BEST3D User's Manual

Boundary Element Solution Technology, 3-Dimensional Version 3.0

Computational Engineering Mechanics Laboratory
Department of Civil Engineering
State University of New York at Buffalo

United Technologies Corporation
Pratt & Whitney
Commercial Engineering
East Hartford, Connecticut

March 1991

Prepared for
Lewis Research Center
Under Contract NAS3-23697

NASA Contractor Report 185298

National Aeronautics and Space Administration
BEST3D
USER'S MANUAL

Boundary Element Solution Technology 3Dimensional

Version 3.0
March 1991

P. K. Banerjee and R. B. Wilson
ACKNOWLEDGEMENTS

It should be emphasised that a large undertaking such as the development of BEST3D is possible only through the collective efforts of a large number of people. To Dr. Chris Chamis and Mr. Dale Hopkins of NASA Lewis Center, Cleveland, who had the vision to conceive this project, and to Dr. Edward Todd of Pratt & Whitney for inspired leadership we express our sincere gratitude. A very significant contribution to the development effort was made by David Snow of Pratt & Whitney, who had the major responsibility for the validation examples and Nancy Miller who developed substantial part of the utility routines.

To Drs. T. G. Davies and G. D. Manolis of the Department of Civil Engineering, State University of New York at Buffalo, who carried out the early part of the investigations for the inelastic and dynamic analyses, respectively, we express our deepest appreciation for their work. A number of former and present graduate students (at SUNY, Buffalo) of exceptional academic ability and programming skill contributed substantially to the development of BEST3D. In particular we are indebted to Drs. S.T. Raveendra and D. Henry for their major contributions in inelastic analyses, to Dr. N. B. Yousif for her contributions in inelastic constitutive equations, to Dr. R. Sen for the early part of the work on the transient dynamic analyses, to Drs. S. Ahmad and Wang for their major contributions in periodic dynamic, transient time domain dynamic analysis algorithms, and to Dr. G. Dargush for his major contributions in thermal problems.
<table>
<thead>
<tr>
<th>SECTION</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0 - Introduction</td>
<td>1.1</td>
</tr>
<tr>
<td>2.0 - Theoretical Background</td>
<td>2.1</td>
</tr>
<tr>
<td>2.1 - Quasi-Static Elastic Analysis</td>
<td>2.2</td>
</tr>
<tr>
<td>2.2 - Inelastic Analysis</td>
<td>2.6</td>
</tr>
<tr>
<td>2.3 - Free-Vibration Analysis</td>
<td>2.13</td>
</tr>
<tr>
<td>2.4 - Periodic Dynamic Analysis</td>
<td>2.15</td>
</tr>
<tr>
<td>2.5 - Transient Dynamic Analysis</td>
<td>2.16</td>
</tr>
<tr>
<td>2.6 - Heat Conduction Analysis</td>
<td>2.18</td>
</tr>
<tr>
<td>2.7 - Stress Analysis with Embedded Holes</td>
<td>2.20</td>
</tr>
<tr>
<td>2.8 - Stress Analysis of Bodies with Composite Inserts</td>
<td>2.25</td>
</tr>
<tr>
<td>3.0 - Numerical Implementation</td>
<td>3.1</td>
</tr>
<tr>
<td>3.1 - Representation of Geometry and Functions</td>
<td>3.2</td>
</tr>
<tr>
<td>3.2 - Surface and Volume Integration</td>
<td>3.4</td>
</tr>
<tr>
<td>3.3 - System Equations</td>
<td>3.5</td>
</tr>
<tr>
<td>3.3.1 - Assembly of Equation for Composite Inserts</td>
<td>3.7</td>
</tr>
<tr>
<td>3.4 - Equation Solution</td>
<td>3.9</td>
</tr>
<tr>
<td>3.5 - Interior Results</td>
<td>3.10</td>
</tr>
<tr>
<td>3.6 - Accuracy and Efficiency</td>
<td>3.11</td>
</tr>
<tr>
<td>4.0 - Guide to Using BEST3D</td>
<td>4.1</td>
</tr>
<tr>
<td>4.1 - Getting Started</td>
<td>4.2</td>
</tr>
<tr>
<td>4.2 - Availability of BEST3D</td>
<td>4.3</td>
</tr>
<tr>
<td>4.3 - Filesystem in BEST3D</td>
<td>4.4</td>
</tr>
<tr>
<td>4.4 - Special Features of BEST3D</td>
<td>4.5</td>
</tr>
<tr>
<td>4.4.1 - Definitions</td>
<td>4.5</td>
</tr>
<tr>
<td>4.4.2 - Mesh Size</td>
<td>4.8</td>
</tr>
</tbody>
</table>
5.3.3 - Nodal Point Definition ........................................ 5.76
5.3.4 - Surface Element Definition ................................... 5.78
5.3.5 - Enclosing Element Definition ................................. 5.95
5.3.6 - Hole Element Definition ..................................... 5.97
5.3.7 - Insert Element Definition ................................... 5.103
5.3.8 - Volume Cell Definition ...................................... 5.110
5.3.9 - Global Shape Function Definition ......................... 5.117
5.3.10 - Sampling Point Definition ................................ 5.120

5.4 - Definition of GMR Compatibility ............................... 5.122
5.4.1 - Interface Definition Input card .............................. 5.124
5.4.2 - Definition of Interface Surface 1 ......................... 5.126
5.4.3 - Definition of Interface Surface 2 ......................... 5.129
5.4.4 - Type of Interface Condition ................................. 5.132
5.4.5 - Cyclic Symmetry Parameter Definition ..................... 5.136
5.4.6 - Additional Interface Control Options ...................... 5.140

5.5 - Definition of Boundary Conditions ............................ 5.141
5.5.1 - Boundary Condition Set Card ................................. 5.143
5.5.2 - Boundary Condition Identification ......................... 5.144
5.5.3 - Identification of Boundary Condition Type ................ 5.145
5.5.4 - Definition of Surface for Application of Boundary Conditions 5.148
5.5.5 - Value Boundary Conditions for Surface Elements .......... 5.156
5.5.6 - Definition of Holes for Application of Boundary Conditions 5.162
5.5.7 - Value Boundary Conditions for Hole Elements ............. 5.167
5.5.8 - Definition Space/Time Variation ........................... 5.170
5.5.9 - Relation Boundary Condition ............................... 5.173

5.6 - Body Force Definition ........................................... 5.177
5.6.1 - Body Force Input card ....................................... 5.178
5.6.2 - Centrifugal Body Force .................................... 5.180
5.6.3 - Inertial Body Force ....................................... 5.185
BEST3D (Boundary Element Solution Technology in Three Dimensions) is an advanced engineering analysis system based upon the boundary element method (BEM). For many problems, BEM offers significant advantages in accuracy, efficiency, and ease of use over the more familiar finite element method.

The overall BEST3D system has two primary objectives:

- Provide new and/or improved capability for the engineering analyst.
- Provide a complementary analysis technique for the verification of finite element or experimental results.

Consequently, BEST3D has been designed to solve large realistic problems from a wide range of engineering disciplines, utilizing state-of-the-art boundary element technology. Table 1.1 highlights some of the general features of the code, while a brief summary of the capabilities of BEST3D is presented by analysis type in Table 1.2.

The primary objective of this User's Manual is to provide an overview of all BEST3D capabilities, along with detailed descriptions of the input data requirements. As a result, the present manual does not provide a thorough development of the theoretical basis for the various types of analysis, nor does it contain a comprehensive collection of realistic example problems. However, sufficient material is included so that, after receiving this manual an engineering analyst should be able to prepare BEST3D input data sets.

In the next chapter, a brief review of the theoretical background is presented for each analysis category. In most cases, only sketchy details are provided, however references are cited to assist the interested reader. Then, Chapter 3 discusses the key aspects of the numerical implementation, while Chapter 4 provides a tutorial for the beginning BEST3D 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 for all the major analysis types. The complete input data sets are heavily commented, and thus should be of considerable assistance to the novice. Actual extracts from various analysis output options are also provided. Chapter 7 includes capsules of a number of realistic engineering analysis problems that have been solved using BEST3D. This chapter is primarily descriptive in nature and is intended merely to illustrate the level of engineering analysis that is possible within the present BEST3D system. Finally, all pertinent references are listed in Chapter 8.
TABLE 1.1
GENERAL FEATURES OF BEST3D

- Three dimensional analysis
- Conforming element approach to provide inter-element continuity of the field variables, along with efficient solutions
- Substructuring to permit multiple materials and more efficient solutions
- Automatic adaptive numerical integration schemes
- Cyclic and planar symmetry
- Local or global boundary condition specification
- Sliding, spring and resistance-type interfaces
- Exterior domains and infinite elements
- Block banded solver routines based upon LINPACK
- Generalized eigenvalue extraction routines based upon EISPACK
- Restart capability to reuse existing integration coefficients
- Free-format, keyword-driven input
- Automatic input data error checking
TABLE 1.2
ENGINEERING ANALYSIS BY BEST3D

A. Elastic Analysis
- Stress analysis and free-vibration analysis
- Analytic representations of holes in 3-D solids with a minimum of discretization
- Isotropic, as well as anisotropic media
- Centrifugal and self-weight body forces
- Spring loaded or sliding interfaces
- Embedded inserts for analysis of 3-D composites.

B. Dynamic Analysis
- Forced periodic response
- Forced transient response in time domain
- Free-vibration analysis

C. Thermal Stress Analysis
- Steady state linear thermal stress analysis
- Nonlinear (thermoplastic) stress analysis
- Nonlinear thermoviscoelastic stress analysis
- Analytic representations of holes in 3-D solids with a minimum of discretization
- A large family of constitutive models for nonlinear analysis
- Hotspots and discontinuities
TABLE 1.2 (continued)
ENGINEERING ANALYSIS BY BEST3D

D. **Heat Transfer Analysis**
   - Steady state heat transfer
   - Transient heat transfer
   - Analytic representations of cooling holes in 3-D with a minimum of discretization
   - Thermal resistance across subregions

E. **Nonlinear Stress Analysis**
   - Iterative elastoplasticity and viscoplasticity
   - Direct variable stiffness type elastoplasticity
   - Cyclic elastoplasticity
   - Inelastic fracture mechanics analysis
   - Thermoplasticity and thermoviscoplasticity
Theoretical Background

The mathematical background of the boundary element method has been known for nearly 100 years. Indeed some of the boundary integral formulations for elastic, elasto-dynamic, 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 [1] 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 [1-13] which give a very comprehensive and thorough account of the existing literature on the method.

In this section, the different types of analyses incorporated in BEST3D are briefly outlined.
It is well known that, for a homogeneous, elastic material, there exist point force solutions. These solutions provide the displacements at any point in an infinite body due to the application of a unit load at any other point in the body. Closed form solutions are available for isotropic materials (the Kelvin solution) and for transversely isotropic materials. Differentiation and the use of the appropriate elastic stress-strain relationship allow calculation of the stresses, strains and tractions due to the point load.

The Reciprocal Work Theorem

\[ \int_S t_i(x)u_i^*(x)ds + \int_V f_i(x)u_i^*(x)dv = \int_S t_i^*(x)u_i(x)ds + \int_V f_i^*(x)u_i(x)dv \] (2.1.1)

is the integral identity satisfied for any two equilibrium states of tractions \((t)\), displacements \((u)\), and body forces \((f)\) existing in a solid occupying the volume \(V\) with surface \(S\). Using the point load solution as one of the two equilibrium states and the solution of the desired boundary value problem as the second, it is possible to obtain [2]:

\[ u_j^*(\xi) = \int_S [G_{ij}(z, \xi)t_i(z) - F_{ij}(z, \xi)u_i(z)]ds + \int_V G_{ij}(z, \xi)f_i(z)dv \] (2.1.2)

the interior displacement identity expressing the displacement at any interior point in terms of surface integrals of the displacements and tractions and volume integrals of mechanical body forces, thermal strains and initial (inelastic) stresses. Since the point load solution is singular at the load point, special attention is required to limiting operations during the integration.

It is possible to take the load point to the boundary of the body, obtaining:

\[ C_{ij}u_i(\xi) = \int_S [G_{ij}(z, \xi)t_i(z) - F_{ij}(z, \xi)u_i(z)]ds + \int_V G_{ij}(z, \xi)f_i(z)dv \] (2.1.3)

a constraint equation relating the known and unknown displacements and tractions on the surface of the body. Further, differentiation and the use of the elastic stress-strain relations allow the evaluation of strains and stresses at arbitrary points within the body. For example,

\[ \sigma_{ij}(\xi) = \int_S [G_{kij}^o(z, \xi)t_k - F_{kij}^o(z, \xi)u_k(z)]ds + \int_V G_{kij}^o(z, \xi)f_k(z)dv \] (2.1.4)

can be written for interior stresses. In the simplest case, a homogeneous elastic problem without thermal or mechanical body forces, all of the volume integrals vanish and (2.1.3) allows the solution of the boundary value problem entirely in terms of boundary geometry and the values of displacement and traction on the boundary. No reference need be made
to the interior of the body unless calculations of interior values of stress or displacement are desired after the boundary solution has been completed. The boundary element method was first developed for this class of problems, and was referred to in the literature as the BIE (Boundary Integral Equation) method.

The volume integrals corresponding to any steady state thermal loading or conservative mechanical body force can be converted to surface integrals, and such problems can thus be handled within the classical BIE formulation [14,15], although this is not the approach adopted in BEST3D. Rather, all of these effects on linear analyses have been incorporated using particular integrals where no surface integrations are required.

This new approach, based on particular integrals, is adopted in BEST3D for the treatment of centrifugal body forces. The method is unique since no volume integration or additional surface integration is required as in alternative formulations. The method is extended to analyses involving thermal body force loading and initial stress loading (e.g. plasticity). Therefore, in a thermal or nonlinear problem, the user has the option to use either conventional volume integration or the new particular integral approach for incorporating the body force effect into the analysis.

The governing differential equation for a homogenous, isotropic body subjected to body force loading can be expressed in operator notation as [35,38,44,45,52]:

\[ L(u_i) + f_i = 0 \]  \hspace{1cm} (2.1.5)

where

\[ L(u_i) = (\lambda + \mu)u_{ij,j} + \mu u_{ij,j} \]

\( u_i \) represents the displacement vector,

\( (f_1, f_2, f_3) = \rho \omega^2 (x_1, x_2, 0) \) for centrifugal loading about the z axis

\( f_i = -\beta T_i \) for thermoelastic loading, with \( \beta = (3\alpha + 2\mu)\alpha \) and

\( f_i = -\sigma_{ij}^{\alpha} \) for initial stress (nonlinear) problems

When more than one type of body force is present, the net body force is the summation of the individual body forces.

A solution satisfying a linear, inhomogeneous, differential equation and boundary conditions can be obtained using the method of particular integrals if the complete solution of the corresponding homogenous equation is known, provided of course, a particular solution can be found. Therefore, the solution of the above equation can be represented as the sum of a complementary function \( u_i^c \) satisfying the homogenous equation [35,38,44,45,46]

\[ L(u_i^c) = 0 \]  \hspace{1cm} (2.1.6)
and a particular integral $u^p_i$ satisfying the inhomogenous equation

$$L(u^p_i) = f_i$$

(2.1.7)

The total solution $u_i$ is expressed as

$$u_i = u^c_i + u^p_i$$

(2.1.8)

The boundary integral equation satisfying the homogenous part of the differential equation is the complementary function in this procedure. The complementary function for displacement at point $\xi$, is expressed as

$$C_{ij}(\xi)u^c_i(\xi) = \int [G_{ij}(x, \xi)t^c_i(x) - F_{ij}(x, \xi)u^c_i(x)]dS(x)$$

(2.1.9)

where the $u^c_i$ and $t^c_i$ are the complementary functions for displacement and traction, respectively. The total solution for displacement $u_i$ and traction $t_i$ are

$$u_i = u^c_i + u^p_i$$

$$t_i = t^c_i + t^p_i$$

(2.1.10)

where $u^p_i$ and $t^p_i$ are the particular integrals for displacement and traction, respectively. The particular integral is classically found via the method of undetermined coefficients, the method of variation of parameters, or obtained by inspection of the inhomogenous differential equation. In the theory of linear, inhomogenous, differential equations, it is understood that the particular integral is not unique, and any expression satisfying equation (2.1.5) is a particular integral, regardless of boundary conditions or how it was obtained.

Particular integrals for centrifugal loading are relatively simple to derive and are given in many text books on elasticity. On the other hand, particular integrals satisfying the inhomogenous differential equation for thermal and initial stress loading are more complex since the distribution of temperature and initial stress are general in nature. To accommodate this general distribution, a device known as the global shape function is employed. It is then possible to derive the particular integrals for displacement, stress, strain and traction in terms of the inhomogenous quantity (either temperature or initial stress).

Once a particular integral is found it is added to the complementary function to form the total solution. The parameters of the complementary function are adjusted to insure that the total solution satisfies the boundary condition, and hence, produces a unique solution to the boundary value problem.

The boundary integral equation (2.1.9) is discretized and integrated for a system of boundary nodes in the manner described in Chapter 3. The resulting equations can be expressed in matrix form as

$$Gt^c - Fu^c = 0$$

(2.1.11)
By introducing equation (2.1.10) into equation (2.1.11), the complementary function can be eliminated;

\[ Gt - Fu = G \Gamma^p - F \Gamma^p \]  

(2.1.12)

The particular integral terms on the right hand side of this equation are functions of known body forces.

In a multi-region problem a set of (complementary) equations, similar to equation (2.1.11), are generated independently for each region. Likewise, the particular integrals of each region are derived independently leading to a set of equations for each region similar in form to equation (2.1.12).

The particular integral approach is also utilized in BEST3D to represent hotspots and discontinuities within the volume. These are taken care of by developing the necessary particular integrals due to an initial stress system \( \sigma^0_{ij} \) within a rectilinear prism of dimension \( a \times b \times c \). The solution can be expressed as [24]:

\[ u_i(\xi) = H_i(x, \xi)T(x) \]

\[ \sigma_{ij}(\xi) = H_{ij}^0(x, \xi)T(x) \]  

(2.1.13)

where \( T \) is the temperature of the hotspot over a volume \( a \times b \times c \) and

\[ u_i(\xi) = K_{ijk}(x, \xi)\sigma^0_{jk}(x) \]

\[ \sigma_{ij}(\xi) = K_{ijkl}(x, \xi)\sigma^0_{kl}(x) \]  

(2.1.14)

where \( \sigma^0_{jk}(x) \) are determined by satisfying the conditions of zero stress within the discontinuity. The functions \( H, H^0, K \) and \( K^0 \) are continuous functions of \( x \) and \( \xi \) and no additional integrations are involved [24].

Elastic analysis of three-dimensional bodies can also be carried out in the presence of embedded holes and inserts as described later in this chapter.
For structural problems which involve inelastic material response, volume integrals must be retained over at least a portion of the body. In these cases, equations (2.1.3) and (2.1.4) are rewritten, in incremental form, in terms of the initial stresses as [18-25]:

\[
C_{ij} \dot{u}_i(\xi) = \int_S \left[ G_{ij}(x,\xi) \dot{t}_i(x) - F_{ij}(x,\xi) \dot{u}_i(x) \right] ds + \int_V B_{ijk}(x,\xi) \dot{\sigma}_{ik}^0(x) dv (2.2.1)
\]

and

\[
\dot{\sigma}_{ij}(\xi) = \int_S \left[ G_{kij}^{0}(x,\xi) \dot{t}_k(x) - F_{kij}^{0}(x,\xi) \dot{u}_k(x) \right] ds + \int_{V-V_s} M_{ijkl}(x,\xi) \dot{\sigma}_{kl}^0(x) dv + J_{ijkl} \dot{\sigma}_j^0(\xi) (2.2.2)
\]

where \( \dot{\sigma}_{kl} \) is a jump term arising out of analytical treatment of the volume integral over a small spherical region around \( \xi \) [25].

The volume integrals in (2.2.1) and (2.2.2) both involve unknown initial stresses, whose evaluation involve the use of the nonlinear constitutive model. Thus equations (2.2.1), (2.2.2) and the nonlinear constitutive model can be considered as a single nonlinear system to be solved for the unknown boundary displacements and tractions and for the initial stresses within the inelastic region.

