nodal based displacement results are always provided if nodal based stress results are provided.

		Column Fields
1)	Stress component σ_{xx}:	(2 - 13)
2)	Stress component σ_{yy}:	(15 - 26)
3)	Stress component σ_{zz}:	(28 - 39)
4)	Stress component σ_{xy}:	(41 - 52)
5)	Stress component σ_{xz}:	(54 - 65)
6)	Stress component σ_{yz}:	(67 - 78)

```
      1               2               3               4               5               6
   STRESS          STRESS          STRESS          STRESS          STRESS          STRESS
     XX              YY              ZZ              XY              XZ              YZ
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
 s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx s.xxxxxxEsxx
----+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

where
    x - any valid number (0 - 9)
    s - is the sign of the mantisa and the exponent.
    . - is the decimal point and it is required.
    E - is an exponent sign and it is required.

9.0 | REFERENCES

Aboudi, J., Pindera, J.J. and Arnold, S.M., 'Thermoelastic Response to Metal Matrix Composites with Large Diameter Fibers Subjected to Thermal Gradients', NASA TM-106344 (1993).

Abramowitz, M. and Stegun, I.A., Handbook of Mathematical Functions, Dover, New York (1974).

Ballarini, R. and Shamin, A., 'Finite Element Modeling of Frictionally Restrained Composite Interfaces', NASA Contractor Report 182281 (April 1989).

Banerjee, P.K., The Boundary Element Methods in Engineering, McGraw Hill, London and New York, 1993.

Banerjee, P.K., 'Integral Equation Methods for Analysis of Piece-wise Non-homogeneous Three-dimensional Elastic Solids of Arbitrary Shape', Int. J. Mech. Sci., 18, pp. 293-303 (1976).

Banerjee, P.K., Ahmad, S. and Manolis, G.D., 'Transient Elastodynamic Analysis of Three-dimensional Problems by Boundary Element Method', Earthquake Engineering and Structural Dynamics, Vol. 14, pp. 933-949 (1986).

Banerjee, P.K. and Butterfield, R., Boundary Element Methods in Engineering Science, McGraw-Hill, London (1981) (Russian Translation, Mir Publishers, Moscow 1994; Chinese Translation, National Defense Press, Beijing, 1988).

Banerjee, P.K., The Boundary Element Methods in Engineering, McGraw Hill, London and New York, 1994.

Banerjee, P.K. and Butterfield, R., Developments in Boundary Element Methods - I, Applied Sci. Publishers, Barking, Essex, UK (1979).

Banerjee, P.K. and Driscoll, R.M., 'Three-dimensional Analysis of Raked Pile Groups', Proc. of Inst. Civil Engineers, Research and Theory, Vol. 61, pp. 653-671 (1976).

Banerjee, P.K. and Henry, D.P., 'Development of BEM for Thermal Stress Analysis of Ceramic Composites', Proceedings of the 2nd Annual HITEMP Review, NASA Lewis Research Center, Cleveland, Ohio (1989).

Banerjee, P.K. and Henry, D. Analysis of Three-dimensional Solids with Holes by BEM, to appear in Int. Jour. Numer. Methods in Engrg. (1991).

Banerjee, P.K. and Henry, D.P., 'Elastic Analysis of Three-dimensional Solids with Fiber Inclusions by BEM,' Int. Jour. of Solids and Structures, Vol. 29, No. 20, pp. 2423-2440 (1992).

Banerjee, P.K. and Henry, D.P., 'Nonlinear Micro and Macromechanical Analysis of Composites by BEM', Proceedings of the Annual HITEMP Review, NASA Conf. Pub. 10104 (1992).

Banerjee, P.K. and Henry, D.P., 'Micro and Macromechanical Analysis of Composites by BEM, Constitutive Behavior of High Temperature Composites', ASME Annual Winter Meeting, November (1992).

Banerjee, P.K. and Henry, D.P., 'Further Development of BEM for Micro and Macromechanical Analysis of Composites', Proceedings of the Annual HITEMP Review, NASA Conf. Pub. 19117 (1993).

Banerjee, P.K. and Henry, D.P., 'Micro and Macro Mechanics of Woven Fiber Composites', HITEMP Review, NASA Conf. Pub. 10146 (1994).