In BEST3D the material models provided are:

(a) Nonlinear strain hardening Von Mises model [2,7,16]
(b) Multi-surface cyclic plasticity model [36]
(c) Thermally sensitive viscoplastic model of Walker [49]

The pertinent details for these models are provided below:

(a) 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

\[
F(J_2) = \frac{1}{2} S_{ij} S_{ij} - \frac{1}{3} \sigma_0^2 = 0 (2.2.3)
\]

where

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

\( \sigma_0 \) = 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

\[ \dot{\varepsilon}_{ij} = \lambda \frac{\partial F}{\partial \sigma_{ij}} \]  

(2.2.4)

where \( \lambda \) 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 surfaces 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

\[ F(\sigma_{ij}, \dot{\varepsilon}_{ij}^p, h) = 0 \]  

(2.2.5)

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

\[ \dot{\sigma}_{ij} = D_{ijkl}^p \dot{\epsilon}_{kl} \]  

(2.2.6a)

where

\[ \dot{\epsilon}_{kl} = \dot{\varepsilon}_{kl}^p + \dot{\varepsilon}_{kl}^{plastic} \]  

(2.2.6b)

and \( D_{ijkl}^p \) is the plastic modulus tensor which can be explicitly written as

\[ D_{ijkl}^p = 2\mu \delta_{ik}\delta_{jl} + \frac{\nu}{1 - 2\nu} \delta_{ij}\delta_{kl} - \frac{3S_{ij}S_{kl}}{2\sigma_c^2(1 + h/3\mu)} \]  

(2.2.7)

(b) Two-surface Cyclic Plasticity Model

The standard isotropic and kinematic hardening models, which are based on distinctively defined elastic and elastoplastic behavior, are inadequate to represent the smooth transition from elastic to fully plastic state that is observed experimentally during cyclic loading of most metals and geologic materials. As a result many other models which are conceptually similar to the nested yield surface model originally proposed by Mroz [46] involving both kinematic and isotropic hardening have been developed. An extremely practical model based on the ideas of Mroz [46] was introduced by Kreig [47] and Dafalias and Popov [48] which they described as the two surface model. The model used here is essentially a modified version [36] of that described by Kreig and is briefly described below.
In this model it is assumed that there exists a loading or yield surface in the domain bounded by another limiting yield or bounding surface. During loading in the plastic state the stress state remains in contact with the inner yield surface, which translates in the stress space during deformation. Once this surface touches the bounding surface, the behaviour during deformation is governed by either isotropic or kinematic hardening rules. Thus, during deformation, the stress point moves through three distinct regions: (a) an elastic region, which is associated with recoverable strains, when the stress state is within the inner yield surface; (b) a metaelastic region, where the hardening is determined from the relative location of the stress state in relation to the bounding surface, when the stress point is on the inner yield surface, but within the bounding surface; and (c) a plastic region, where the stiffness is given by plastic hardening parameter associated with the bounding surface when the stress point is on the bounding surface. The stiffness of the material changes dramatically as the stress state moves through the regions.

In the elastic region, the behavior is governed by elastic constitutive relationships which are well known. The rules governing the behavior in the metaelastic region, which provide the smooth transition from elastic to plastic behavior, is determined as follows.

The yield functions associated with the loading and bounding surfaces can be expressed, respectively (both having the same shape and orientation) as:

\[ F(\sigma_{ij}, \varepsilon_{ij}^p) = 0 \]  \hspace{1cm} (2.2.8a)

and

\[ F(\sigma_{ij}^b, \varepsilon_{ij}^p) = 0, \]  \hspace{1cm} (2.2.8b)

where

\[ \sigma_{ij} = \sigma_{ij} - \alpha_{ij}, \]
\[ \sigma_{ij}^b = \text{stress state on the loading surface}, \]
\[ \sigma_{ij}^b = \text{stress state on the bounding surface}, \]
\[ \varepsilon_{ij}^p = \text{plastic strain}, \]
\[ \alpha_{ij} = \text{location of the center of loading (inner yield) surface}. \]

During deformation from the stress state \( A \), the translation of the loading surface is along \( AA^b \) (see Fig. 2.2.1), where \( A^b \) is a unique image of the current stress state on the bounding surface. The location of \( A^b \) is determined by the intersection of a line drawn parallel to \( OA \) through \( O' \) and the bounding surface. The translation of the loading surface is then given by [36]:

\[ \Delta \sigma_{ij} = \frac{h_0}{\sigma_o}[(\sigma_o^b - \sigma_o)\sigma_{ij} - (\alpha_{ij}\sigma_o^b - \alpha_{ij}\sigma_o)], \]  \hspace{1cm} (2.2.9)

where

\[ \sigma_o = \text{an equivalent yield stress associated with the loading surface}, \]
\( \sigma^b \) = an equivalent yield stress associated with the bounding surface,
\( \alpha_{ij} \) = location of the center of loading surface,
\( \alpha^b_{ij} \) = location of the center of bounding surface,
\( \mu_o \) = a scalar parameter yet to be determined.

Since the stress point remains on the loading surface during deformation, the consistency conditions must be satisfied. This gives the value of:

\[
\mu_o = \frac{\delta F/\delta \sigma_{ij}}{\delta \sigma_{kl} (\sigma^b_{kl} - \sigma_{kl})}
\]  (2.2.10)

Using the flow rule and consistency condition, the governing stress-strain relationship can be obtained as

\[
\dot{\sigma}_{ij} = \left[ D_{ijkl}^{\varepsilon} - \frac{D_{ijkl}^{\varepsilon} \left( \frac{\partial F}{\partial \sigma_{ij}} \right) \left( \frac{\partial F}{\partial \sigma_{kl}} \right) D_{ijkl}^{\varepsilon}}{H_e + H_p} \right] \dot{\varepsilon}_{kl},
\]  (2.2.11)

where

\[
H_e = \left( \frac{\partial F}{\partial \sigma^b_{ij}} \right) D_{ijkl}^{\varepsilon} \left( \frac{\partial F}{\partial \sigma^b_{kl}} \right),
\]  (2.2.12)

and \( H_p \) is the plastic hardening modulus.

\( H_p \) is determined from the position of the stress state relative to the loading and bounding surfaces and the one-dimensional stress-strain behavior of the material as

\[
H_p = h \left[ \frac{\beta}{\sigma_{ref}} \right]^n,
\]  (2.2.13)

where \( h \) is the hardening of the bounding surface. \( \beta_{ij} \) represents the distance between the current stress state to the last location of the loading surface center when the material was elastic and \( \beta = [\beta_{ij} \beta_{kl}]^{1/2} \). \( \sigma_{ref} \) is the distance from the last location of the loading surface center for elastic behavior to the point \( A^b \) on the bounding surface (Fig. 2.2.1) obtained by drawing a line parallel to \( \sigma_{ij} \) and locating the intersection with the bounding surface.

The exponential power \( n \) is given by the expression

\[
n = \frac{\log(\text{inner surface hardening} / \text{outer hardening})}{\log(\text{inner yield stress} / \text{outer yield stress})}
\]  (2.2.14)

During cyclic loading, the loading and unloading are determined from the sign (positive for loading and negative for unloading) of the loading function defined as

\[
dL = N_{ij} \dot{\sigma}_{ij}
\]  (2.2.15)

where \( N_{ij} \) is the normal at \( \sigma^b_{ij} \) and is given by
Equations (2.2.9)-(2.2.16) provide the necessary and complete mechanical description of the material for monotonic as well as cyclic loading and requires only three inelastic parameters \( h, n \) and \( \sigma_{ref} \).

![Diagram of two surface plasticity model](image)

**Fig. 2.2.1 Two surface plasticity model**

(c) The Walker Viscoplastic Model

The thermally sensitive Walker viscoplasticity model which is implemented in BEST3D is described in detail in Walker[49] and Cassenti and Thompson [50].

Reference [49] describes the basic theory; while Reference [50] describes modifications to the form of the basic theory, and modifications to the material parameters for Hastelloy X. The modifications provide more accuracy at relatively low temperatures.
The inelastic strain consists of two components: a time dependent power law creep component, containing the material constants $n$ and $k$, and a time independent plastic component. The back stress $\Omega$ is a key variable in many viscoplastic material models. Its evolution is given below:

\[
\dot{\varepsilon}_{ij} = \frac{(3/2)\varepsilon_{ij} - \Omega_{ij}}{\sqrt{2/3(2s_{ij} - \Omega_{ij})}} \left\{ \exp \left[ \frac{\sqrt{2/3(3/2s_{ij} - \Omega_{ij})(3/2s_{ij} - \Omega_{ij})}}{\beta} \right] - 1 \right\}
\]

\[
K = K_1 - K_2e^{-n_t R} - K_3e^{-n_p L}
\]

\[
\dot{\Omega}_{ij} = \Omega_{ij} + \Omega_{ij1} + \Omega_{ij2}
\]

\[
\dot{\Omega}_{ij1} = n_2\varepsilon_{ij} - \Omega_{ij1} \left( \left[ n_3 + n_4e^{-n_5\log R/R_4} \right] + \frac{1}{n_2} \frac{\partial n_2}{\partial \Theta} \dot{\Theta} - \frac{1}{n_3} \frac{\partial n_3}{\partial \Theta} \dot{\Theta} \right) + \left( \frac{n_2}{n_3} - \Omega_{ij1} \right) \frac{\partial n_3}{\partial \Theta} \dot{\varepsilon}_{ij} \dot{\Theta}
\]

\[
\dot{\Omega}_{ij2} = n_{11}\varepsilon_{ij} - \dot{\Omega}_{ij2} \left( n_9 \dot{R} + n_{10} + \frac{1}{n_{11}} \frac{\partial n_{11}}{\partial \Theta} \dot{\Theta} - \frac{1}{n_9} \frac{\partial n_9}{\partial \Theta} \dot{\Theta} \right) + \left( \frac{n_{11}}{n_9} - \Theta_{ij2} \right) \frac{\partial n_9}{\partial \Theta} \dot{\varepsilon}_{ij} \dot{\Theta}
\]

\[
\dot{\Omega}_{ij} = 3\dot{\Omega} \left[ \frac{C_{ik}C_{kj}}{C_{pq}C_{pq}} + \frac{C_{ik}C_{kj}}{C_{pq}C_{pq}} - \frac{C_{ik}C_{kj}}{C_{pq}C_{pq}} \left( \frac{C_{rs}C_{rs}}{C_{uv}C_{uv}} \right) \right] + \left[ 3\frac{C_{ik}C_{kj}}{C_{pq}C_{pq}} - \delta_{ij} \right] \frac{\partial n_3}{\partial \Theta} \dot{\Theta}
\]

\[
s_{ij} = \sigma_{ij} - \frac{1}{3} \delta_{ij} \sigma_{kk}
\]

Material Constants: $\lambda, \mu, \hat{\omega}, \beta, n_1, n_2, n_3, n_4, n_5, n_6, n_7, n_8, n_9, n_{10}, n_{11}, K_1, \ldots, \text{and } K_3$ depend on temperature, $\Theta$. 
Inelastic Solution Algorithms:

Two inelastic solution algorithms have been implemented in BEST3D to solve the nonlinear system. The first method is an iterative procedure which 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. This method is applicable to all of the material models listed above.

The second procedure is a direct, variable stiffness algorithm that does not require iteration. Instead, the procedure exploits certain features of the constitutive relationship to express the unknown nonlinear initial stress tensor as a scalar quantity which can then be eliminated from the boundary equation system through a back substitution of the (modified) stress rate equation. This method is restricted to the Von Mises model (with strain hardening) under monotonic loading conditions. Explicit details can be found in references [25,37,39,40,45]
A new method for free-vibration analysis by boundary element method using particular integrals has been implemented in the current version of BEST3D. This method utilizes a fictitious vector function to approximate the inertia forces and then uses the well known concept of complementary functions and particular integrals to solve the resulting governing differential equations [29,53].

The governing differential equation for free-vibration of an elastic, homogenous and isotropic solid can be written in operator notation as

\[ L(u_i) + \rho w^2 u_i = 0 \]  \hspace{1cm} (2.3.1)

The solution of the above equation can be represented as the sum of a complementary function, \( u_i^c \), satisfying

\[ L(u_i^c) = 0 \]  \hspace{1cm} (2.3.2)

and a particular integral, \( u_i^p \), satisfying

\[ L(u_i^p) + \rho w^2 u_i = 0 \]  \hspace{1cm} (2.3.3)

In equation (2.3.3), the unknown displacement \( u_i \) within the domain is approximated by an infinite summation of the products of an unknown fictitious density function, \( \phi_k \), and a known function \( D_{ik} \). More specifically

\[ u_i(x) \simeq \sum_{m=1}^{\infty} D_{ik}(x, \xi^m)\phi_k(\xi^m) \]  \hspace{1cm} (2.3.4)

The direct boundary integral equation related to the displacement function \( u^c \) can be written as

\[ G_{ij}u^c(\xi) = \int_s [G_{ij}(x, \xi)\phi_j(\xi) - F_{ij}(x, \xi)u^c(x)]ds \]  \hspace{1cm} (2.3.5)

where \( G_{ij} \) and \( F_{ij} \) are the elasto-static kernels (i.e. fundamental solutions of eq. 2.3.2).

By usual discretization of the boundary \( S \), equation (2.3.6) can be expressed in a matrix form as

\[ [G]\{t^c\} - [F]\{u^c\} = \{0\} \]  \hspace{1cm} (2.3.6)
The complementary solutions are related to the real displacement \( u_i \) and tractions \( t_i \) via

\[
\begin{align*}
    u_i^c &= u_i - u_i^p \\
    t_i^c &= t_i - t_i^p
\end{align*}
\]  

Substituting equations (2.3.4), (2.3.7) and (2.3.8) into equation (2.3.6), the following equations [29] are obtained

\[
[G]{t} - [F]{u} = \rho w^2 [M]{u}
\]  

By incorporating the known boundary conditions (i.e. at a node either \( t_i = 0 \) or \( u_i = 0 \)) in eq. 2.3.9, an algebraic expression results for eigenvalue extraction, i.e.

\[
[A]{x} = \rho w^2 [M]{x}
\]  

Finally, equation (2.3.10) is solved by using an eigenvalue extraction routine developed by Garbow of Argonne National Laboratory to obtain the eigen frequencies and mode shapes.
By using the dynamic reciprocal theorem, the boundary integral representation for periodic dynamics can be written in the frequency domain as Banerjee and Butterfield [2]:

\[ C_{ij}u_i(\xi, \omega) = \int_S [G_{ij}(x, \xi, \omega) t_i(x, \omega) - F_{ij}(x, \xi, \omega) u_i(x, \omega)] \, dS(x) \]  

(2.4.1)

where \( S \) is the surface enveloping the body and the fundamental solution \( G_{ij} \) and \( F_{ij} \) are defined in Banerjee et al [41] and Ahmad and Banerjee [51]. It should be noted here that although the functions \( G_{ij} \) and \( F_{ij} \) becomes identical to their static counterpart as \( \omega \) tends to zero, it is important to evaluate this limit carefully because of the presence of \( \omega \) in the denominator.

Once the boundary solution is obtained, the interior version of equation (2.4.1) can also be used to find the interior displacements; and the interior stresses can then be obtained using the strain displacement and stress strain relations as:

\[ \sigma_{jk}(\xi, \omega) = \int_S [G_{ijk}(x, \xi, \omega) t_i(x, \omega) - F_{ijk}(x, \xi, \omega) u_i(x, \omega)] \, dS(x) \]  

(2.4.2)

The stresses at the surface can be calculated by combining the constitutive equations, the directional derivatives of the displacement vector and the values of field variables in an accurate matrix formulation.

The boundary integral formulations (2.4.1) and (2.4.2) can also take account of internal viscous dissipation of energy (damping) using complex elastic moduli in the usual manner.
The direct boundary integral formulation for a general, transient, elastodynamic problem can be constructed by combining the fundamental point-force solution (Stokes solution) of the governing equation of motion (Navier-Cauchy equations) with the dynamic reciprocal theorem. For zero initial conditions and zero body forces, the boundary integral formulation for transient elastodynamics reduces to \([2,26,27,28]\)

\[
C_{ij}(x) = \int_S \left[ G_{ij}(x) * u_i(z) - F_{ij}(x) * u_i(z) \right] dS(z)
\] (2.5.1)

where

\[
G_{ij} * u_i = \int_0^T G_{ij}(x,T;\xi,\tau) t_i(x,\tau) d\tau \quad (2.5.2a)
\]

\[
F_{ij} * u_i = \int_0^T F_{ij}(x,T;\xi,\tau) u_i(x,\tau) d\tau \quad (2.5.2b)
\]

are Riemann convolution integrals, while \(\xi\) and \(z\) are the space positions of the receiver (field point) and the source (source point). The fundamental solutions \(G_{ij}\) and \(F_{ij}\) are the displacements \(u_i\) and tractions \(t_i\) at a point \(z\) at time \(T\) due to a unit force vector applied at a point \(\xi\) at a preceding time \(\tau\). The tensor \(C_{ij}\) is the discontinuity (or jump) tensor, arising from the singularity of the \(F_{ij}\) kernel.

Equation (2.5.1) represents an exact formulation involving integration over the surface as well as the time history. It should also be noted that this is an implicit time-domain formulation because the response at time \(T\) is calculated by taking into account the history of surface tractions and displacements up to and including the time \(T\). Furthermore, equation (2.5.1) is valid for both regular and unbounded domains.

This formulation has two major advantages:

1. It is expressed entirely in terms of real variables, allowing much improved computational efficiency.
2. Causality is preserved: that is, the structure of the mathematical formulation is such that events at a given time can be affected only by prior history.

Once the boundary solution is obtained, the stresses at the boundary can be calculated by combining the constitutive equations, the directional derivatives of the displacement vector and the values of field variables in an accurate matrix formulation. For calculating displacements at interior points equation (2.5.1) can be used with \(C_{ij} = \delta_{ij}\), and the interior stresses can be obtained from

\[
\sigma_{jk}(x) = \int_S \left[ G_{jk}^D(x) * t_i(z) - F_{jk}^D(x) * u_i(z) \right] dS(z)
\] (2.5.3)
In order to obtain the transient response at a time $T_N$, the time axis is discretized into $N$ equal time intervals, i.e.

$$T_N = \sum_{n=1}^{N} n\Delta T$$  \hspace{1cm} (2.5.4)

where $\Delta T$ is the time step.

Utilizing equations (2.5.4), and (2.5.2), equation (2.5.1) can be written as

$$C_{ij}u_i(\xi, T_N) = \int_T^{T_N} \int_S [G_{ij}t_i - F_{ij}u_i] \, dSdr = \int_T^{T_N} \int_S [G_{ij}t_i - F_{ij}u_i] \, dSdr$$  \hspace{1cm} (2.5.5)

where the integral on the right hand side is the contribution due to the past dynamic history.

It is of interest that equation (2.5.5), like equation (2.5.1) still remains an exact formulation of the problem since no approximation has yet been introduced. However in order to solve equation (2.5.5) one has to approximate the time variation of the field quantities in addition to the usual approximation of spatial variation. For this purpose a linear interpolation function is used. The details can be found in the paper by Ahmad and Banerjee [28].
The governing differential equation for transient heat conduction can be written as
\[ \frac{k}{\partial x_i \partial x_i} - \rho c \frac{\partial T}{\partial t} + \psi = 0, \]  
with temperature \( T \), conductivity \( k \), density \( \rho \), specific heat \( c \), and body sources \( \psi \). The corresponding reciprocal theorem
\[ \int_S q(x) \ast T^*(x) ds + \int_V \psi(x) \ast T^*(x) dv = \int_S q^*(x) \ast T(x) ds + \int_V \psi^*(x) \ast T(x) dv \]  
relates any two states of temperature \( T \), heat flux \( q \), and \( \psi \) body sources occupying in the volume \( V \) with surface \( S \). Note that since convolutions appear in (2.6.2), the entire time history is involved. Using the infinite space fundamental solution of (2.6.1) as one of the states, and the solution of the desired boundary value problem with zero body sources as the other, produces the following boundary integral equation:
\[ C(\xi)T(\xi, t) = \int_S [G(x, \xi, t) \ast q(x, t) - F(x, \xi, t) \ast T(x, t)] ds(x). \]  
In (2.6.3), the kernel functions \( G \) and \( F \) are derived from the fundamental solutions, while \( C(\xi) \) is the discontinuity term arising from the singularity of the \( F \) kernel.