Banerjee, P.K. and Henry, D.P., 'Development of BEM for Micro and Macromechanical Studies of Woven Composites, Paper 50, HITEMP Review, NASA Conf. Pub. 10178 (1995).

Banerjee, P.K., Henry, D.P. and Dargush, G.F., 'Further Development of BEM for Thermomechanical Study of Ceramic Composites', Proceedings of the 3rd Annual HITEMP Review, NASA Lewis Research Center, Cleveland, Ohio (1990).

Banerjee, P.K., Henry, D.P. and Dargush, G.F., 'Nonlinear Thermomechanical Analysis of Ceramic Composites by BEM', Proceedings of the 4th Annual HITEMP Review, NASA Lewis Research Center, Cleveland, Ohio (1991).

Banerjee, P.K., Henry, D.P. and Dargush, G.F., 'Micromechanical Studies of Composites by BEM, Enhancing Analysis Techniques for Composite Materials', Proc. ASME Winter Annual Meeting, Atlanta (1991).

Banerjee, P.K., Henry, D.P., Dargush, G.F. and Hopkins, D.A., Boundary Element Solution Technology, Composite Modeling System; BEST-CMS User Manual, Version 2.0, NASA Lewis Research Center (1991).

Banerjee, P.K., Henry, D.P., Dargush, G.F., Hopkins, D.A. and Goldberg, R.K., BEST-CMS User's Manual, Version 3.0 Computational Engineering Mechanics Laboratory, Department of Civil Engineering, State University of New York at Buffalo (1994).

Banerjee, P.K. and Mukherjee, S., Developments in Boundary Element Methods - III, Applied Sci. Publishers, Barking, Essex, UK (1984).

Banerjee, P.K. and Shaw, R.P., Developments in Boundary Element Methods - II, Applied Sci. Publishers, Barking, Essex, UK (1982).

Banerjee, P.K. and Watson, J.O., Editors, Developments in Boundary Element Methods IV, Elsevier Applied Science Publishers, Barking, Essex, U.K. (1986).

Banerjee, P.K., Wilson, R.B. and Miller, N., 'Development of Large BEM System for Three-dimensional Inelastic Analysis', in Advanced Topics in Boundary Element Analysis, ed. T.A. Cruse, A.B. Pifko and H. Armen, AMD-V72, ASME, New York (1985).

Banerjee, P.K., Wilson, R.B. and Miller, N., 'Advanced Elastic and Inelastic Stress Analysis of Gas Turbine Engine Structures by BEM, Int. Jour. Numer. Methods in Engrg., Vol. 26, pp. 393-411 (1988).

Behrens, E., 'Thermal Conductivities of Composite Materials,' J. Composite Materials, Vol. 2, p. 2 (1968).

Bhatt, H., Donaldson, D.Y., Hasselman, D.P.H. and Bhatt, R.T., 'Role of Interfacial Carbon Layer in the Thermal Diffusivity/Conductivity of Silicon Carbide Fiber-Reinforced Reaction-Bonded Silicon Nitride Matrix Composites', J. Am. Ceram., Vol. 2, pp. 334-340 (1992).

Butterfield, R. and Banerjee, P.K., 'Analysis of Axially Loaded Compressible Piles and Pile Groups', Geotechnique, Vol. 21, No. 1, pp. 43-60 (1971).

Byrd, P.F. and Friedman, M.D., Handbook of Elliptic Integrals for Engineers and Physicists, Springer-Verlag, Berlin (1954).

Caruso, J.J. and Chamis, C.C., 'Assessment of Simplified Composite Micromechanics Using Three-dimensional Finite-element Analysis', J. Comp. Tech. Res., Vol. 8, No. 8, pp. 77-83 (1986).

Chamis, C.C. and Sinclair, J.H., 'The Effects of Eccentricities on Fracture of Off-axis Fiber Composites', Polym. Eng. and Sci., Vol. 19, No,. 5 (1979).

Chen, W.F., Limit Analysis and Soil Plasticity, Elsevier, New York (1975).

Cruse, T.A.,' An Improved Boundary Integral Equation Method for Three-dimensional Elastic Stress Analysis', Comp. and Struct., V4, pp. 741-754 (1974).

Dargush, G.F., 'Boundary Element Methods for the Analogous Problem of Thermomechanics and Soil Consolidation', Ph.D. Thesis, State University of New York at Buffalo (1987).