Under steady-state conditions the governing equation simplifies to
\[ \frac{k}{\partial x_i \partial x_i} + \psi = 0, \]  
and the boundary integral equation becomes
\[ C(\xi)T(\xi) = \int_S [G(x, \xi)q(x) - F(x, \xi)T(x)] ds(x). \]  
\( G \) and \( F \) are now the well-known potential flow kernel functions. Equation (2.6.5) is, of course, time independent. For more information, please refer to Banerjee and Butterfield [2] and Dargush and Banerjee [42].

**BEST3D** contains multiregion capability for both steady-state and transient heat conduction as boundary-only problems. The spatial and temporal discretization of (2.6.5) and (2.6.3) is discussed in some detail in Chapter 3.

The application of the boundary element method for heat conduction analysis effectively reduces the dimensionality of the problem by one. However, in certain instances, further reduction is possible. For example, in axisymmetric problems, the required circumferential integration can be performed in a semi-analytical manner at the kernel level.
result, numerical integration is only needed along the boundary in the r-z plane. Similarly, in other situations, meaningful approximations can be introduced to simplify an analysis. Such is the case for circular, small diameter holes embedded in a three-dimensional heat conducting body.

Consider a cylindrical exclusion in an infinite region. The boundary integral equation (2.6.3) is valid at any point \( \xi \), where \( S \) represents the surface of the exclusion. Remote from the hole, it is reasonable to assume constant temperature and flux in the circumferential direction along the surface of the hole. With that assumption, the integration required by (2.6.3) in the circumferential direction can be performed in a semi-analytical manner and incorporated in the kernel functions. Ignoring the contribution of the circular disks at the ends of the hole, all that remains is a line integral along the centerline \( C \). Thus, 

\[
C(\xi)T(\xi, t) = \int_S [G^c(x, \xi, t) * q(z, t) - F^c(x, \xi, t) * T(z, t)] \, dz(x),
\]  

(2.6.6)

where the kernels \( G^c \) and \( F^c \) include the circumferential integration. The remaining line integration can be performed numerically. In general, there can be multiple exclusions within a finite three-dimensional body. The boundary integral equation for a region with outer boundary \( S \) and \( N \) holes generalizes to

\[
C(\xi)T(\xi, t) = \int_S [G(x, \xi, t) * q(z, t) - F(x, \xi, t) * T(z, t)] \, ds(x)
\]

\[
+ \sum_{n=1}^{N} \left[ G^c(x, \xi, t) * q(z, t) - F^c(x, \xi, t) * T(z, t) \right] \, dc_n(x).
\]

(2.6.7)

The embedded hole capability is available for both steady-state and transient problems.

It should be noted that many other physical processes are governed by equations of the form of (2.6.1) or (2.6.4). For example, the steady flow of inviscid incompressible fluids reduces to (2.6.4), as does problems of electrostatics, while the diffusion of chemical substance is represented by (2.6.1). In all of these cases, with an appropriate translation of the field variables, BEST3D is directly applicable.
The regular boundary integral equation for displacement of a three-dimensional body with embedded holes may be written as

\[ c_{ij} u_i(\xi) = \int_{S} G_{ij}(x,\xi) t_j(x) - F_{ij}(x,\xi) u_j(x) \, dS(x) \]
\[ + \sum_{m=1}^{M} \int_{s_m} G_{ij}(x,\xi) t_j(x) ds^m(x) - \sum_{m=1}^{M} F_{ij}(x,\xi) u_j(x) ds^m(x) \]  

(2.7.1)

where \( G_{ij} \) and \( F_{ij} \) is the Kelvin point force solution and corresponding traction kernel.

\( u_i \) and \( t_i \) are boundary displacement and tractions.

\( S \) is the outer surface of the body.

\( s_m \) is the surface of the \( m \)th hole.

\( C_{ij} \) are the constants determined by the relative smoothness of the body.

If body forces or nonlinear effects are present they can be incorporated in equation (2.7.1) through a volume integral or using a particular integral.

Discretization of equation (2.7.1) in the usual boundary element manner requires a very fine discretization of the holes. Alternatively, BEST3D offers a new concept called 'Hole Elements' for the treatment of the embedded holes. This formulation is derived from equation (2.7.1) employing a circular shape function and appropriate transformations and analytical integration about the circumference of the hole. In this formulation the following assumptions are made. First, the hole is of a tubular shape and is assumed to be of circular cross section and closed at the ends with a flat circular surface. The displacement about the circumference of the hole is assumed to be of a sine-cosine variation and the pressure is assumed to be constant. The variation of the geometry and field variables in the longitudinal direction is linear or quadratic.

A hole which has curvature along its length will differ in surface area about the circumference on the curved portion of the tube. This is neglected in the integration since the 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 hole which is most commonly encountered. Finally, we note that the holes should not intersect the outer surface of the body or intersect other holes. This minor restriction can be lifted if results at these locations are not of interest. As long as the holes do not coincide with nodal points of other elements, the errors will be localized and will not effect the overall boundary element solution.

The essential part of the formulation is the conversion of the two-dimensional surface integration of the hole to a one-dimensional integration. In equation (2.7.1) the last two
integrals under the summation are the integrals associated with the hole which are to be modified. In the present formulation if the holes in the three-dimensional body are assumed traction free then only the \( F_{ij} \) (traction) integral needs to be derived. If pressure is present in the holes then the \( G_{ij} \) (displacement) kernel is also necessary.

The kernel functions for the holes are derived by analytically integrating the three-dimensional kernels in the circumferential direction. To facilitate the integration the kernel functions are first expressed in local coordinates with the center of the coordinate system coinciding with the center of the hole and the z axis aligned with the centerline of the tube. The relative translation \( \xi_i \) is added to the field coordinate \( \xi_i \) and the rotation is applied using the appropriate vector transformation.

\[
\xi_i = a_{ij} \xi_j + \xi'_i
\]

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 displacement must also be transformed between the local and the global systems by

\[
u_i = a_{ik} \nu_k
\]

or

\[
u_j = a_{mj} \nu_m
\]

the integration point \( z_i \) for a ring can now be expressed in cylindrical coordinates relative to the center of the hole as

\[
X_1 = R \cos \theta \\
X_2 = R \sin \theta \\
X_3 = z
\]

where \( R \) represents the radius of the hole, 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 hole. For a straight hole \( n_r = 1 \) and \( n_z = 0 \), and on the flat surface closing the end of the hole \( n_r = 0 \) and \( n_z = 1 \). Next a circular shape function is employed to approximate the variation in the displacement about the circumference of the hole. The circular shape function which is multiplied and integrated
with the 3-D \( F_{ij} \) kernel, allowing the nodal values to be brought outside the integral. The shape function is expressed as

\[
   u_i = M^\gamma U_1^\gamma \quad (\text{summation is implied}, \gamma = 1, 2, 3)
\]

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 \( U_1^\gamma \) are the nodal displacements.

If internal pressure is present in the hole, the pressure is assumed to be constant about the circumference. The traction vector in equation (2.7.1) however, is written as a global vector and must be converted to a local scalar variable. This transformation vector is multiplied and integrated along with the \( G_{ij} \) kernel allowing the constant pressure value to be brought outside the integration. The load transformation is

\[
   t_i = N_i P
\]

where \( N_i \) is the transformation vector of the normal vectors.

\[
   N_1 = n_r \cos \theta
\]

\[
   N_2 = n_r \sin \theta
\]

\[
   N_3 = n_z
\]

and \( P \) is the internal pressure in the hole. If the pressure in the hole is zero, the second last term under the summation in equation (2.7.1) vanishes.

The last two terms in equation (2.7.1) can now be analytically integrated in the circumferential direction. For the \( m \)th hole this can be expressed as

\[
   \int_{C_m} G_{ij}(x,\xi)t_j(z)ds^n(z) = \int_{C_m} a_{ik} \int_{0}^{2\pi} G_{kj}(R,\theta,z,\xi)N_j R d\theta dC^m
\]

\[
   = \int_{C_m} G^{H_i}(R,z,\xi)pdC^m(z)
\]

\[
   \int_{C_m} F_{ij}(x,\xi)u_j(z)ds^n(z) = \int_{C_m} a_{ik} \int_{0}^{2\pi} F_{kj}(R,\theta,z,\xi)M^\gamma R d\theta a_{m_j} U^n dC^m
\]

\[
   = \int_{C_m} F_i^{H_j}(R,z,\xi)U_j^\gamma dC^m(z)
\]
where \( C^m \) represents the curvilinear centerline of the hole and \( G_{ij} \) and \( F_{ij} \) represent the analytically integrated hole kernels. Note, since the transformation vector \( a_\theta \) is independent of \( \theta \), it may be taken outside the \( d\theta \) integration.

The analytic circumferential integration of the kernels yields functions containing elliptical integrals and related functions. The elliptical integrals and related functions are expressed numerically by common series approximations.

Additional series approximations had to be derived for a range of input values (coordinate locations) that were found to cause numerical instabilities in these common approximations. The new series were derived using a best fit polynomial approximation (as a function of elliptic moduli) using values of the integrals calculated by very accurate numerical integration in the circumferential direction.

The derivation of the hole kernels corresponding to stress or strain equation can be calculated in two ways. In the first method the strain equation is derived from the displacement equation (2.7.1) by differentiation and application of the strain-displacement equation. The stress equation is then found using Hooke's Law. However, due to the complexity of the kernel functions of the displacement equation, an alternative method is employed in which appropriate transformations and integration in the circumferential is carried out in the manner identical to the derivation of the displacement kernel functions for hole element.

In constructing a boundary element system, the equation (2.7.1) is usually written for all nodal points used in the representation of the boundary and hole field variables. Hence, for every node used along the length of a hole element, three (three degree of freedom) equations must be written since the circular shape function's variations three nodes. Therefore, for a hole, a single quadratic (3-nodal) shape function in the longitudinal direction would result in a total of nine (3 degree of freedom) equations.

The equations for the nodes on the outer boundary are written at the coordinates of the nodes in the usual BEM manner. The jump terms and the Cauchy-principle-value of the singular integration associated with the singular elements are evaluated using the rigid body translation technique. In the present formulation, the summation of the coefficients corresponding to the holes should be zero for the equations written for the nodes on the outer boundary. It is recommended to assume the summation to be exactly zero and not to include any residue from the summation of the coefficients of the holes in the rigid body calculation for the points on the outer boundary.

The equations written for the nodes defining the hole may be handled in a similar manner. However, small errors may be introduced into the calculation of the hole coefficients of the diagonal terms either due to errors in the singular integration of the complex hole kernels or due to the assumptions made approximating the integration of a curve tubular hole. In a rigid body coefficient calculation of the diagonal term of the \( F \) (traction) matrix, the summation of these errors may have some influence on the results. Therefore, it was decided to write the equations corresponding to the nodes of the holes slightly inside the hole (offset from the surface of the hole one-fourth the distance of the radius). Likewise,
equations for the nodes at the end of the hole are moved slightly off the surface of the end to inside the hole. Note, the nodal values of the displacement and pressures associated with the hole elements still lie on the surface of the hole. By moving the equation point inside the hole, the singular integration is circumvented and all the values of $C_q(\xi)$ (equation (2.7.1)) are zero for these equations written inside the void of the hole. However, care is still required for the delicate integration of a hole element for a node contained in that element.

After all source nodes and the boundary element system have been integrated over all surface elements and hole elements (of a subregion) the system for all subregions may be assembled incorporating all special interface conditions (perfect body, springs, sliding, etc.) in the usual manner. All nodal displacement at the holes will be contained in the systems unknowns and any given boundary conditions of internal pressure within the hole will be multiplied by their respected coefficients and added to the right-hand side. Specified and unknown boundary conditions on the outer boundary are collected and assembled with their coefficients in the usual manner and the system may be solved yielding results for both the unknown conditions on the outer boundary and for the displacement on the surface of the holes.

Results for displacement, stresses and strains at points in the domain of the body may be calculated using the discretized integral equations. Results for points on the outer boundary or on the surface of the hole are best calculated using shape function and the boundary stress/strain calculation normally used in boundary element method.
### 2.8 STRESS ANALYSIS OF BODIES WITH COMPOSITE INSERTS

The conventional boundary integral equation for displacement is the starting point for the composite formulation. The displacement for a point $\xi$ inside the elastic composite matrix is given below

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

$$+ \sum_{n=1}^{N} \int_{S^n} [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]dS^n(x) \quad i, j = 1, 2, 3$$  \hspace{1cm} (2.8.1)

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 relative smoothness at $\xi$

$u_i, t_i$ are displacements and tractions

$S, S^n$ are surfaces of the matrix and holes (left for fiber) respectively

$N$ is the number of fibers

The conventional boundary integral equation for displacement can also be written for each of the $N$ insert fibers. For the displacement at a point $\xi$ inside the $m^{th}$ insert we can write

$$C_{ij}^m(\xi)u_i(\xi) = \int_{S^m} [G_{ij}^m(x,\xi)t_i(x) - F_{ij}^m(x,\xi)u_i(x)]dS^m(x) \quad i, j = 1, 2, 3$$  \hspace{1cm} (2.8.2)

$G_{ij}^m, F_{ij}^m$ are the fundamental solutions of the $m^{th}$ insert

$C_{ij}^m$ are constants determined by the relative smoothness at $\xi$ in insert $m$

$u_i, t_i$ are displacement and tractions associated with the $m^{th}$ insert

$S^m$ the surface of the $m^{th}$ insert

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

$$u_j(x) = u_j(x)$$ \hspace{1cm} (2.8.3a)
\[ i_j(x) = -t_j(x) \]  
\[ (2.8.3b) \]

For a stiff insert in which the elastic modulus 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. 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 \( m \)th insert

\[ F_{ij}^m(x, \xi) = -F_{ij}(x, \xi) \]  
\[ (2.8.3c) \]

Substitution of equations (2.8.3) into equation (2.8.2) yields the following modified boundary integral equation for insert \( m \).

\[ C_{ij}^m(\xi)u_i(\xi) = \int_{S_m} [-G_{ij}^m(x, \xi)t_i(\xi) + F_{ij}(x, \xi)u_i(x)]dS^m(x) \]  
\[ (2.8.4) \]

Finally adding \( N \) insert equation (2.8.4) to equation (2.8.1) and cancelling terms, yields the modified boundary integral equation for the composite matrix

\[ \mathcal{C}_{ij}(\xi)u_i(\xi) = \int_{S} [G_{ij}(x, \xi)t_i(x) - F_{ij}(x, \xi)u_i(x)]dS(x) \]
\[ + \sum_{n=1}^{N} \int_{S_n} [G_{ij}(x, \xi)t_i(x)dS^n(x) \]  
\[ (2.8.5) \]

where

\[ \mathcal{C}_{ij}(x, \xi) = G_{ij}(x, \xi) - G_{ij}^m(x, \xi) \]

\( \mathcal{C}_{ij} \) constants dependent on the geometry at \( \xi \)

Next, the two-dimensional kernel integral over the surface of the hole and insert is converted into a line integral. By performing on analytical integration in the circumferential direction on the surface of the hole (or insert) a considerable amount of computational time can be saved in the numerical integration. In this process the holes and inserts are assumed to be circular and a circumferential variation of \( a_0 + a_1 \cos \theta + a_2 \sin \theta \) is assumed in the displacements and tractions on the surface of the hole and inserts. Furthermore, tensor transformations on the kernels are necessary for inserts oriented a oblique angles with respect to the global axes. The resulting kernels, contain a large family of elliptical integrals

Once again the analytical integration is complete, the inserts are discretized, assembled and solved as described in Section 3.3.
The integral equation formulations discussed in the preceding section becomes of practical interest only when numerical techniques are employed for their solution, since analytical solutions exist only for the simplest geometries and loadings. In basic terms, the outline of the numerical solution process is:

1 - Approximate the part geometry and the variation of field quantities (e.g., displacement, traction, temperature) using a suitable set of computationally convenient simple functions.
2 - Evaluate the surface and volume integrals numerically.
3 - Assemble a set of algebraic equations by utilizing the boundary conditions specified for the problem.
4 - Solve the equation system.
5 - Solve for interior results if desired.

In the case of a transient problem (or a quasi-static analysis with multiple load cases) steps 3 and 4 are repeated. In a nonlinear analysis, iteration at each load or step will normally be required in order to satisfy the constitutive equation. The basic steps in the numerical solution process are very briefly discussed in the sections which follow. Full details are available in the sources previously cited [2-13].
3.1 REPRESENTATION OF GEOMETRY AND FUNCTIONS

The method employed in BEST3D for the approximation of both part geometry and field quantities is based on the use of the isoparametric shape functions originally developed for use in finite element analysis. The entire boundary of the part is modelled as the union of isoparametric patches. Over each patch the variation of each of the cartesian coordinates is approximated as

$$z_i = N^k(\eta)z^k_i$$

where the $z^k_i$ are the nodal values of the coordinate and $N^k$ are interpolation functions which take the value 1.0 at a single node and vanish at all others. Continuity along element boundaries is ensured.

In BEST3D the variation of (known and unknown) field variables is carried out over each surface patch using the isoparametric shape functions. It is frequently the case that full quadratic variation of the field quantities is not required, so the option of using linear, quadratic or a mixture of linear and quadratic interpolation functions is provided. Geometry is always modelled using the quadratic interpolation functions. Four special modelling capabilities should be noted:

1 - Substructuring capability is provided. This allows a part to be modelled as an assembly of several generic modelling regions (GMR). The GMRs, each of which must be a complete structure, are joined by enforcing appropriate compatibility conditions across common surface patches.

2 - Linear and quadratic variation of field quantities may be mixed within a single GMR.

3 - Infinite elements are provided for use in problems in which boundaries extend to infinity. A decay function is introduced to ensure appropriate behavior of the solution at infinity.

4 - The surface patches can be specialized for fracture mechanics analysis by an appropriate relocation of certain midside geometry nodes.

Volume modelling in BEST3D is also based on the use of the isoparametric shape functions. Both volume cell geometry and the variation of field quantities within the cell are mapped using the quadratic isoparametric shape functions. Volume modelling is required in the boundary element method only when thermal loads and/or nonlinear response are involved, and only the portion of the part in which these effects occur must be modelled. Nodes of the volume cells lying on the surface need not match nodes of the surface mesh in either location or node numbering.

An additional volume modelling capability using rectilinear cells in three dimensions is also provided in BEST3D. This newly developed capability is presently used for the modelling of embedded holes and for the definitions of hot spots within a structure.
An alternative analysis using the method of particular integrals, with global shape functions, has been developed also for inelastic analysis. This newly developed capability allows the user complete freedom from volume modelling. One merely needs to add some representative interior points where plasticity is likely to occur and isolate that region into a complete GMR. Investigation of its use in nonlinear analysis is continuing.
With the exceptions of volume integrals over rectilinear cells and of jump terms in principal value integrals, all surface and volume integrations in BEST3D are done numerically, using product Gaussian quadrature rules. All of the integrals involve the integration, of products of the patch interpolation functions and the kernel functions, over a particular surface patch (or volume cell). In the process of creating the equation system the source point is successively located at each node of the body at which unknowns must be determined. During the calculation of any given integral, the source point may be part of the surface patch (or cell) being integrated (singular case) or not (nonsingular case). Error estimates have been developed to allow determination of appropriate integration orders and element subdivisions for both the singular and nonsingular cases [2,25,30-34]. Similar discussions also apply to the volume integration.

The surface and volume integrations in BEST3D require almost no user intervention. Acceptable default values for the few required tolerances have been set internally, based on experience with a variety of linear and nonlinear problems. It should be noted that surface integration in the time embedded dynamic formulation has somewhat different requirements than in the quasi-static case. Reduced integration tolerances may be required for dynamic problems with high forcing frequencies and for transient problems. This essentially allows the program to subdivide the element to appropriately account for oscillatory behavior of the integrand.
The intermediate result of the surface and volume integrations is a set of coefficients which function as multipliers of field quantities (e.g., displacements, tractions, strains, stresses, temperatures, pressures). Some of the field quantities are known from the boundary conditions. During the assembly of the equation system the known and unknown variables are separated and expressed in appropriate local coordinate systems. The coefficients multiplying each set of variables are collected in matrix form for later use. Boundary conditions are imposed, including any required modifications to the coefficient matrices. As a result of these operations, the equations for the entire system, can be written in matrix form as

\[ Ax = By + R + C\sigma^o \]  

and

\[ \sigma = A^o x + B^o y + R^o + C^o \sigma^o \]  

where:

- \( x \) = vector of unknown displacements and tractions
- \( y \) = vector of known displacements and tractions
- \( \sigma^o \) = vector of (unknown) initial stress, where applicable
- \( R, R^o \) = vectors of body force or source contribution.

Note that equation (3.3.2) is necessary for problems in which interior stresses are required.

In linear periodic analysis \( R \) and \( R^o \) are assumed to be zero and in transient time-domain analysis these represent the past dynamic history of boundary excitation.

In any substructured (multi-GMR) problems, the matrix \( A \) in (3.3.1) contains large blocks of zeros, since separate GMRs communicate only through common surface elements. In order to save both storage space and computer time the matrix is stored in a block basis, with the zero blocks being ignored. The matrices in (3.3.2) are block diagonal, since interior results in any GMR involve only surface and volume integrations relative to that GMR.

In an inelastic analysis the constitutive equations of the form

\[ \sigma_{ij}(\xi) = D_{ijkl}^\varepsilon \xi_{kl} \]  

with

\[ \dot{\sigma}_{ij}(\xi) = (D_{ijkl}^\varepsilon - D_{ijkl}^\varepsilon_0) \xi_{kl} \]

defining the initial stress rates for an elastoplastic material and the form

\[ \dot{\sigma}_{ij}(\xi) = D_{ijkl}^\varepsilon_0 \sigma_{kl} \]  

defining the initial stress rate for a viscoplastic material are necessary to establish the magnitude of the vector \( \sigma^o \).
Two inelastic solution algorithms are contained in BEST3D for the solution of vector \( \mathbf{z} \) (and vector \( \sigma^0 \)). These algorithms, which are both incremental procedures, are described below.

**Iterative procedure** [25] In this procedure equations (3.3.1) and (3.3.2) are used together with equation (33) to solve the system in an incremental fashion. The \( \sigma^0 \) 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), the Two-surface cyclic plasticity model and the thermally sensitive viscoplastic model.

**Variable stiffness (direct) procedure** [37] This procedure exploits the plastic flow rule to express the unknown initial stress rate tensor as a non-negative scalar quantity \( \lambda \).

\[
\dot{\sigma}_{ij}^0 = K_{ij} \lambda
\]  

(3.3.4)

and from the other constitutive relations, the current stress rate is related to \( \lambda \) by

\[
\dot{\lambda} = L_{ij} \dot{\sigma}_{ij}
\]  

(3.3.5)

where \( K_{ij} \) and \( L_{ij} \) are tensor quantities dependent on the current state of the stress.

Equation (3.3.4) is used on a nodal basis to eliminate the \( \sigma^0 \) vector from the equations (3.3.1) and (3.3.2). Employing equation (3.3.5) (on a nodal basis), equation (3.3.2) can be back substituted into equation (3.3.1) yielding the final system equation for a given increment

\[
A^* \mathbf{z} = B^* \mathbf{y} + R^*
\]

the \( \mathbf{z} \) vector is the only unknown and standard numerical techniques are employed for its solution.

This method, in the present version of BEST3D, is restricted to the Von Mises model (with strain hardening) under monotonic loading.
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 used is to retain in the system only traction variables on the matrix/insert interface. This is in contrast to a general multiregion problem where both displacement and tractions are retained on an interface. The elimination of the displacements 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.

The boundary integral equation for the composite matrix derived in Section 2.11 is used to generate a system of equations for nodes on the outer surface of the matrix and for nodes on the surface of the holes containing the inserts. Written in matrix notation, we have

On Matrix Surface:

\[ G_t - F_u + Gt^H = 0 \] (3.3.6a)

On Hole for Insert:

\[ G_t - F_u + Gt^H = Iu^H \] (3.3.6b)

where

- \( t \) and \( u \) are traction and displacement vectors on the outer surface of the components
- \( t^H \) and \( u^H \) are traction and displacement vectors on the hole
- \( I \) is the identity matrix

Our goal is to eliminate \( u^H \) from the system. To this end, the integral equation for the inserts derived in Section 2.11 is written for each node on an insert, collocating slightly outside the insert [where \( C_H = 0 \)], we obtain

\[ F'u^H = G't^H \] (3.7)

Pre-multiplying equation (3.3.6b) by the \( F' \) matrix in equation (3.7) yields

\[ F'G_t - F'F_u + F'Gt^H = F'u^h \] (3.8)

Equation (3.7) can now be set equal to equation (3.8) and the final form of the system is derived.
On Matrix:

\[ G_t - F_u + G_t^H = 0 \]

On Hole:

\[ F'G_t - F'F_u + (F'G - G')t^H = 0 \]  \hspace{1cm} (3.9)

At every point on the outer surface, either the traction or the displacement is specified and on the surface of the hole only the tractions are retained. Therefore, the number of equations in the system are equal to the final number of unknowns, and hence, the system may be solved. Thereafter, equation (3.36) is used to determine the displacement on the matrix/insert interface.

Since the displacement 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 note that 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 section.

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

\[ A^b x = B^b y \]  \hspace{1cm} (3.10)

where

\[ x \text{ is the vector of unknown variables at boundary and interface nodes,} \]

\[ y \text{ is the vector of known variables, and} \]

\[ A^b, B^b \text{ are the coefficient matrices} \]

Standard numerical procedures can be used to solve the unknowns in equation (3.10). Details are described in the section.

Once all the displacements and tractions are known on the matrix outersurface and on the matrix/insert interface, interior quantities of displacement, stress and strain can be determined at any point in the composite matrix or in the insert. For displacement either the conventional boundary displacement integral equation can be employed or alternatively the modified equations can be used.

Equations for strains are derived from the displacement equations and the strain-displacement relations. Thereafter, equations for stress are obtained by substituting the resulting strain equations into Hooke's law. For points on the boundary, the stress/strain boundary calculation normally employed in boundary elements is used.
The first step in the process is the decomposition of the system matrix $A$. In order to minimize the time requirements this is done using the block form of the matrix. The decomposed form is stored for later use. At each load increment or time step all of the known field variables are calculated and multiplied by appropriate coefficient matrices to form the load vector. The decomposed form of the system matrix is then used to calculate the unknown field variables. In a homogeneous, quasistatic linear analysis, this completes the solution, and the process is repeated for the next time step.

The solution process for the time embedded transient algorithm is very similar to the above case, except that the evaluation of the load vector requires a new surface integration at each time step. However, the system matrix remains unchanged from one time step to the next.

In an inelastic analysis two solution algorithms have been implemented: An iterative algorithm [25]; and a (direct) variable [37] stiffness type approach. In the first procedure an iterative process is required in order to satisfy the constitutive relations. This iteration involves use of both the interior stress identity and the constitutive equations. In order to improve convergence of this process, initial stresses for each load increment are extrapolated from past history except at points of load reversal. This reduces the computational costs substantially for most elastoplastic and viscoplastic problems based on the iterative method.

The second approach, known as the variable stiffness method, exploits certain features of the constitutive relationship to express the unknown nonlinear initial stress tensor as a scalar quantity which can be eliminated from the boundary equation system through a back substitution of the (modified) stress rate equation. Consequently, a new system matrix is generated and must be solved at each load increment.

A new solver was written for BEST3D. It operates at the submatrix level, using software from the LINPACK package 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 time dependent and/or nonlinear behavior, is highly efficient.
3.5 INTERIOR RESULTS

Once all the displacements and tractions are known on the boundary interior quantities of displacement, stress and strain can be determined at any point in or on the body. For displacement the conventional boundary displacement integral equation is employed for point in the interior of the body. The displacements for points on the surface of the body can be calculated by simply using the appropriate shape function for the element on which the point is contained.

Equations for strains are derived from the displacement integral equations and the strain-displacement relations. Thereafter, equations for stress are obtained by substituting the resulting strain equations into the appropriate constitutive equation.

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 \( \sigma_{ij} \) referred to these local axes (indicated by overbars) are then given by:

\[
\begin{align*}
\bar{\sigma}_{11} &= \frac{\nu}{1-\nu} \bar{t}_3 + \frac{E \nu}{1-\nu^2} (\bar{\varepsilon}_{11} + \bar{\varepsilon}_{22}) + \frac{E}{1+\nu} \bar{\varepsilon}_{11} \\
\bar{\sigma}_{12} &= \bar{\sigma}_{21} = \frac{E}{2(1+\nu)} \bar{\varepsilon}_{12} \\
\bar{\sigma}_{22} &= \frac{\nu}{1-\nu} \bar{t}_3 + \frac{E \nu}{1-\nu^2} (\bar{\varepsilon}_{11} + \bar{\varepsilon}_{22}) + \frac{E}{1+\nu} \bar{\varepsilon}_{22} \\
\bar{\sigma}_{33} &= \bar{t}_3 \\
\bar{\sigma}_{32} &= \bar{\sigma}_{23} = \bar{t}_2 \\
\bar{\sigma}_{31} &= \bar{\sigma}_{13} = \bar{t}_1
\end{align*}
\]  

(3.13)

where \( E \) is the Young's modulus, \( \varepsilon_{ij} \) defines the components of the elastic strains in the local axes obtained from the derivative of the shape functions and nodal displacements (minus nonlinear strains) and \( \bar{t}_i \) are the traction on the boundary.
Boundary integral formulations for any problem represent the exact statement of the problem posed. In order to maximize the benefits it is essential not to introduce any unnecessary approximations purely for the ease of programming. In this regard, the use of nonconforming elements should be avoided. Although the use of such elements reduces skilled programming effort by nearly an order of magnitude, it becomes virtually impossible to analyze any realistic three-dimensional problem by such a system. Indeed, a pilot study to compare the relative efficiency and accuracy of conforming and nonconforming elements shows that conforming elements are more accurate for the same number of quadratic elements. Additionally, computing costs are much higher for nonconforming elements [30]. For example, with 6, 10, 14 and 18 elements the nonconforming element consumed respectively 4, 5, 7 and 10 times more CPU time than conforming elements. It should be noted that the early work of Lachat and Watson [31,32] on conforming elements provided the inspiration for the present implementation.
Since BEST3D 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.
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 conjunction 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 AVAILABILITY OF BEST3D

BEST3D is written in FORTRAN 77 and is therefore adaptable to any computer which uses such a compiler. Since most of the development work has been carried out on IBM and CRAY mainframes as well as HP-9000 and SUN-4 desktop workstations, it is most easily ported to these systems.
BEST3D 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 BEST3D 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.

In the current version of BEST3D the volume of data stored on disc during the execution of the code is high. This fact is rather critical to a workstation based user who must have atleast 300MB user area to run any reasonable practical problem. It is proposed to develop a more efficient disc usage strategy in the next release of BEST3D.
4.4 SPECIAL FEATURES OF BEST3D

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, volume, hole and insert discretizations, or they may be used to define a point of a thermal hot spot or a sampling point. Points are usually user defined, however, BEST3D 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 BEST3D 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.

Source Points - refers to boundary source points, or boundary and volume source points in an analysis. Source points are used in the functional representation of variables across a boundary element, across a volume cell or in a global shape function representation. In a system equation, unknowns are retained at 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 **BEST3D** 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)

**Volume Source Points** - are points in a volume discretization which are used to represent the functional variation of certain variables through the volume of the body via volume cells or a global shape function variation. These are required only in nonlinear analysis or when the body is subjected to certain types of body forces. In the case of nonlinear analysis, unknowns are retained at volume source points which have to be solved for, along with the unknowns at the boundary source points. This entails writing additional equations at each volume source point. In the case of body forces, the variables are known quantities and additional equations are therefore unnecessary. Volume source points are selected by **BEST3D** based on the type of functional variation selected by the user. For the volume cell approach see VOLU and TYPE cards under **GMR input in Chapter 5.** For the global shape function approach, volume source points are taken at every node defined in the global shape function (see GLOB and NODE cards under **GMR input in Chapter 5.**)

**Sampling Points** - are (user defined) points in the interior of the body or on the surface of the body for which results are requested. Results at sampling point are calculated after the system equation is solved. Sampling points are input on a separate list (see SAMP card in **GMR input in Chapter 5) and are totally independent of the point list used for boundary and volume discretization and independent of the point list for holes and inserts. A sampling point may coincide with a boundary and volume discretization point (but not with a hole or insert). Sampling points should use unique node numbering.

**Hole Elements** - are curvilinear line elements in three dimensional space that are used to represent a surface of a tubular shaped hole in a 3-D body. The line element represents the centerline of the hole and the surface of the hole is at a prescribed radius from the centerline. Basically, the hole element is a computationally efficient and convenient way to model a tubular hole in a 3-D body.

**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. The insert element is modelled similar to a hole element except, instead of a void, the material of a prescribed modulus is assumed to fill the void.
**Volume Cells** - Certain analyses require an integration of some variable over all or part of the volume of the body. In this cases the volume is divided into smaller parts called volume cells, where interpolation functions (of some order) are used to represent the variation of the variable to be integrated across the volume cell. In some analyses a global shape function may be used as an alternative to volume cells.

**Global Shape Functions** - in certain analyses a volume integral is present in the governing boundary integral equation. Instead of using an expensive volume integration across the body (via volume cell discretization) the problem can be conveniently recast using the particular integral approach to eliminate the presence of the volume integral. If this approach is used and if the variable involved in the particular integral is allowed to assume an arbitrary variation across the body (or subregion) then a global shape function is used to model this variation through the body. The global shape function is a general interpolation function with an arbitrary number of volume source points located at any (reasonable) random position. (see GLOB under **GMR input in chapter 5)

**Generic Modeling Region (GMR)** - in a boundary element analysis the body under investigation may be fictitiously divided in a number of smaller parts for convienience 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.)
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.
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.
4.4.4 PROBLEMS OF EDGES AND CORNERS

In the case of a potential flow problem the potential 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)

\[ \beta u^p = \sum_{q=1}^{N} \left[ \left( \int_{\Delta s} G^{pq} N^q ds \right) t_n - \left( \int_{\Delta s} F^{pq} N^q ds \right) u_n \right] \]

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

\[ G t - F u = 0 \]

where

\[ F = 3m \times 3m \text{ matrix for three-dimensional problems} \]

and

\[ G = 3m \times (3m + 4) \text{ matrix for three-dimensional problems} \]

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 \( 3m \times 3m \) for three-dimensional problems. If the displacements alone are specified at the corner (Fig. 4.1c) it is not possible to solve Eq. (4.2) and an alternative approach must be found.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{Fig41.pdf}
\caption{The problems at a corner of the boundary}
\end{figure}

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 continuous if correct results are to be obtained.
The basic input required by BEST3D 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 BEST3D is intended to be as simple as possible, consistent with the demands of a general purpose analysis program. 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 ELASTODYNAMICS - TEST CASE
TRANSIENT 8 0.5
SYMMETRY QUARTER
```

Improper Usage:

```
transient 8 0.5
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 5 1.0
ELEMENT 1 6 8
```

Improper Usage:

```
CTHERMALPLASTICITY STEADY 5 1.0
PLASTICITY UThermal 5 STEADY 1.0
```
3. **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:**

```
SYlqlqETRY
qUA
```

4. **Floating Point Numbers**

Any real parameters may be input in either FORTRAN E or F format. In the latter case, an Fn.0 is assumed with \( n \leq 8 \).

**Proper Usage:**

```
EMOD 30.E+6
ALPHA 1.E-06
POINTS
0.004 1.110 0.0
```

**Improper Usage:**

```
EMOD 30000000.0
ALPHA 1.-6
POINTS
4.0-3 1.110 0
```
5. 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 BEST3D input processor.

Proper Usage:

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

6. Blank Lines

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

7. 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.

8. ** 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
**BODY
**BCCHANGE
```

There may be multiple data blocks of each type, except for the **CASE block.
1. Ordering of Input Items
   While there is some flexibility in the ordering of lines within BEST3D 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, is recommended.
General Limits of BEST3D

It should be noted that there are certain limits which must be observed in the preparation of input for BEST3D. 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 present limits are summarized below. It is anticipated that certain of these limits may be relaxed in future versions of BEST3D.

<table>
<thead>
<tr>
<th>ENTITY</th>
<th>LIMIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>total source points</td>
<td>1200</td>
</tr>
<tr>
<td>total elements (including holes)</td>
<td>600</td>
</tr>
<tr>
<td>total volume source points</td>
<td>600</td>
</tr>
<tr>
<td>generic modelling regions (GMRs)</td>
<td>15</td>
</tr>
<tr>
<td>surfaces per GMR</td>
<td>15</td>
</tr>
<tr>
<td>source points per GMR</td>
<td>600</td>
</tr>
<tr>
<td>elements (including holes) per GMR</td>
<td>300</td>
</tr>
<tr>
<td>holes per GMR</td>
<td>100</td>
</tr>
<tr>
<td>infinite elements per GMR</td>
<td>11</td>
</tr>
<tr>
<td>interface and cyclic symmetry boundary condition sets</td>
<td>20</td>
</tr>
<tr>
<td>element pairs (interfaces and cyclic)</td>
<td>99</td>
</tr>
<tr>
<td>node pairs (interfaces and cyclic)</td>
<td>350</td>
</tr>
<tr>
<td>spring boundary condition sets</td>
<td>60</td>
</tr>
<tr>
<td>time points in boundary condition sets</td>
<td>20</td>
</tr>
<tr>
<td>user specified element number</td>
<td>99999</td>
</tr>
<tr>
<td>user specified node number</td>
<td>99999</td>
</tr>
</tbody>
</table>

- Definition of the terminology used in this table can be found in Section 4.4.
Individual Data Items

The remainder of this chapter provides detailed information on each of the data items available within BEST3D. 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)
5.6 BODY FORCE DEFINITION (**BODY)
5.7 BOUNDARY CONDITION CHANGE DEFINITION (**BCCH)
5.1 CASE CONTROL

This input section provides BEST3D 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.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1.1</td>
<td>Case Control</td>
<td>Start of case control input</td>
</tr>
<tr>
<td></td>
<td>Input Card</td>
<td></td>
</tr>
<tr>
<td><strong>CASE</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.1.2</td>
<td>Title</td>
<td>Title of job</td>
</tr>
<tr>
<td></td>
<td>TITL</td>
<td></td>
</tr>
<tr>
<td>5.1.3</td>
<td>Times for Output</td>
<td>Times of solution output</td>
</tr>
<tr>
<td></td>
<td>TIME</td>
<td></td>
</tr>
<tr>
<td>5.1.4</td>
<td>Type of Analysis</td>
<td>Elastic analysis Free vibration analysis Heat Transfer analysis Plasticity analysis Transient elastodynamic analysis</td>
</tr>
<tr>
<td></td>
<td>ELAS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FORC</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FREE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>HEAT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PLAS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TRAN</td>
<td></td>
</tr>
<tr>
<td>5.1.5</td>
<td>Geometric and Loading Symmetry Control</td>
<td>Symmetry about Y-Z plane Symmetry about X-Z and Y-Z planes Symmetry about X-Z, Y-Z, and X-Y planes</td>
</tr>
<tr>
<td></td>
<td>SYMM HALF</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SYMM QUAR</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SYMM OCTA</td>
<td></td>
</tr>
<tr>
<td>SECTION</td>
<td>KEYWORD</td>
<td>PURPOSE</td>
</tr>
<tr>
<td>---------</td>
<td>---------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>5.1.6</td>
<td>Special Loading Control</td>
<td>Complex-valued boundary conditions in periodic dynamic analysis</td>
</tr>
<tr>
<td></td>
<td>LOAD COMP</td>
<td></td>
</tr>
<tr>
<td>5.1.7</td>
<td>Restart Facility</td>
<td>Save integration files for future runs</td>
</tr>
<tr>
<td></td>
<td>REST WRIT</td>
<td>Use integration files from previous run</td>
</tr>
<tr>
<td></td>
<td>REST READ</td>
<td></td>
</tr>
<tr>
<td>5.1.8</td>
<td>Integration Precision and Efficiency Optimization</td>
<td>Numerical integration precision</td>
</tr>
<tr>
<td></td>
<td>PREC</td>
<td></td>
</tr>
<tr>
<td>5.1.9</td>
<td>Output Options</td>
<td>Produce echo of input data</td>
</tr>
<tr>
<td></td>
<td>ECHO</td>
<td>Printout displacement and traction results</td>
</tr>
<tr>
<td></td>
<td>PRIN BOUN</td>
<td>Print boundary displacement, stress, strain at nodal points</td>
</tr>
<tr>
<td></td>
<td>PRIN NODA</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PRIN LOAD</td>
<td>Print load calculation</td>
</tr>
</tbody>
</table>

**Case Control**
**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.
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**
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.