Dargush, G. and Banerjee, P.K. Development of an Integrated BEM for Hot Fluid Structure Interaction, Earth to Orbit Propulsion System, NASA Conf. Publication No. 3012 (1988).

Dargush. G.F. and Banerjee, P.K., 'Advanced Development of the Boundary Element for Steady-state Heat Conduction', Int. J. Num. Methods in Eng., Vol. 28, pp. 2123-2142 (1989).

Dargush, G.F. and Banerjee, P.K., 'BEM Analysis for Three-dimensional Problems of Transient Thermoelasticity', Int. Jour. Solids and Structures, Vol. 26, pp. 199-216 (1990).

Dargush, G. and Banerjee, P.K., 'Application of the Boundary Element Method to Transient Heat Conduction', Int. Jour. Numer. Methods in Engrg., Vol. 31, to appear (1991).

Davidge, R.W., 'Fibre-reinforced Ceramics', Composites, Vol. 18, No. 2, pp. 92-97 (1987).

Dongarra, J.J. et al, Linpack Users' Guide, SIAM, Philadelphia, Pa (1979).

Garg, K.S., Svalbonas, V. and Gutman, G., Analysis of Structural Composite Materials, M. Decker, New York (1973).

Goldberg, R. and Comisky, M., COMGEN-BEM, Boundary Element Model Generation for Composite Material Micromechanical Analysis, NASA Lewis Research Center.

Goldberg, R.K. and Hopkins, D.A., 'Utilization of Boundary Element Modeling Interface Effects in SiC/RBSN Composites', HITEMP Review, Advanced High Temperature Engine Materials Technology Program, NASA CP-19117, pp. 74-1 to 74-10 (1993).

Goldberg, R.K. and Hopkins, D.A., 'Utilization of BEM in Modeling Thermoelastic Behavior of SiC/RBSN and Functionally Graded SiC/TI-16-3 Composites', 7th NASA HITEMP Conference, pp. 69-1 to 69-3 (1994).

Goldberg, R and Hopkins, D., 'Application of BEM to Micromechanical Analysis of Composite Material, Computers and Structures', Vol. 56, No. 2, pp. 721-731 (1995).

Goldberg, R. and Hopkins, D., 'Thermal Analysis of a Functionally Graded Material Subject to Thermal Gradient using BEM', Composite Engineering, Vol. 5, No. 7, pp. 793-806 (1995).

Hashin, Z. and Rosen, B.W., 'The Elastic Moduli of Fiber Reinforced Materials', J. Appl. Mech., Vol. 31, p. 223 (1964).

Hastings, C., Jr., Approximations for Digital Computers, Princeton University Press, Princeton (1955).

Henry, D.P., 'Advanced Development of the Boundary Element Method for Elastic and Inelastic Thermal Stress Analysis', Ph.D. dissertation, State University of New York at Buffalo (1987).

Henry, D. and Banerjee, P.K., 'A Thermoplastic BEM Analysis for Substructured Axisymmetric Bodies', Journal of Engineering Mechanics, ASCE, Vol. 113, No. 12, pp. 1880-1900 (1987).

Henry, D. and Banerjee, P.K., 'A New Boundary Element Formulation for Two and Three-dimensional problems of Thermoelasticity Using Particular Integrals', Int. Jour. Numer. Methods in Engrg., Vol. 26, pp. 2061-2077 (1988a).

Henry, D.P. and Banerjee, P.K., 'A Variable Stiffness Type Boundary Element Formulation for Axisymmetric Elastoplastic Media', Int. Jour. Numer. Methods in Engrg., Vol 26, No. 5, pp. 1005-1027 (1988b).

Henry, D.P. and Banerjee, P.K. 'Elastic Stress Analysis of Three-dimensional Solids with Small Holes by BEM', Int. Jour. Numer. Methods in Engrg., Vol. 31, pp. 369-384 (1991).

Hill, R., 'Theory of Mechanical Properties of Fibre-strenghtened Materials: III. Self-consistent Model', J. Mech. Phys. Solids, Vol. 13, p. 189 (1965).