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
5.1.4 TYPE OF ANALYSIS

ELAS

Status - OPTIONAL
Full Keyword - ELASTIC
Function - Identifies an elastic analysis.
Input Variables - NONE
Additional Information -

Elastic analysis is the default analysis type. Therefore, if none of the keywords from Section 5.1.5 are present, an elastic analysis will be performed.

Examples of Use -

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

**CASE
TITLE REACTOR PRESSURE VESSEL - LOAD CASE 2B
ELASTIC
FORC  `W1 W2 W3 ... WN

Status - OPTIONAL
Full Keyword - FORCED
Function - Identifies a steady-state forced vibration analysis.
Input Variables -
   W1 (Real) - REQUIRED
   Defines the frequency (in rad/time) of the sinusoidal forcing function at which
   output is required.
   W2 .. WN (Real) - OPTIONAL

Additional Information -
   This input can be continued on more than one card, if required. Each card must
   begin with the keyword FORC. A maximum of twenty forcing frequencies may
   be selected for analysis. A minimum of one forcing frequency must be input.

Examples of Use -
1. Determine the response of an axle driven by loads at frequencies of 15., 30.
   and 60.

   **CASE
   TITLE AXLE - STEADY-STATE FORCED RESPONSE
   FORCED 15. 30. 60.
FREE NMOD CTYPE

Status - OPTIONAL

Full Keyword - FREE-VIBRATION

Function - Identifies a free-vibration analysis.

Input Variables -

NMOD (Integer) - OPTIONAL
Sets the number of eigen frequencies required, starting from the lowest frequency. Default is 5.

CTYPE (Alphanumeric) - OPTIONAL
Allowable value is UNCONstrained. It identifies a problem geometry with unconstrained boundaries.

Additional Information - NONE

Examples of Use -

1. Determine the first four modes of free vibration of a turbine blade.

**CASE
   TITLE TURBINE BLADE A7311 - MODE SHAPES
   FREE 4
HEAT ITYPE NSTEP DELTAT

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.

NSTEP (I) - REQUIRED for transient problems
Sets the number of time steps for which the transient heat conduction analysis is to be carried out.

DELTAT (FP) - REQUIRED for transient problems.
Defines the time step size.

Additional Information -
In the present version, only a constant time step DELTAT is permitted.

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

**CASE
    TITLE TUBE-AND-FIN HEX - STEADY-STATE
    HEAT
PLAS  TYPE  DLOAD  EPSNORM

Status - OPTIONAL

Full Keyword - PLASTICITY

Function - Identifies an analysis involving nonlinear material response.

Input Variables -

TYPE  (Alphanumeric)  -  OPTIONAL
Allowable values are ITER or DIRE. Default is ITER.

ITER - Iterative plasticity algorithm - initial stress is found through an iterative procedure.

DIRE - Direct plasticity algorithm - (modified) stress equations are back substituted into the boundary system to eliminate the unknown initial stresses (for mono-tonic loading only and homogeneous material only).

DLOAD  (Real)  -  OPTIONAL  -  (default = .05)
Sets the maximum permitted stress change, as a fraction of the material yield stress, during a single load increment. This variable determines the size of the internal load increment used in BEST3D.

EPSNORM  (Real)  -  OPTIONAL  -  (default = .005)
Tolerance used to determine convergence of the nonlinear algorithm at a particular load step. For convergence, the sum of the changes in the initial stress divided by the yield stress must be less than EPSNORM.

Additional Information -

The variable DLOAD must be input if EPSNORM is input.

If this card is input, an appropriate volume cell discretization or Particular Integral (Nodal) representation must be defined in the geometry input section, described later.

Examples of Use -

1. Perform elastoplastic analysis of a pressure vessel utilizing the direct plasticity algorithm.

**CASE
TITLE  PRESSURE VESSEL - PLASTICITY
TIMES  1.0  2.0  3.0  4.0
PLASTICITY  DIRECT
TRAN  ITYPE  NSTEP  DELTAT

Status - OPTIONAL
Full Keyword - TRANSIENT
Function - Identifies a transient elastodynamic analysis.
Input Variables -
  ITYPE (Alphanumeric) - OPTIONAL
  Allowable value is LAPLace. Default is time domain transient dynamic analysis.
    LAPLace - Identifies Laplace domain transient dynamic analysis.
  NSTEP (Integer) - REQUIRED
  Sets the number of time steps for which the transient analysis is to be carried out.
  DELTAT (Real) - REQUIRED
  Defines the time step size. Time step size should be DELTAT ≤ \frac{L}{C_p}, where \( C_p \) is the pressure wave velocity and \( L \) is the width of the smallest element.

Additional Input -

TIME VARI  CTYPE

Status - REQUIRED (if TRAN is input for a time domain analysis)
Full Keyword - TIME VARIATION
Function - Identifies type of temporal variation of displacements and tractions.
Input variables -
  CTYPE (Alphanumeric) - REQUIRED
    CONStant - constant variation of field quantities in time.
    LINEar - linear variation of field quantities in time.
Examples of Use -

1. Conduct a transient elastodynamic analysis of a spherical tank using a linear time variation of field variables.

**CASE
TITLE SPHERICAL TANK - SUDDEN PRESSURIZATION
TRANSIENT 10 0.01
TIME VARIATION LINEAR
**GEOMETRIC AND LOADING SYMMETRY CONTROL**

**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.

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.

Symmetry is not available for problems using Global Shape Function discretization of the volume.

In a problem with anisotropic material, the material must also be symmetric with respect to the symmetry imposed using the SYMM card.

Examples of Use -

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

**CASE**

**TITLE** HOLLOW SPHERE WITH INTERNAL PRESSURE

**ELASTIC**

**SYMmery OCTAL**
Case Control

Model this Quadrant only

Figure for **CASE: SYMM card
Two-dimensional Quarter symmetry model
LOAD TYPE

Status - OPTIONAL

Full Keyword - LOADING

Function - Identifies a problem having special loadings (boundary conditions)

Input Variables -

- TYPE (Alphanumeric) - REQUIRED
- Allowable values are COMP, SYMM
  - COMP - Complex-valued loadings (boundary condition) for periodic elastodynamic analysis only

Additional Information -

- When the control card "LOAD COMP" is used in periodic dynamics, all the boundary conditions have to be input with both real parts and imaginary parts.

Examples of Use -

1. Specify a complex valued loading and displacement boundary condition for periodic elastodynamic (Forced vibration) analysis. The boundary condition set must specify the real and imaginary parts for both loads and displacements.

**CASE
  TITLE EMBEDDED STRIP FOOTING
  SYMMETRY HALF
  FORCED .001 0.9 1.5
  $ COMPLEX-VALUED LOADING AND DISPLACEMENT B.C
  LOADING COMPLEX
**5.1.7 RESTART FACILITY**

**REST RTYPE**

**Status - OPTIONAL**

**Full Keyword - RESTART**

**Function -** Enables the restart facility for integration.

**Input Variables -**

- RTYPE (Alphanumeric) - REQUIRED
  - Allowables 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.

When employing RESTart READ it is the user's responsibility to ensure that the proper integration files exist.

Geometry and material properties must be the same for both the RESTart WRITe and RESTart READ data sets. However, no checking is done by BEST3D. 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
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 recommended.
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.

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

```plaintext
**CASE
TITLE PRESSURE VESSEL
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, and INTE.

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 isotropic 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

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 always printed.

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
This input section defines the linear and, when required, the nonlinear material 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.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2.1 Material Property Input Card</td>
<td><strong>MATE</strong></td>
<td>Beginning of a material property input set</td>
</tr>
<tr>
<td>5.2.2 Material Identification</td>
<td>ID</td>
<td>Identifier of a material type</td>
</tr>
<tr>
<td></td>
<td>LIBR</td>
<td>material library</td>
</tr>
<tr>
<td>5.2.3 Mass Parameter</td>
<td>DENS</td>
<td>material mass density</td>
</tr>
<tr>
<td>5.2.4 Isotropic Elastic Parameters</td>
<td>EMOD</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td></td>
<td>POIS</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>5.2.5 Isotropic Thermal Parameters</td>
<td>COND</td>
<td>conductivity of material</td>
</tr>
<tr>
<td></td>
<td>SPEC</td>
<td>specific heat</td>
</tr>
<tr>
<td>5.2.6 Isotropic Temperature-dependent Thermoelastic Parameters</td>
<td>TEMP</td>
<td>temperature values at which elastic material properties will be defined</td>
</tr>
<tr>
<td></td>
<td>EMOD</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td></td>
<td>ALPH</td>
<td>co-efficients of thermal expansion</td>
</tr>
</tbody>
</table>
### Material Property Definition

#### 5.2.7 Anisotropic Elastic Parameters

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANIS</td>
<td>identifier for anisotropic material</td>
</tr>
<tr>
<td>STIF</td>
<td>material stiffnesses</td>
</tr>
<tr>
<td>COMP</td>
<td>material compliances</td>
</tr>
<tr>
<td>TECH</td>
<td>Technical constants for an orthotropic material</td>
</tr>
<tr>
<td>ORIE</td>
<td>orientation of the axis of transverse isotropy in 3-D, or, of one material axis with respect to the corresponding geometric axis for anisotropy in 2-D</td>
</tr>
</tbody>
</table>

#### 5.2.8 Isotropic Viscous Parameters

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>DAMP</td>
<td>viscous damping co-efficient</td>
</tr>
</tbody>
</table>

#### 5.2.9 Additional Elastoplastic and Viscoplastic Parameters

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>INEL</td>
<td>signals the beginning of an inelastic material model input</td>
</tr>
<tr>
<td>TIME</td>
<td>time for reversal of loading</td>
</tr>
</tbody>
</table>

##### 5.2.9.1 Von Mises Model

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>VON</td>
<td>Von Mises material model</td>
</tr>
<tr>
<td>YIEL</td>
<td>proportional limits of linear elastic behavior</td>
</tr>
<tr>
<td>CURV</td>
<td>stress-strain curve for Von Mises model with isotropic hardening</td>
</tr>
</tbody>
</table>

##### 5.2.9.2 Two Surface Model

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>TWO</td>
<td>two-surface material model</td>
</tr>
<tr>
<td>YIEL</td>
<td>proportional limits of linear elastic behavior</td>
</tr>
<tr>
<td>HARD</td>
<td>inner and outer proportional limits</td>
</tr>
</tbody>
</table>

##### 5.2.9.3 Walker’s Model

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>WALK</td>
<td>Hastaloy-X viscoplastic material model</td>
</tr>
</tbody>
</table>

**Note:** Refer to the following table for a list of required material properties corresponding to a particular type of analysis.
A list of material properties required for different types of analysis are defined below:

### REQUIRED MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>TYPE OF ANALYSIS</th>
<th>MATERIAL PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Isotropic Elastic Stress Analysis</td>
<td>EMOD, POIS</td>
</tr>
<tr>
<td></td>
<td>(TEMP: optional)</td>
</tr>
<tr>
<td></td>
<td>(ALPH: if thermal body force is present)</td>
</tr>
<tr>
<td></td>
<td>(DENS: if centrifugal body force is present)</td>
</tr>
<tr>
<td></td>
<td>(DENS: if inertial body force is present)</td>
</tr>
<tr>
<td>2. Dynamic Analysis:</td>
<td></td>
</tr>
<tr>
<td>2a. Free-Vibration Analysis</td>
<td>EMOD, POIS, DENS</td>
</tr>
<tr>
<td></td>
<td>(TEMP: optional)</td>
</tr>
<tr>
<td>2b. Steady-state or Laplace Domain Transient Analysis</td>
<td>EMOD, POIS, DENS, DAMP</td>
</tr>
<tr>
<td></td>
<td>(TEMP: optional)</td>
</tr>
<tr>
<td>2c. Time Domain Transient Dynamic Analysis</td>
<td>EMOD, POIS, DENS</td>
</tr>
<tr>
<td></td>
<td>(TEMP: optional)</td>
</tr>
<tr>
<td>3. Anisotropic Elastic Stress Analysis</td>
<td>STIF (or COMP or TECH), ORIE</td>
</tr>
<tr>
<td></td>
<td>(DENS: if centrifugal body force is present)</td>
</tr>
<tr>
<td></td>
<td>(DENS: if inertial body force is present)</td>
</tr>
<tr>
<td>4. Plasticity Analysis:</td>
<td></td>
</tr>
<tr>
<td>4a. Von Mises model</td>
<td>EMOD, POIS, YIEL, CURV</td>
</tr>
<tr>
<td></td>
<td>(TEMP: optional)</td>
</tr>
<tr>
<td></td>
<td>(ALPH: if thermal body force is present)</td>
</tr>
<tr>
<td></td>
<td>(DENS: if centrifugal body force is present)</td>
</tr>
<tr>
<td></td>
<td>(DENS: if inertial body force is present)</td>
</tr>
</tbody>
</table>
### TYPE OF ANALYSIS

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>4b.</td>
<td>Two surface model</td>
</tr>
<tr>
<td>4c.</td>
<td>Viscoplastic (Walker) model</td>
</tr>
</tbody>
</table>

### Heat Transfer Analysis

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5a.</td>
<td>Steady-state (Potential Flow)</td>
</tr>
<tr>
<td>5b.</td>
<td>Transient (Diffusion)</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>EMOD, POIS, YIEL, HARD</td>
<td></td>
</tr>
<tr>
<td>(TEMP: optional)</td>
<td></td>
</tr>
<tr>
<td>(ALPH: if thermal body force is present)</td>
<td></td>
</tr>
<tr>
<td>(DENS: if centrifugal body force is present)</td>
<td></td>
</tr>
<tr>
<td>(DENS: if inertial body force is present)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Library ID</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>COND</td>
<td>COND, DENS, SPEC</td>
</tr>
</tbody>
</table>
**MATERIAl PROPERTY INPUT CARD**

**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
5.2.2 MATERIAL IDENTIFICATION

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.
Material Property Definition

LIBR

Status - OPTIONAL

Full Keyword - LIBRARY

Function - Indicates the material stated by name on the material ID card is contained in a material library.

Input Variables - NONE

Additional Information -

In the present version of the program the material library can only be used for viscoplastic analysis using the Walker's model.

The materials listed below are presently available in the material library. The name listed below should appear on the material ID name.

Available Material Library

<table>
<thead>
<tr>
<th>ID Name</th>
<th>Material Name</th>
<th>Type of Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1100H</td>
<td>IN-100</td>
<td>Precipitation Aged Superalloy (Nickel Based Material)</td>
</tr>
<tr>
<td>P1455H</td>
<td>PWA1455</td>
<td>Precipitation Aged Superalloy (Nickel Based Material)</td>
</tr>
<tr>
<td>P1038H</td>
<td>HASTELOYS-X</td>
<td>Solid Solution Strengthened Superalloy</td>
</tr>
<tr>
<td>P1038X</td>
<td>HASTELOYS-X</td>
<td>Solid Solution Strengthened Superalloy</td>
</tr>
</tbody>
</table>

NOTE: If HASTELOYS-X is desired, then P1038H is recommended. P1038X is similar, however, it uses a different viscoplastic algorithm (integral form).

Examples of Use -

1. Material input for Walker's model

```plaintext
**MATE
ID
LIBR

INELASTIC
TIME  5.0  10.0  15.0
WALKER
$
```

BEST3D User Manual April, 1990 Page 5.34
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
5.2.4 ISOTROPIC ELASTIC PARAMETERS

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 an elastic material.

**MATERIAL
  ID MAT1
  EMOD    1.E6
  POIS    0.36
  DENs    0.15
Material Property Definition

**POIS**

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 - $1.0 < \text{POI} \leq 0.5$

Additional Information - NONE

Examples of Use -


```plaintext
**MATE
  ID  STEEL
  ENOD  30.E6
  POIS  0.30
```

BEST3D User Manual
April, 1990
Page 5.37
5.2.5 ISOTROPIC THERMAL PARAMETERS

COND CD1

Status - REQUIRED (for 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 a steady-state heat conduction.

**MATE
  ID ALUM
  CONDUCTIVITY 25.0
SPEC    SP1

Status - REQUIRED (for 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 -

   **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.
It is important to note that for anisotropy, no more than one value of temperature, viz, TEM1 can be given, i.e., the elastic anisotropic material properties are essentially kept as temperature independent in this version of the code.

Examples of Use -

1. Define a material with temperature varying Young's modulus.

**MATE
   ID MATOIT
   TEMP 50.0 300.0 500.0
   EMOD 30.3+6 29E+6 27.5E+6
   POIS 0.3
Material Property Definition

**Figures for **MATE: TEMP, EMOD and ALPHA cards

Definition for temperature dependent elastic modulus and thermal coefficient of expansion
Material Property Definition

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 1.0E6 0.95E6 0.9E6 0.83E6 0.7E6
      EMOD 0.6E6 0.35E6
      POIS 0.36
      DEHS 0.15
ALPH AL1 AL2 ... ALN

Status - REQUIRED (for thermal stress analysis)

Full Keyword - ALPHA

Function - Defines the values of co-efficient 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 10.0E3 9.8E3 8.0E3
 ALPH 13.0E-6 0.0E-6 0.0E-6
 POIS 0.33
```
ANISOTROPIC ELASTIC PARAMETERS

5.2.7

Material Property Definition

ANIS          ITYPE

Status - OPTIONAL

Full Keyword - ANISOTROPY

Function - This card identifies the material as (macroscopic, homogeneous) anisotropic and defines the type of anisotropy.

Input Variables -

ITYPE (Alphanumeric) - REQUIRED

Allowable values are GENE, ORTH, TRAN and CUBI

- GENE - General anisotropy. In 3D general anisotropy requires 21 independent constants.
- ORTH - Orthotropy or Orthogonal anisotropy (three planes of elastic symmetry). In 3D orthotropy requires 9 independent constants.
- TRAN - Traverse isotropy or Cross isotropy or Plane of isotropy. (Directionally solidified material.) In 3D this requires 5 independent constants.
- CUBI - Cubic crystal anisotropy. This requires 3 independent constants in 3D.

Additional Information -

Note that for complete isotropy or insignificant anisotropy, the anisotropic elastic material properties should not be used, rather an isotropic elastic analysis should be opted for.

Anisotropy is not available in axisymmetric analysis. The anisotropic material properties are input (as defined on the subsequent pages) for an arbitrary material coordinate system. The material coordinate system can then be rotated to a particular orientation (using the ORIE card) with respect to the global coordinate system which is used to define the geometry of the body.

If geometric and loading symmetry (the SYMM card in **CASE input) is being used in an anisotropic analysis, then the material (in its final orientation) must also be symmetric with respect to the imposed symmetry. This is the users responsibility. Moreover, the SYMM card should not be used with general anisotropy.
Examples of Use -

1. Define option for anisotropic elastic analysis together with the type of anisotropy.

```plaintext
**MATE
ID MAT1
TEMP 70.0
DENSITY 10.0
ANIS ORTHO
TECH 1.31E+5 0.13E+5 0.13E+5 0.038 0.038 0.492 0.064E+5
TECH 0.032E+5 0.032E+5
```
Material Property Definition

STIF  S11 ... SNN

Status - OPTIONAL (if ANIS is input, either STIF, COMP or TECH must be input)

Full Keyword - STIFFNESSES

Function - Defines the material stiffness for an anisotropic material.

Input Variables -

S11 ... SNN (Real) - REQUIRED in the order defined below

GENE S11, S12, S13, S14, S15, S16, S22, S23, S24, S25, S26
S33, S34, S35, S36, S44, S45, S46, S55, S56, S66

OTHO S11, S12, S13, S22, S23, S33, S44, S55, S66

TRAN S11, S13, S33, S44, S55

CUBI S11, S12, S44

Additional Information -

The relevant stiffnesses appear in the stress-strain relationship as shown for the following cases.

GENE - General Anisotropy

\[
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{xz}
\end{pmatrix}
= 
\begin{pmatrix}
S_{11} & S_{12} & S_{13} & S_{14} & S_{15} & S_{16} \\
S_{22} & S_{23} & S_{24} & S_{25} & S_{26} \\
S_{33} & S_{34} & S_{35} & S_{36} \\
S_{44} & S_{45} & S_{46} & \gamma_{xy} & \gamma_{xz} & \gamma_{yz}
\end{pmatrix}
\begin{pmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{zz} \\
\gamma_{xy} \\
\gamma_{xz}
\end{pmatrix}
\]

OTHO - Orthotropy

\[
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{xz} \\
\sigma_{yz}
\end{pmatrix}
= 
\begin{pmatrix}
S_{11} & S_{12} & 0 & 0 & 0 \\
S_{22} & S_{23} & 0 & 0 & 0 \\
S_{33} & 0 & 0 & 0 & 0 \\
S_{44} & 0 & 0 & 0 & 0 \\
S_{55} & 0 & 0 & 0 & 0 \\
S_{66} & 0 & 0 & 0 & 0
\end{pmatrix}
\begin{pmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{zz} \\
\gamma_{xy} \\
\gamma_{xz} \\
\gamma_{yz}
\end{pmatrix}
\]
TRAN - Transverse Isotropy

\[
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{yz}
\end{pmatrix} =
\begin{pmatrix}
S_{11} & (S_{11} - 2S_{44}) & S_{13} & 0 & 0 \\
S_{11} & S_{13} & 0 & 0 & 0 \\
S_{33} & S_{33} & 0 & 0 & 0 \\
&S_{44} & 0 & 0 & 0 \\
& & & S_{55} & 0 \\
& & & & S_{55}
\end{pmatrix}
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{yz}
\end{pmatrix}
\]

CUBI - Cubic Crystal Anisotropy

\[
\begin{pmatrix}
\sigma_{zz} \\
\sigma_{yy} \\
\sigma_{xx} \\
\sigma_{yz}
\end{pmatrix} =
\begin{pmatrix}
S_{11} & S_{12} & S_{12} & 0 & 0 & 0 \\
S_{11} & S_{12} & 0 & 0 & 0 & 0 \\
S_{11} & 0 & 0 & 0 & 0 & 0 \\
&S_{44} & 0 & 0 & 0 & 0 & 0 \\
& & & S_{44} & 0 & 0 \\
& & & & & S_{44}
\end{pmatrix}
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{yz}
\end{pmatrix}
\]

IMPORTANT NOTES:

- The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.
- The transversely isotropic material must be input using the convention that the axis of isotropy is the z-axis. The actual orientation of the material for this and other types of anisotropy is set using the ORIE card.
- The material stiffnesses appearing on the STIF card may be split and put onto two or more cards with each new card beginning with the keyword STIF. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input stiffness coefficients for a 3D transverse isotropic medium relative to the principal material axes.

**MAT

<table>
<thead>
<tr>
<th>ID</th>
<th>MAT1</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEMP</td>
<td>70.0</td>
</tr>
<tr>
<td>DENSITY</td>
<td>1.0</td>
</tr>
<tr>
<td>ANIS</td>
<td>TRAN</td>
</tr>
<tr>
<td>STIFF</td>
<td>2.1601E+7 0.9691E+6 2.0832E+7 0.65E+6 1.0279E+6</td>
</tr>
</tbody>
</table>
Material Property Definition

COMP C11 ... CNN

Status - OPTIONAL (if ANIS is input, either STIF, COMP or TECH must be input)

Full Keyword - COMPLIANCES

Function - Defines the material compliances for an anisotropic material.

Input Variables -

C11 ... CNN (Real) - REQUIRED in the order defined below

GENE C11, C12, C13, C14, C15, C16, C22, C23, C24, C25, C26, C33, C34, C35, C36, C44, C45, C46, C55, C56, C66

OTHO C11, C12, C13, C22, C23, C33, C44, C55, C66

TRAN C11, C13, C33, C44, C55

CUBI C11, C12, C44

Additional Information -

The relevant compliances appear in the stress-strain relationship as shown for the following cases.

GENE - General Anisotropy

\[
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{xy} \\
\gamma_{yz}
\end{pmatrix}
= \begin{pmatrix}
C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\
C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\
C_{33} & C_{34} & C_{35} & C_{36} \\
C_{44} & C_{45} & C_{46} \\
Symm.
\end{pmatrix}
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz}
\end{pmatrix}
\]

OTHO - Orthotropy

\[
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{xy} \\
\gamma_{yz}
\end{pmatrix}
= \begin{pmatrix}
C_{11} & C_{12} & 0 & 0 & 0 \\
C_{22} & C_{23} & 0 & 0 & 0 \\
C_{33} & C_{34} & C_{35} & C_{36} \\
Symm.
\end{pmatrix}
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz}
\end{pmatrix}
\]

TRAN - Transverse Isotropy

\[
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{xy} \\
\gamma_{yz}
\end{pmatrix}
= \begin{pmatrix}
C_{11} & (C_{11} - C_{12})/2 & 0 & 0 & 0 \\
C_{13} & C_{13} & 0 & 0 & 0 \\
C_{33} & C_{33} & C_{34} & 0 & 0 \\
Symm.
\end{pmatrix}
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz}
\end{pmatrix}
\]

BEST3D User Manual April, 1990 Page 5.48
CUBI - Cubic Crystal Anisotropy

\[
\begin{pmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{zz} \\
\gamma_{xy} \\
\gamma_{yz}
\end{pmatrix} =
\begin{pmatrix}
C_{11} & C_{12} & 0 & 0 & 0 \\
C_{11} & C_{12} & 0 & 0 & 0 \\
C_{11} & 0 & C_{44} & 0 & 0 \\
0 & 0 & C_{44} & 0 & 0 \\
Symm. & & & &
\end{pmatrix}
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz}
\end{pmatrix}
\]

IMPORTANT NOTES:

- The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.
- The transversely isotropic material must be input using the convention that the axis of isotropy is the z-axis. The actual orientation of the material for this and other types of anisotropy use the ORIE card.
- The material compliances appearing on the COMP card may be split and put onto two or more cards with each new card beginning with the keyword COMP. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input compliance coefficients for a 3D transverse isotropic medium relative to the principal material axis.

**MATE
ID MAT1
TEMP 70.0
DENSITY 1.0
AXIS TRAN
COMP 4.7619E-7 -1.4762E-9 6.4516E-7 1.5365E-8 7.2750E-8
Material Property Definition

TECH E1 E2 E3 ν12 ν13 ν23 G12 G13 G23

Status - OPTIONAL (if ANIS is input, either STIF, COMP, or TECH must be input)

Full Keyword - TECHNICAL

Function - Inputs pertinent technical (or engineering) constants for an orthotropic material.

Input Variables -
E1, E2, E3, ν12, ν13, ν23, G12, G13, G23 (Real) - REQUIRED

Additional Information -

The TECH card can only be used for orthotropic analysis.

For the present case of an orthotropic material the technical constants may be described most conveniently by the following strain-stress relationship:

\[
\begin{pmatrix}
\epsilon_{xx} \\
\epsilon_{yy} \\
\epsilon_{zz} \\
\gamma_{yz} \\
\gamma_{xz} \\
\gamma_{xy}
\end{pmatrix}
= \begin{pmatrix}
1/E_1 & -\nu_{12}/E_1 & -\nu_{13}/E_1 & 0 & 0 & 0 \\
1/E_2 & -\nu_{23}/E_2 & 0 & 0 & 0 & 0 \\
1/E_3 & 0 & 0 & 0 & 0 & 0 \\
1/G_{12} & 0 & 0 & 1/G_{13} & 0 & 0 \\
1/G_{23} & 0 & 0 & 0 & 1/G_{23} & 0 \\
Symm.
\end{pmatrix}
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{xz} \\
\sigma_{yz}
\end{pmatrix}
\]

where \( E_1\nu_{21} = E_2\nu_{12}, E_2\nu_{32} = E_3\nu_{23}, E_3\nu_{13} = E_1\nu_{31} \)

IMPORTANT NOTES:

- The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.

- The material parameter appearing on the TECH card may be split and put onto two or more cards with each new card beginning with the keyword TECH. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input technical constants (also called ‘engineering constants’) relevant to a 3D analysis for an orthotropic medium relative to the principal material axes.

**MATE
ID MAT1
TEMP 70.0
DENSITY 1.0
ANIS ORTH
TECH 21.0E+6 1.55E+6 1.55E+6
TECH 0.31 0.31 0.49
TECH 0.65E+6 0.5E+6 0.5E+6
ORIE   A1 ... BN

Status - OPTIONAL
Full Keyword - ORIENTATION
Function - Defines the orientation of material axes in an anisotropic analysis with respect to the global axes of the problem. The geometry of the body is defined with respect to the global axes.

Input Variables -
A1,A2,A3,B1,B2,B3 (Real) - REQUIRED

where
A1, A2, A3 are the cartesian components of a vector parallel to the material x-axis in the global (geometric) coordinate system, and
B1, B2, B3 are the cartesian components of a vector parallel to the material y-axis in the global (geometric) coordinate system.

Additional Information -
This card can be omitted if the material axes are aligned with the geometric axes.

The STIFF, COMP, TECH and ALPH cards when present must precede the ORIE card.

If geometric and loading symmetry (the SYMM card) is being used in an anisotropic analysis then the material (after rotation) must be symmetric with respect to the imposed symmetry. This is the user's responsibility. Moreover, the SYMM card should not be used with general anisotropy.

The x-axis and the y-axis defined on the ORIE card must be at right angles. The third (z) axis will be calculated by BEST3D.

Examples of Use -
1. For 3D transverse isotropy, provide the directional cosines for material axes at an oblique rotation.
**5.2.8 ISOTROPIC VISCOUS PARAMETERS**

**DAMP**   **DAMPR**

Status - REQUIRED (for steady-state forced vibration or Laplace domain transient dynamic analysis)

Full Keyword - DAMPING

Function - Defines the viscous damping coefficient for the material.

Input Variables -

DAMPR (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Define the viscous damping coefficient for material MAT1

```
**MATERIAL
ID MAT1
TEMP 70.
EMOD 2.6666
POIS 0.3333
DENS 1.0
DAMP 0.05
```
INEL

Status - REQUIRED (for plastic 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 four models to be chosen. These models are specified using the VON, TWO or WALK keywords. Only one of these cards may be used in a single material property input data set. Input for the 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.

```plaintext
**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 extrapolate in the wrong direction at times when loading reverses.

A maximum of 20 time steps may be input.

Examples of Use -

1. Specify a reversal of loading takes place at two time step numbers during cyclic loading of a material.

   **MATERIAL
   ID MAT1
   TEMP 70.0
   ENOD 23.8E+03
   POIS 0.3
   INELASTIC
   TWO SURFACE
   YIELD 100.0 400.0
   HARD 35.7E+03 11.9E+03
   TIME 7 17
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. For applications involving cyclic loading and reverse yielding use of the two-surface model (keyword=TWO) is recommended.

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
Material Property Definition

Von Mises model with variable strain hardening

\[ \sigma_{\text{eq}} = 2000.0 \]

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
**Material Property Definition**

**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 Nine points (excluding the initial point can be input by the user.

Linear interpolation is used between points. Perfectly plastic material is assumed after the last point.
Material Property Definition

Examples of Use -

1. Define the equivalent stress-equivalent plastic strain curve for a Von Mises material.

```
**MATERIAL
ID MAT1
TEMP 70.0
E MOD 7000.0
POIS 0.2
INELASTIC
VON MISES
YIELD 100.
CURVE
  150.  0.25
  180.  0.5
  190.  0.75
  200.  1.0
```
TWO

Status - OPTIONAL
Full Keyword - TWO SURFACE
Function - Selects use of the two-surface inelastic material model.
Input Variables - NONE
Additional Information -
   This model is recommended for either monotonic or cyclic loading situations.
Examples of Use -
   1. Select the use of the two-surface inelastic material model.

***MATERIAL
   ID MAT1
   TEMP 70.0
   EMOD 23.8E+03
   POIS 0.3
   INELASTIC
   TWO SURFACE
Material Property Definition

Two-surface plasticity model

Figure for **MATE: TWO (YIELD and HARD) cards
TWO SURFACE equivalent stress/equivalent plastic strain hardening curve
Material Property Definition

YIEL  Y1  Y2

Status - REQUIRED (if TWO is input)

Full Keyword - YIELD

Function - Defines the proportional limits for the two-surface inelastic material models.

Input Variables -

Y1 (Real) - REQUIRED
Inner proportional limit.

Y2 (Real) - REQUIRED
Outer proportional limit.

Additional Information -

Y1 is the normal proportional limit, at which the transition away from purely elastic behavior occurs. Y2 is the stress value at which equivalent plastic strain can be reasonably approximated as a linear function of stress. Alternatively, Y2 can be taken as any stress larger than the highest stress anticipated to occur in the solution.

Examples of Use -

1. Specify the values of inner and outer yield stresses for the two-surface material model.

***MATE

ID MAT1
TEMP 70.0
EMOD 23.8E+03
POIS 0.3
INELASTIC
TWO SURFACE
YIELD 100.0 400.0
HARD HR1 HR2

Status - REQUIRED (with TWO only)

Full Keyword - HARD

Function - Defines the hardening parameters at the inner and outer proportional limits for the two surface model.

Input Variables -

HR1 (Real) - REQUIRED
HR1 = at Y1

HR2 (Real) - REQUIRED
HR2 = at Y2

Additional Information -

Based on the values of the inner and outer hardening and proportional limits, BEST3D calculates a Ramberg-Osgood stress-strain law to be used in the region between the two proportional limits.

Examples of Use -

1. Specify the hardening parameter values for the inner and outer surfaces for a two-surface material model.

**MATE
ID MAT1
TEMP 70.0
EMOD 23.8E+03
POIS 0.3
INELASTIC
TWO SURFACE
YIELD 100.0 400.0
HARD 35.7E+03 11.9E+03
WALK

Status - OPTIONAL
Full Keyword - WALKER
Function - Selects the viscoplastic material model developed by Kevin Walker.
Input Variables - NONE
Additional Information -

The Walker's model must be used in conjunction with one of the predefined materials contained in the material library. The name of the material from the material library must be stated by name of the material ID card and the LIBR card must be included. No other material parameter values are required or allowed for this model.

References for the Walker viscoplastic model:

Examples of Use - Material input for Walker's model.

```plaintext
**MATE
ID P1038H
LIBR
  INEL
  TIME 5.0 10.0 15.0
  WALKER
$(end of material input)
```
5.3 DEFINITION OF GEOMETRY

In the current version of BEST3D, surface geometry is defined using six, seven, eight and nine noded parametric surface patches. These patches can be defined to have either linear or quadratic variation of the primary field variables. Additionally, holes on the interior of a 3-D body can be defined via two and three noded line elements. The three-noded hole 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 - Hole or Insert definition (if desired)
5 - Volume cell connectivity or global shape function definition (if required)
6 - Sampling point definition (if desired)

(Note: Holes, Inserts, Volume cells and Global shape functions are not presently available for all Analysis Types.)

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

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.3.1</td>
<td>Geometry Input Card</td>
<td>**GMR</td>
</tr>
<tr>
<td>5.3.2</td>
<td>Region Identification</td>
<td>ID, MATE, TREF, TINI, EXTE, HALF</td>
</tr>
</tbody>
</table>

BEST3D User Manual
April, 1990
Page 5.65
<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.3.3</td>
<td>Nodal Point Definition</td>
<td>nodal points for boundary and volume discretization</td>
</tr>
<tr>
<td></td>
<td>POIN</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Coordinates)</td>
</tr>
<tr>
<td>5.3.4</td>
<td>Surface Element Definition</td>
<td>beginning of surface discretization</td>
</tr>
<tr>
<td></td>
<td>SURF</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TYPE LINE</td>
<td>linear surface variation of field quantities</td>
</tr>
<tr>
<td></td>
<td>TYPE QUAD</td>
<td>quadratic surface variation of field quantities</td>
</tr>
<tr>
<td></td>
<td>ELEM</td>
<td>element list</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(element connectivity)</td>
</tr>
<tr>
<td></td>
<td>TRAN</td>
<td>surface generation by translation</td>
</tr>
<tr>
<td></td>
<td>REFE</td>
<td>axis of rotation for surface generation</td>
</tr>
<tr>
<td></td>
<td>DIRE</td>
<td>direction of axis of rotation</td>
</tr>
<tr>
<td></td>
<td>ROTA</td>
<td>angle of rotation for surface generation</td>
</tr>
<tr>
<td></td>
<td>NORM</td>
<td>defines outer normal of surface</td>
</tr>
<tr>
<td>5.3.5</td>
<td>Enclosing Element Definition</td>
<td>enclosing element list</td>
</tr>
<tr>
<td></td>
<td>ENCL</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>(enclosing element connectivity)</td>
</tr>
<tr>
<td>5.3.6</td>
<td>Hole Element Definition</td>
<td>start of hole definition</td>
</tr>
<tr>
<td></td>
<td>HOLE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>POIN</td>
<td>nodal points for hole discretization</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(coordinates)</td>
</tr>
<tr>
<td></td>
<td>TYPE LINE</td>
<td>linear variation of field quantities for 3-noded holes</td>
</tr>
<tr>
<td></td>
<td>TYPE QUAD</td>
<td>quadratic variation of field quantities for 3-noded holes</td>
</tr>
<tr>
<td></td>
<td>ELEM</td>
<td>hole element list</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(element connectivity and radius of hole)</td>
</tr>
</tbody>
</table>

Definition of Geometry
### Definition of Geometry

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
</table>
| 5.3.7       | Insert Element Definition | beginning of Insert definition  
|             | INSE          | nodal points for insert discretization  
|             | POIN (coordinates) | linear variation of field quantities of Insert  
|             | TYPE LINE     | quadratic variation of field quantities of Insert  
|             | TYPE QUAD     | start of connectivity for each Insert  
|             | ELEM (element connectivity and radius of insert) | |
| 5.3.8       | Volume Cell Definition | beginning of volume discretization  
|             | VOLU          | linear variation of cell quantities  
|             | TYPE LINE     | quadratic variation of cell quantities  
|             | TYPE QUAD     | volume cell definition  
|             | CELL (cell connectivity) | region completely filled with cells  
| 5.3.9       | Global Shape Function Definition | global shape function in GMR  
|             | GLOB          | global shape function node list  
|             | NODE (node list) | |
| 5.3.10      | Sampling Points | start of definition of sampling points  
|             | SAMP (coordinates) | |
**GMR**

Status - REQUIRED

Full Keyword - GMREGION

Function - This card signals the beginning of the definition of a generic modelling region.

Input Variables - NONE

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, Boundary Condition set, and Body Force Definitions. Each GMR must be a closed region of three dimensional space. However, under the following two circumstances, the region may be open:

1. In planar symmetry problems, the body may be sliced into symmetric parts and only one of these parts requires discretization. The interior section exposed by the plane cutting the body does not represent a boundary, and therefore it does not require discretization.

2. In GMRs with boundaries extending to infinity, a GMR may have open boundaries. However, this must be indicated through the use of the EXTE or HALF cards or by enclosing the open boundary with Enclosing elements (see the ENCL card). Note: One of the above three devices MUST be used in an infinite region. Infinite elements may optionally be used but it cannot replace the use of the EXTE, HALF or ENCL cards.

A GMR may have multiple internal boundaries in addition to a single external boundary.
Examples of Use -

**GMR**

**ID REG1**
MAT STEEL
TREF 70.0
TINI 0.0

POINT
1 10.0 0.0 2.0
2 10.0 1.0 2.0

Definition of Geometry
**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 **BEST3D** output file.

The NAME must be unique compared to all the other GMR names defined in the problem.

Examples of Use -

```plaintext
**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 - This card 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 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**
Definition of Geometry

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.