Hopkins, D.A. and Chamis, C.C., 'A Unique Set of Micromechanics Equation for High Temperature Metal Matrix Composites', First Symposium on Testing Technology of Metal Matrix Composites, Nashville, Tenessee, NASA Technical Memorandum 87154 (November 1985).

Jones, R.M., Mechanics of Composite Materials, McGraw-Hill (1975).

Lachat, J.C. and Watson, J.O. 'Effective Numerical Treatment of Boundary Integral Equations: A Formulation for Three-dimensional Elastostatics', Int. J. Num. Meth. in Eng., 10, pp., 991-1005 (1976).

McCullough, R.L., Concepts of Fiber-resin Composites, M. Decker, New York (1971).

Rizzo, F.J. and Shippy, D.J., 'A Formulation and Solution Procedure for the General Non-homogeneous Elastic Inclusion Problems', Int. J. Solids and Struct., 4, pp. 1161-1179 (1968).

Rizzo, R.R., 'More On the Influence of End Constraints on Off-axis Tensile Tests', J. Composite Materials, Vol. 3, p. 202 (1969).

Selvadurai, A.P.S., 'Some Aspects of Non-linear Interfaces in Geomechanics: Boundary Element Modelling', Third International Conference in Geomechanics, Niagara Falls, Canada, p. 463 (May 1988).

Sen, R., Kausel, E. and Banerjee, P.K., 'Dynamic Behavior of Axially and Laterally Loaded Piles and Pile Groups Embedded in Inhomogeneous Soil', Int. Jour. Numerical Analytical Methods in Geomechanics, Vol. 9, No. 6, pp. 507-524 (1985).

Stroud, A.H. and Secrest, D., <u>Gaussian Quadratic Formulas</u>, Prentice-Hall, New York (1966).

Tripp, D.E., Hermann, J.H., and Gyekenyesi, J.P., 'A Review of Failure Models for Uni-directional Ceramic Composites Under Monotonic Loads', NASA Technical Memorandum 101421 (June 1989).

Vinson, J.R. and Chou, T.W., <u>Composite Materials and Their Use in Structures</u>, John Wiley and Sons (1975).

Walker, K.P., Research and Development Program for Nonlinear Structural Modeling with Advanced Time-temperature Dependent Constitutive Relationships, NASA CR-165533 (1981).

Walker, K.P., Jordon, E.H., and Freed, A.D., 'Nonlinear Mesomechanics of Composites with Periodic Microstructure: First Report', NASA Technical Memorandum 102051 (June 1989).

Wang, H.C., 'A General Purpose Development of BEM for Axisymmetric Solids', Ph.D. Dissertation, State University of New York at Buffalo (1989).

Watson, J.O., 'Advanced Implementation of the Boundary Element Method for Two- and Three-dimensional Elastostatics', in Banerjee, P.K. and Butterfield, R., <u>Developments in Boundary Element Methods - I</u>, Applied Science Publishers, London, pp. 31-64 (1979).

Whitney, J.M. and Riley,, M.B., 'Elastic Properties of Fiber Reinforced Composite Materials', J. AIAA, Vol. 4, p. 1537 (1966).

Wilson, R.B., Bak, M.J., Nakazawa, S., and Banerjee, P.K., '3-D Inelastic Analysis Method for Hot Section Components (Base Program)', NASA Contract Report 174700 (February 1984).

Wilson, R.B., Snow, D.W. and Banerjee, P.K. 'Stress Analysis of Gas Turbine Engine Structure Using Boundary Element Methods, AMD-Vol. 72, pp. 45-64 (1985).

BEST-CMS User's Guide Comment Sheet (Release 4.0)

NAME: _____ DATE: _____

AFFILIATION: _____

ADDRESS: _____

TELEPHONE: _____

1. ERRORS/CORRECTIONS (Specify page, figure number, etc.)

_____

_____

_____

_____

_____

2. SUGGESTIONS/COMMENTS (e.g., overall manual organization and contents, future

ideas, need for other manuals/primers, complaints, etc.)

_____

_____

_____

_____

_____

_____

Please return to:

Dr. D.P. Henry or Dr. P.K. Banerjee
Computational Engineering Mechanics Laboratory
Department of Civil Engineering
State University of New York at Buffalo
204 Ketter Hall, Buffalo, NY 14260, USA

or phone (716) 645-2198