   **CHR
   ID REG1
   MATE MAT1
   TREF 70.0
   TINI 0.0
EXTE

Status - OPTIONAL
Full Keyword - EXTERIOR
Function - This card identifies that the present GMR is a part of a infinite region.
Input Variables - NONE
Additional Information -

The entire outer boundary of the GMR must extend to infinity.

Infinite elements should not be used in the GMR.

In an analysis of a problem of a body of infinite extent, it is not neccessary to fix the boundary of the body for the sole purpose of preventing rigid body motions. Basically, the mathematics of the problem assumes zero displacement at infinity.

When the entire outer boundary of a GMR is at infinity (e.g cavity in an infinite space) the outer boundary can not and should not be modeled. Instead the EXTE card should be inserted in the GMR input to indicate this fact. The purpose of this card is to account for the contributions of the unmodeled infinite boundary in the calculation of the diagonal terms of the F matrix (Rigid body translation technique).

An alternative method to account for infinite boundaries is to model the infinite boundary with enclosing elements (see ENCL card). However, this is not recommended in problems when the entire outer boundary extends to infinity, since the use of enclosing elements would be more expensive then using the EXTE card.

Examples of Use -

1. Specify that the region GMR1 is part of an infinite region.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  EXTERIOR
  POINTS
     1 0.0 212.00 0.0
     2 41.36 207.93 0.0
     . . . 
     . . . 
     . . . 

BEST3D User Manual  April, 1990  Page 5.74
HALF

Status - OPTIONAL
Full Keyword - HALF
Function - This card identifies that the present GMR is part of a half-space.
Input Variables - NONE
Additional Information -

In an analysis of a problem of a body of infinite extent, it is not necessary to fix the boundary of the body for the sole purpose of preventing rigid body motions. Basically, the mathematics of the problem assumes zero displacement at infinity.

In order to use the HALF card, the GMR must be a half space and the half-space must be modeled with regular boundary elements about the area of interest. As the boundary extends to infinity, the boundary elements can be truncated at a reasonable distance away from the area of interest (or infinite elements can be used).

The purpose of this card is to account for the contribution of the unmodeled (semi-infinite) boundary in the calculation of the diagonal terms of the \( F \) matrix (Rigid body translation technique)

The part of the half-space that is modeled with regular boundary elements may contain irregularities such as a trench, however, the distant elements that extend towards infinity must lie on a plane.

An alternative method to account for the semi-infinite boundary is to model the boundary with enclosing elements (see ENCL card), however, this is not recommended in half-space problems since the use of enclosing elements would be more expensive than using the HALF card.

Examples of Use -

1. Specify that the region GMR1 is a part of a half space.

    **GMR
    ID GMR1
    MAT MAT1
    TREF 70
    HALF
    POINTS
    1 0.00 0.00 0.00
    2 0.70 0.70 0.00
    ...
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 element and volume cell discretization (or global shape function definition) of the GMR.

Input Variables - NONE

Additional Information -

Under certain conditions BEST3D 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 hole and insert elements discretization CANNOT be defined here. Instead, the nodal points for holes and inserts must be defined under their respective section.

Sampling Points for which results are requested (at any point on or in the body) is input under the Sampling Point 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 9999.

Nodal coordinates for both surface and volume discretization should be input here. If a node is not referenced in the surface or volume discretization, then it is ignored.

Nodal points used for hole and insert elements CANNOT be defined here. Instead, the nodal points for holes and inserts must be defined under their respective section.

Sampling Points for which results are requested (at any point on or in the body) is input under the Sampling Point section.

Examples of Use -
1. Define a set of nodal coordinates in GMR1.

**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

BEST3D User Manual April, 1990 Page 5.77
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).

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 versions of BEST3D 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  106  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 SURFNEW 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 +
Definition of Geometry

**TYPE**

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 -

- **ETYPE** (Alphanumeric) - REQUIRED
- Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

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 surfaces of different variation type meet, the element sides along the junction are constrained to have linear variation.

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
ELEMENT
  101 1 2 3 103 203 202 201 101
  102 3 4 5 105 205 204 203 103
  ... ... ... ... ... ...
  ... ... ... ... ... ...
  ... ... ... ... ... ...
```
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
Definition of Geometry

$\texttt{IFLAG NEL NODE1 ... NODEN NODEREF}$

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 -

- $\texttt{IFLAG}$ (Alphanumeric) - OPTIONAL
  IFLAG is omitted except in the case of an infinite element. IFLAG = I identifies an infinite element.
- $\texttt{NEL}$ (Integer) - REQUIRED
  User element number.
- $\texttt{NODE1 ... NODEN}$ (Integer) - REQUIRED
  User node numbers of the six, seven, eight or nine node for defining the geometry of the element. Every surface patch must have three, six, seven, eight or nine nodes, regardless of whether TYPE = LINE or QUAD. (The shape functions for geometry is always quadratic)
- $\texttt{NODEREF}$ (Integer) - OPTIONAL
  User node number for reference node used in infinite element definition. If IFLAG = I and NODEREF is omitted the reference node will be taken to be the origin.

Additional Information -

This card is input once for each element.

The input card need not specify whether a three, six, seven, eight or nine node element is being defined. The starting point in 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 9999.

In the definition of an infinite element the first node input must lie on the fixed edge of the element.

All of the nodes referenced in the surface element connectivity must have been defined previously in POINts.

In GMRs of infinite extent, infinite elements may or may not be used. However, whenever a GMR is of infinite extent either the EXTE or HALF card must be used or ENCLosing elements must be defined.

Examples of Use -
1. Specify the connectivity definition for elements of the surface SIDE using four 8-noded quadratic elements.

<table>
<thead>
<tr>
<th>SURFACE SIDE</th>
<th>TYPE QUAD ELEMENT</th>
<th>TYPE QUAD ELEMENT</th>
<th>TYPE QUAD ELEMENT</th>
<th>TYPE QUAD ELEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>101 1 2 3 103 203 202 201 101</td>
<td>102 3 4 5 105 205 204 203 103</td>
<td>103 5 6 7 107 207 206 205 105</td>
<td>104 7 8 1 105 201 208 207 107</td>
<td></td>
</tr>
</tbody>
</table>
Definition of Geometry

Linear 3-noded Element

Linear 4-noded Element

Quadratic 6-noded Element

Quadratic 8-noded Element

Quadratic 7-noded Element

Quadratic 9-noded Element

Figure for **GMR: element Connectivity card
Three-dimensional (surface) boundary elements
NODE A OR C MUST BE THE FIRST NODE INPUT ON AN INFINITE ELEMENT
Definition of Geometry

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 - 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 SURFNEW 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 SURFNEW 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 SURFNEW 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 SURFNEW 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 +
Definition of Geometry

**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.

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 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**

<table>
<thead>
<tr>
<th>TYPE</th>
<th>QUAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEMENT</td>
<td></td>
</tr>
<tr>
<td>101</td>
<td>1</td>
</tr>
<tr>
<td>102</td>
<td>3</td>
</tr>
<tr>
<td>103</td>
<td>5</td>
</tr>
<tr>
<td>104</td>
<td>7</td>
</tr>
</tbody>
</table>

NORMAL 101 +
Element groups 1 and 2 are disjoint sections and therefore normals for one element in each section must be defined.
Definition of Geometry

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 ENCLOSED ELEMENT DEFINITION

ENCEL

Status - OPTIONAL

Full Keyword - ENCLOSING

Function - Signals the beginning of the connectivity definition for enclosing elements of the GMR. Enclosing elements are used in GMRs of infinite extent in order to create a fictitious boundary required for correct calculation of the matrix coefficients.

Input Variables - NONE

SIGN - (symbol) - REQUIRED

Additional Information -

In a GMR of infinite extent, it is necessary to use either the EXTE or HALF card if enclosing elements are not used.

The nodes in an enclosing element do not become boundary source points (part of the system equation) unless they are also part of a regular boundary. The only purpose of enclosing elements is to define an arbitrary surface for integration so that the contribution of the unmodelled infinite boundary can be taken into account in the calculation of the diagonal terms of the F matrix (Rigid Body Translation Technique).

The geometry of the surface defined by the enclosing elements is arbitrary since the contribution (for a particular source point) of any surface enclosing the region is equivalent. Therefore, the discretization of enclosing elements should be crude, utilizing the minimum number of enclosing elements necessary to enclose the region. It is, however, recommended that the surface defined by the enclosing elements does not pass too close (relative to the size of the enclosing element) to a boundary source point belonging to a regular element contained in that particular region.

In an analysis of a problem of a body of infinite extent, it is not necessary to fix the boundary of the body for the sole purpose of preventing rigid body translation. Basically, the mathematics of the problem assumes zero displacement at infinity.
**Definition of Geometry**

(NONE)   NEL NODE1 .... NODEN

Status - REQUIRED (minimum of one card if ENCL is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the connectivity for a single enclosing element.

Input Variables -

NEL (Integer) - REQUIRED
User element number (required for user's purpose only)

NNODE (Integer) - REQUIRED
User number for the node for defining the geometry of the enclosing element.

NODEN = 8

Additional Information -

Only EIGHT noded elements are allowed.

All of the connectivity for enclosing elements must be defined such that their normals are positive.

Examples of Use -

1. Define enclosing elements

ENCLOSING ELEMENTS

1001 1 2 3 103 203 202 201 101
1002 3 4 5 105 205 204 203 103
etc....
HOLE ELEMENT DEFINITION

**HOLE**

**Status** - OPTIONAL

**Full Keyword** - HOLE

**Function** - This card initiates the definition of embedded holes within the current GMR.

**Input Variables** - NONE

**Additional Information** -
In the present implementation, a hole 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.

**Examples of Use** -
1. Define a hole consisting of two quadratic hole elements.

```plaintext
HOLE
POINTS
1001 0.5 0.5 0.0
1002 0.5 0.5 0.25
1003 0.5 0.5 0.5
1004 0.5 0.5 0.75
1005 0.5 0.5 1.0
TYPE QUAD
ELEMENT
101 0.1 1001 1002 1003
102 0.1 1003 1004 1005
$(end of hole input)
```
POIN

Status - REQUIRED (if HOLE is input)
Full Keyword - POINTS
Function - This card initiates the definition of hole nodal points for the current GMR.
Input Variables - NONE
Additional Information -
Hole nodal points must be defined separately from the surface nodal points.

(NONE) NNODE X Y Z

Status - REQUIRED (if HOLE is input)
Full Keyword - NO KEYWORD REQUIRED
Function - This card defines the node number and the Cartesian coordinates for hole node.
Input Variables -
    NNODE (Integer) - REQUIRED
    User number for the hole node.
    X,Y,Z (Real) - REQUIRED Cartesian coordinates of the of the hole node.
Additional Information -
This card is input once for each point.
User node numbering must be unique within the problem. This includes surface, hole and interior points.
All point numbers must be less than or equal to 9999.
All hole nodes should be located on the interior of the part. Additionally, hole nodes should not be coincident with surface or sampling points.

Examples of Use -
1. Define a set of node numbers and the cartesian coordinates for nodes on two hole elements.

    HOLE 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
Definition of Geometry

**TYPE**

Status - REQUIRED (if HOLE is input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities in the longitudinal direction for three-noded hole elements.

Input Variables -

**ETYPE** (Alphanumeric) - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

The TYPE card defines the variation for three-noded hole elements only.

Only one TYPE card for holes is allowed per GMR.

All two-noded hole elements are assumed as linear longitudinal variation of the field quantities.

All of the three-noded hole elements in a single GMR must have the same type of variation.

This card has no effect on the circumferential variation of the field variables on the hole.

Examples of Use -

1. Specify that all three-noded hole elements appearing in the element list for hole elements will have a linear variation of field quantities in the longitudinal direction.

**HOLE**

POUNTS

<table>
<thead>
<tr>
<th>POINT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>1.0</td>
<td>0.0</td>
<td>1.0</td>
</tr>
<tr>
<td>102</td>
<td>1.0</td>
<td>0.25</td>
<td>1.0</td>
</tr>
<tr>
<td>103</td>
<td>1.0</td>
<td>0.5</td>
<td>1.0</td>
</tr>
<tr>
<td>104</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

**ETYPE LINEAR**

**ELEMENT**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1001</td>
<td>0.2</td>
<td>101</td>
<td>102</td>
</tr>
<tr>
<td>1002</td>
<td>0.2</td>
<td>103</td>
<td>104</td>
</tr>
</tbody>
</table>
ELEM

Status - REQUIRED (if HOLE is input)
Full Keyword - ELEMENTS
Function - Signals the beginning of the connectivity definition for hole elements in the current surface.
Input Variables - NONE
Additional Information - NONE
Definition of Geometry

(NONE) NEL RADIUS NODE1 ... NODEN

Status - REQUIRED (if HOLE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the radius and connectivity for a single hole element.

Input Variables -

NEL (Integer) - REQUIRED
User hole element number.

RADIUS (Real) - REQUIRED
Radius of the hole.

NODE1 ... NODEN (Integer) - REQUIRED
User node numbers of the two or three hole nodes defining the geometry of the hole.

Additional Information -

This card is input once for each hole element.

The input card need not specify whether a two or three node element is being defined. This is determined strictly by the number of connectivity nodes included on the card.

User element numbers, including both surface and hole elements, must be unique within a problem.

All hole element numbers must be less than or equal to 9999.

All of the nodes referenced in the hole element connectivity must have been defined previously as POINTs under the HOLE keyword. That is, hole elements can only connect hole nodes. No surface nodes can be referenced by a hole element.

Two noded hole elements have a linear geometric representation, while three-noded elements use quadratic shape functions for their geometry.

The radius is assumed constant along the entire length of the hole element.

Examples of Use -

1. Define a set of data for two hole elements, specifying the element number, the radius and the connectivity. One hole element is three-noded with linear variation of field quantities and the other element is two-noded.
Definition of Geometry

<table>
<thead>
<tr>
<th>TYPE</th>
<th>ELEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1001 0.2 101 102 103</td>
</tr>
<tr>
<td></td>
<td>1002 0.2 103 104</td>
</tr>
</tbody>
</table>

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
5.3.7  INSERT ELEMENT DEFINITION

INSE   EVAL

Status - OPTIONAL

Full Keyword - INSERT

Function - This card initiates the definition of Insert inclusions within the current GMR and defines the Modulus of the Insert.

Input Variables -

   EVAL  (Alphanumeric)  - REQUIRED
   Elastic Modulus of the Insert.

Additional Information -

   The Poisson's ratio of the insert is assumed to be the same as that of the GMR.
   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.

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
   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)
Definition of Geometry

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.
Input Variables -
  NNODE (Integer) - REQUIRED
  User number for the insert nodes.
  X,Y,Z (Real) - REQUIRED Cartesian coordinates of the insert nodes.
Additional Information -
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 9999.
All insert nodes should be located on the interior of the part. Additionally, insert nodes should not be coincident with surface or sampling points.
Examples of Use -
1. Define a set of node numbers and the cartesian coordinates for nodes on insert elements.

INSERT 30.0E+06
POINNS 101 1.0 0.0 1.0
Definition of Geometry

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
TYPE  ETYYPE

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 -

ETYYPE (Alphanumeric) - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

The TYPE card defines the variation for three-noded insert elements only.

Only one TYPE card for inserts is allowed per GMR.

All two-noded insert elements require linear longitudinal variation of the field quantities.

All of the three-noded insert elements in a single GMR must have the same type of variation.

This card has no effect on the circumferential variation of the field variables on the 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 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.

Additional Information -
This card is input once for each insert element.
The input card need not specify whether a two or three node element is being defined. This is determined strictly by the number of connectivity nodes included on the card.
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 9999.
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.
Two noded insert elements have a linear geometric representation, while three-noded elements use quadratic shape functions for their geometry.
The radius is assumed constant along the entire length of the insert element.

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.
<table>
<thead>
<tr>
<th>POINTS</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>1.0</td>
<td>0.0</td>
<td>1.0</td>
</tr>
<tr>
<td>102</td>
<td>1.0</td>
<td>0.25</td>
<td>1.0</td>
</tr>
<tr>
<td>103</td>
<td>1.0</td>
<td>0.5</td>
<td>1.0</td>
</tr>
<tr>
<td>104</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>201</td>
<td>2.0</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>202</td>
<td>2.0</td>
<td>1.5</td>
<td>0.5</td>
</tr>
<tr>
<td>203</td>
<td>2.0</td>
<td>1.5</td>
<td>1.0</td>
</tr>
</tbody>
</table>

| TYPE QUAD ELEMENT | |
|---|---|---|---|---|
| 1001 | 0.2 | 101 | 102 | 103 |
| 1002 | 0.2 | 103 | 104 |

| ELEMENT | |
|---|---|---|---|
| 2001 | 0.2 | 201 | 202 | 203 |
Definition of Geometry

5.3.8 VOLUME CELL DEFINITION

VOLUME NAME

Status - OPTIONAL
Full Keyword - VOLUME
Function - This card initiates the definition of a volume for the current GMR.
Input Variables -
   NAME (Alphanumeric) - OPTIONAL
   The name of the volume being defined. (For user's use only)
Additional Information -
   In the present version of BEST3D, only one volume discretization per GMR is allowed. This means only one type (see next card definition) of cells, QUAD or LINE, can exist in a single GMR.
   Hot spots and holes may be embedded within cells. Volume cells are not required if a global shape function discretization is used. It is only required for GMRs with thermal body force loading or when nonlinear effects are present.

Examples of Use -
1. Define three, 8-Noded linear volume cells.

VOLUME
   TYPE LINEAR
   CELL
      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
   FULL
$(end of volume cell input)$
Definition of Geometry

TYPE TP

Status - REQUIRED (if VOLU is input)
Full Keyword - TYPE
Function - This card defines the variation of field quantities over the volume cells of the current GMR.

Input Variables -

TP (Alphanumeric) - REQUIRED
Allowable values are LINE and QUAD.

Additional Information -
A single GMR may contain either linear or quadratic cells, but not both.
If TP = QUAD is used and there are cells present, which are using linear geometry, then linear field variables will be assumed for those cells.

Examples of Use -
1. Specify that the variation of field quantities over the volume cells in GMR1 is quadratic in nature.

***GMR
ID GMR1
.
.
.
VOLUME
TYPE QUAD
CELL

Status - REQUIRED (if VOLU is input)
Full Keyword - CELLS
Function - Signals the beginning of the definition of volume cell input connectivity.
Input Variables - NONE
Additional Information -

Cell connectivity information is input on data cards following this card.

(NONE)    NCELL N1 N2 .... NK

Status - REQUIRED (If VOLU is input)
Full Keyword - NO KEYWORD REQUIRED
Function - Defines a volume cell in terms of previously defined nodal points.
Input Variables -

NCELL (Integer) - REQUIRED
User identification for cell being defined.
N1,N2,...,NK (Integer) - REQUIRED
User nodal point numbers for cell nodes.
NK = 4, 5, 6, 7, 8, 9, 10, 11, 15, 16, 20 or 21

Additional Information -

If necessary, this card may be input more than once for each cell. The cell number must be repeated on each card.

Cell numbering must begin at a corner and be numbered consecutively in either direction completing the face of a cell with the least number of nodes. The numbering of the remaining nodes begins at the same corner as the first face and continues in the same direction completing the mid-side nodes of the cell first (if present) and then the opposite face. If a central node is present, it is placed last.

Nodal points of the surface discretization may also be used in the volume discretization (ie. a cell face may match up with a boundary element). This is recommended when possible, since it somewhat reduces the computation required. Nodal points of the volume discretization lying on the surface need not, however, be nodal points of the surface mesh. In the event that such points do coincide, they can have different nodal point numbers in the surface and volume discretizations, although this should be avoided when possible.
Definition of Geometry

Examples of Use -

1. Define a set of volume cells consisting of a cell number and the connectivity information. There are three 8-noded volume cells with quadratic variation.

<table>
<thead>
<tr>
<th>VOLUME</th>
<th>TYPE</th>
<th>QUAD</th>
<th>CELL</th>
</tr>
</thead>
<tbody>
<tr>
<td>501</td>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>103</td>
<td>203</td>
</tr>
<tr>
<td>502</td>
<td>3</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>105</td>
<td>205</td>
</tr>
<tr>
<td>503</td>
<td>5</td>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td></td>
<td>107</td>
<td>207</td>
</tr>
</tbody>
</table>
Figure for **GM**: Volume Cell Connectivity card
Linear, three-dimensional volume cells
Definition of Geometry

Figure for **GMR**: Volume Cell Connectivity card
Quadratic, three-dimensional volume cells
FULL

Status - OPTIONAL
Full Keyword - FULL region of cells
Function - Identifies the GMR is completely filled with cells and that the Initial Stress Expansion Technique should be used to accurately calculate the coefficient corresponding to a singular point in the volume integration.

Input Variables - NONE
Additional Information -

The GMR must be completely filled by cells.
If FULL is not used, the relevant coefficients are calculated by volume integration complemented by the analytical jump term.
For highly accurate results, it is recommended that a GMR be completely filled with cells and that the FULL card option be exercised.

Examples of Use -
1. Specify that the GMR (GMR1) is completely filled with volume cells.

```
VOLUME
TYPE QUAD
CELL
  501 1 2 3 103 203 202 201 101
  502 3 4 5 105 205 204 203 103
  503 5 6 7 107 207 206 205 105
FULL
```
5.3.9 [GLOBAL SHAPE FUNCTION DEFINITION]

GLOB

Status - OPTIONAL
Full Keyword - GLOBAL-SHAPE-FUNCTION-REPRESENTATION
Function - Identifies a global shape function representation will be used in this GMR.
Input Variables - NONE
Additional Information -
   Not required if volume integration is used, and only required in GMR's with thermal load or which yielding occurs.

Examples of Use -
1. Define a 24-noded global shape function

GLOBAL SHAPE FUNCTION
NODE
  1 2 3 4 5 6 21 22 23 101 102 103
  104 105 201 202 203 301 302 303 401
  402 403 404
$ (end of global shape function input)
NODE

Status - OPTIONAL (If GLOB is input)
Full Keyword - NODE LIST
Function - Signals the beginning of a list of nodes that will be used for the global shape function.
Input Variables - NONE

Additional Information -

These nodes are the volume source points of the system. If a thermal gradient is present in the GMR, then a temperature data should be provided for these nodes in the Body Force Input Section. Otherwise these nodes are assumed to remain at the initial temperature of the GMR throughout the analysis.

A list of nodes to be used in the global shape function is input on data cards following the NODE card.

If NODE is not input all geometry points in the GMR will be used for the nodes of the global shape function. However, it is recommended that an explicit node list be input.

(NONE) N1 N2 ... NK

Status - REQUIRED (if NODE is input)
Full Keyword - NO KEYWORD REQUIRED
Function - Defines the node list for the global shape function.
Input Variables -

N1,N2,...,NK (Integer) - REQUIRED
Nodes of the global shape function. Maximum of 20 nodes per line.

Additional Information -

Points for sampling points can be located anywhere in the GMR and these points are automatically added to the node list and therefore need not be specified on the node list.

This card can be repeated as many times as necessary with a maximum of 20 nodes per line and a maximum of 150 nodes per GMR (including interior points)

The nodal points may be contained in the regular nodal point definition for the GMR or on the hole or insert point list. These points may lie on the boundary or be contained in the interior of the region. Points in the interior
Definition of Geometry

are recommended in order to produce an accurate representation of the interior distribution of the field variables.

No special ordering is necessary, however, ordering by proximity is preferred. Furthermore, a somewhat regular distribution of the global shape function nodes is recommended.

Sampling points need not be included in this list.

Examples of Use -

1. Specify a list of 24 nodes for a global shape function.

GLOBAL SHAPE FUNCTION

NODE

1  2  3  4  5  6  21  22  23  101  102  103
104  105  201  202  203  301  302  303  401  402
403  404
SAMP

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 - NONE

ITYP1 (Alphanumeric) - REQUIRED

Additional Information -
This card is used to define points at which displacements, stresses, strains, temperatures, pressures and fluxes are to be calculated. The print flag for sampling points must be set in **CASE input. If, however, nothing is specified in **CASE for sampling points, this flag is set by default.

This card is followed by data cards defining the node number and coordinates of the sampling points.

Examples of Use -

1. Request result information at three interior points

<table>
<thead>
<tr>
<th>SAMPLING-POINTS</th>
<th>1001</th>
<th>0.333</th>
<th>0.25</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1002</td>
<td>0.25</td>
<td>0.1</td>
<td>10.0</td>
</tr>
<tr>
<td></td>
<td>1003</td>
<td>0.2</td>
<td>0.5</td>
<td>20.2</td>
</tr>
</tbody>
</table>
**Definition of Geometry**

**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 volume discretization, or discretization of Holes and Inserts.

Point numbers must be less than or equal to 9999.
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 BEST3D compatibility is defined between the interface surfaces of each pair of contacting regions. Four 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 - Spring or resistance contact: Spring or thermal resistance is imposed between regions.

4 - 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 either case.

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 figure for **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.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.4.1</td>
<td>Interface Definition Input Card **INTE</td>
<td>Start of interface compatibility condition</td>
</tr>
<tr>
<td>5.4.2</td>
<td>Definition of interface surface 1 GMR SURF ELEM</td>
<td>name of first GMR surface on first GMR element of surface</td>
</tr>
</tbody>
</table>

BEST3D User Manual April, 1990 Page 5.122
Definition of GMR Compatibility

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.4.3</td>
<td>Definition of interface surface 2</td>
</tr>
<tr>
<td></td>
<td>GMR</td>
</tr>
<tr>
<td></td>
<td>SURF</td>
</tr>
<tr>
<td></td>
<td>ELEM</td>
</tr>
</tbody>
</table>

| 5.4.4   | Type of interface conditions |
|         | BOND    | bonded interface |
|         | SLID    | sliding interface |
|         | SPRIR   | spring interface |
|         | RESI    | thermal resistance across interface |

| 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 |
**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
```
Definition of GMR Compatibility

Acceptable Connections

Unacceptable Connection

Figure for **INIE; card
Connections among GMRs
Definition of GMR Compatibility

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 GMRI
SURFACE SURF1
ELEMENTS 101 102 103 104
Definition of GMR Compatibility

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 BEST3D 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, BEST3D will apply the interface compatibility condition only to the elements specified on this list.

In the present version of BEST3D, 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.

```plaintext
**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
Definition of GMR Compatibility

**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 **BEST3D** 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** **GHRI**  
**SURFACE** **SURF1**  
**ELEMENTS** 101  102  
**GMR** **GHRI**  
**SURFACE** **SURF2**  
**ELEMENTS** 201  202  
```
Definition of GMR Compatibility

**INTERFACE**
GMR GMR1
SURFACE SURF1
ELEMENTS 101 102 103
GMR GMR2
SURFACE SURF2
ELEMENTS 210 211 212

BEST3D User Manual
April, 1990
Page 5.131
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
Definition of GMR Compatibility

**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. 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
```
**Definition of GMR Compatibility**

**SPRI**  IDIR  KN

Status - OPTIONAL

Full Keyword - SPRING

Function - Identifies an interface with spring resistance in the normal direction between the corresponding surfaces. The normal tractions across this interface are linearly related to the difference in normal displacements between the two surfaces.

Input Variables -
- **IDIR** (Integer) - REQUIRED
  direction must be normal to the element (IDIR = 1)
- **KN** (Real) - REQUIRED
  Spring coefficient ($K_n$) normal to the element

Additional Information -
The SPRIng option utilizes the relationship $t_n = K_n (d_n^2 - d_n')$ where
- $d_n'$ normal displacement of GMR 1.
- $d_n''$ normal displacement of GMR 2.
- $t_n$ normal traction of GMR 1.

In the present version of the code a spring can only be defined in the direction normal to the element (i.e. IDIR = 1 only). In the direction tangential to the element a bonded relation is assumed unless the SLID card is also included.

The user is responsible for providing $K_n$ in the proper units, consistent with the specifications of material properties, geometry, and boundary conditions.

The spring coefficient $K_n$ should be a positive real number ($K_n > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -
1. Defines a normal spring with a spring constant of 1.0E+03 units at the interface of two elements which are free to slide in the tangential direction.

```
**INTERFACE
GMR REG1
  SURFACE TOP
  ELEMENT 101
GMR REG4
  SURFACE BOTTOM
  ELEMENT 401
SPRING 1 1.0E+03
SLIDING
```
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 (FP) - REQUIRED
  Thermal resistance coefficient (R)

Additional Information -
The RESIstance option utilizes the relationship:

\[ q = \frac{1}{R} (\theta_1 - \theta_2) \]

where
\[ \theta_1 \] local temperature of GMR 1.
\[ \theta_2 \] local temperature of GMR 2.
\[ q \] 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.

Examples of Use -
1. Defines thermal resistance at the interface between two regions which were at the same initial temperature (otherwise, an extra card TDIF should be inserted between the RESI and **BCSET cards).
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.

CYCLE-A - 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.

CYCLE-B - 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 (or velocity) boundary conditions not be applied to any of the second surface nodes.

In the present version of BEST3D, 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.
Definition of GMR Compatibility

**INTERFACE
  GMR GMRI
  SURFACE SURF1
  ELEMENT 3
  GMR GMRI
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION 0. 0. 1.

**BCSET
**INTERFACE**
GMR GMR1
SURFACE SURF1
ELEMENT 3
GMR GMR1
SURFACE SURF1
ELEMENT 5
CYCLIC
ANGLE 20
DIRECTION 0. 0. 1.
**BCSET**

Definition of GMR Compatibility

**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.
Definition of GMR Compatibility

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.
**BCSET
Definition of GMR Compatibility

### 5.4.6 ADDITIONAL INTERFACE CONTROL OPTIONS

**TDIF**

Status - OPTIONAL

Full Keyword - TDIFFERENCE

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 BEST3D, 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
REST 1.0
TDIF
**BCSET
```
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 BEST3D 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 5.5.8.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.5.1</td>
<td>Boundary Condition Set Card</td>
<td>start of the B.C. definition</td>
</tr>
<tr>
<td></td>
<td><strong>BCSET</strong></td>
<td></td>
</tr>
<tr>
<td>5.5.2</td>
<td>Boundary Condition Identification</td>
<td>name of B.C. set</td>
</tr>
<tr>
<td></td>
<td>ID</td>
<td></td>
</tr>
<tr>
<td>5.5.3</td>
<td>Identification of Boundary Condition Type</td>
<td>for specified B.C. value input</td>
</tr>
<tr>
<td></td>
<td>VALU</td>
<td></td>
</tr>
<tr>
<td></td>
<td>RELA</td>
<td>for B.C. relation between boundary quantities</td>
</tr>
<tr>
<td></td>
<td>LOCA</td>
<td>for local definition of B.C.</td>
</tr>
</tbody>
</table>
### Definition of Boundary Conditions

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.5.4</td>
<td><strong>GMR</strong></td>
<td>identifies a GMR</td>
</tr>
<tr>
<td></td>
<td><strong>SURF</strong></td>
<td>identifies the surface for this B.C. set</td>
</tr>
<tr>
<td></td>
<td><strong>ELEM</strong></td>
<td>identifies surface elements</td>
</tr>
<tr>
<td></td>
<td><strong>POIN</strong></td>
<td>identifies surface points</td>
</tr>
<tr>
<td></td>
<td><strong>TIME</strong></td>
<td>defines the time for input</td>
</tr>
</tbody>
</table>

| 5.5.5   | **TRAC** | traction B.C. input |
|         | **DISP** | displacement B.C. input |
|         | **FLUX** | flux B.C. input |
|         | **TEMP** | temp B.C. input |

| 5.5.6   | **GMR** | identifies a GMR |
|         | **HOLES** | indicates B.C. on holes |
|         | **ELEM** | identifies the hole elements |
|         | **POIN** | identifies the hole points |
|         | **TIME** | defines the time for input |

| 5.5.7   | **FLUX** | flux B.C. input |
|         | **TEMP** | temp B.C. input |
|         | **PRES** | pressure B.C. input in hole |

| 5.5.8   | **SPLI** | source (field variable) point list |
|         | **T**    | nodal value of B.C. |

| 5.5.9   | **SPRI** | spring relation (between displacement and traction) |
|         | **CONV** | convection relation (between temperature and flux) |
**BCSET**

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, and must precede any BODYFORCE data.

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)

```plaintext
**BCSET
ID U1FIX
VALUE
LOCAL
GMR REG2
   SURFACE SIDE1
   DISP 1
   SPLIST ALL
   T 1 0.0
```
Definition of Boundary Conditions

5.5.2 BOUNDARY CONDITION IDENTIFICATION

<table>
<thead>
<tr>
<th>ID</th>
<th>NAME</th>
</tr>
</thead>
</table>

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
**Definition of Boundary Conditions**

### 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 neither VALU nor RELA appears 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
```
Definition of Boundary Conditions

**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
GHR GHR1
SURFACE SUItF1
  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 BEST3D 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).
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 BEST3D 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.
**Definition of Boundary Conditions**

**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.

```plaintext
**BCSET
    ID TRAC1
    VALUE
    GMR1 REG1
    SURFACE SURF1
    ELEMENTS 101 102
    TRAC 1
    SPLIST ALL
    T 1 100.0
$ end of input data
```
**Definition of Boundary Conditions**

**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 BEST3D 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.

```plaintext
**BCSET  
 ID  TRAC1  
 GMR  GMR1  
 SURFACE SURF1  
 ELEMENTS 101 102 103  
 TRAC 1  
 SPLIST ALL  
 T 1 100.0  
**BCSET
```
Definition of Boundary Conditions

Result

Boundary condition applied at nodes 1 through 8.
Over elements 101 through 105.

Boundary condition applied at nodes 3, 4, 5 and 6.
Over element 101 only.

Boundary condition applied at nodes 3, 4, 5 and 6.
Over elements 102, 103 and 105.

Boundary condition applied at node 3.
Over element 101 only.

Figure for **BCSET: SURF, ELEM and POIN cards
3-D boundary subset definition
Definition of Boundary Conditions

\textbf{ELEM} \textbf{EL1 EL2 ... ELN}

\textbf{Status} - \textit{OPTIONAL}

\textbf{Full Keyword} - \textit{ELEMENTS}

\textbf{Function} - Specifies the elements of the surface IDSUR to which a boundary condition is to be applied.

\textbf{Input Variables} -
\begin{itemize}
  \item \textbf{EL1,EL2,...,ELN (Integer)} - \textit{REQUIRED}
  \item User element numbers of the elements of surface IDSUR which are to be included within the boundary condition set.
\end{itemize}

\textbf{Additional Information} -
\begin{itemize}
  \item The effect of this card is to restrict the application of the boundary condition to a portion of the surface IDSUR.
  \item This input may be continued on more than one card. Each card must begin with the keyword ELEM.
  \item If the ELEM card is not followed by a POIN card, then BEST3D will apply the boundary condition to all of the source points in the specified elements.
\end{itemize}

\textbf{Examples of Use} -
\begin{itemize}
  \item Specifies three elements on the surface SURF1 on which traction boundary conditions are given.
\end{itemize}

\begin{verbatim}
**BCSET
  ID DISP2
  SURFACE SURF1
  ELEMENTS 101 102 103
  DISP 2
  SPLIST ALL
  T 1 0.0
\end{verbatim}
**BCSET**  
**ID** BC1  
**VALUE**  
GER. 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

BEST3D User Manual  
April, 1990  
Page 5.153
**Definition of Boundary Conditions**

**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 specified 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 5.5.8).**
Definition of Boundary Conditions

Example:

```
**BCSET
ID TRAC1
VALUE
GMR GMR1
SURF SURF1
ELEMENT 3
TIME 1.0 2.0 3.0 3.5 4.0 5.0
TRAC 1
SPLIT 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
Definition of Boundary Conditions

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.
Definition of Boundary Conditions

**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, y and z directions are zero) of three elements.
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
DISP 3
SPLIST ALL
T 3 0.0
**BCSET
ID TRAC1
Definition of Boundary Conditions

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 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
Definition of Boundary Conditions

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 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
5.5.6 Definition of Boundary Conditions

### DEFINITION OF HOLES FOR APPLICATION OF BOUNDARY CONDITIONS

**GMR IDGMR**

- **Status** - REQUIRED
- **Full Keyword** - GMR
- **Function** - Identifies the GMR on the surface of which the boundary condition is to be defined.
- **Input Variables** -
  
<table>
<thead>
<tr>
<th>IDGMR</th>
<th>(Alphanumeric)</th>
<th>REQUIRED</th>
</tr>
</thead>
<tbody>
<tr>
<td>IDGMR</td>
<td>(Alphanumeric)</td>
<td>REQUIRED</td>
</tr>
</tbody>
</table>

  **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. Define a linear increasing pressure inside a hole (elements 203, 204) for stress analysis.

```plaintext
**BCSET
GMR REG1
HOLE
ELEMENT 203 204
TIME 0.0 10.0
PRESSURE
SPLIST ALL
T 1 0.0
T 2 100.0
```
HOLE

Status - OPTIONAL
Full Keyword - HOLE
Function - Identifies this as a boundary condition set for hole elements within the selected GMR.
Input Variables - NONE
Additional Information -

Either this keyword or the SURF keyword must be input for each boundary condition set.

If the HOLE keyword line is not followed by an ELEM line, then BEST3D will apply the boundary condition to all of the hole elements in the current GMR.

Examples of Use -

1. Heat transfer – Define a temperature (for all time) at points in elements 509, 510.

```plaintext
**BCSET
GMR GMR5
HOLE
ELEMENT 509
POINTS 5025 5026 5027
PRESSURE
SPLIST 5025 5025 5027
T 1 100.0 150.0 200.0
```
Definition of Boundary Conditions

ELEM  EL1 EL2 ... ELN

Status - OPTIONAL
Full Keyword - ELEMENTS
Function - Specifies the hole elements to which a boundary condition is to be applied.
Input Variables -
   EL1,EL2,...,ELN (Integer) - REQUIRED
   User element numbers of the hole elements 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 group of hole elements within the previously specified GMR.
   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, the BEST3D will apply the boundary condition to all of the source points in the specified hole element.

Examples of Use -
   1. Define a flux of 100 for points in elements 207, 208, 209 and 311 (for all time).

   **BCSET
      GMR REG5
      HOLE
      ELEMENT 207 208 209 311
      FLUX
      SPLIST ALL
      T 1 100.0
Definition of Boundary Conditions

POIN P1 P2 ... PN

Status - OPTIONAL
Full Keyword - POINTS
Function - Restricts the application of a boundary condition to a subset of the source points associated with hole elements specified above.

Input Variables -
   P1,P2,...,PN (Integer) - REQUIRED

Additional Information -
This card restricts the application of the boundary condition to the nodal hole 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 BEST3D will apply the boundary condition to all of the hole points specified in this list. Also, if the POIN card is included, then the SPLIST card (see section 5.5.8) must explicitly list the same set of hole nodes.

Examples of Use -

1. Define a linear increasing pressure (in time) that varies from point to point in an element.

   **BCSET
   GNk REG9
   HOILE
   ELEMENT 207
   POINTS 2005 2007 2011
   TIME 0.0 1.0
   PRESSURE
   SPLIST 2005 2007 2011
   T 1 0.0 0.0 0.0
   T 2 10.0 15.0 20.0

BEST3D User Manual April, 1990 Page 5.165
Definition of Boundary Conditions

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 specified 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 5.5.8).

Examples of Use -

1. Define a time-dependent (4 time points) temperature for two points in element 907.

**BCSET
GMRF REGIO
HOLE
ELEMENT 907
POINTS 9003 9004
TIME 0.0 5.0 10.0 20.0
TEMPERATURE
SPLIST 9003 9004
T 1 0.0 0.0
T 2 10.0 20.0
T 3 20.0 50.0
T 4 30.0 60.0
PRES

Status - OPTIONAL
Full Keyword - PRESSURE
Function - Indicates that the pressure will be specified for all nodes of the current boundary condition set.
Input Variables - NONE
Additional Information -
This card can only be used in a boundary condition data set containing the VALU card.
When applicable, the default condition is to set pressure to zero.
The PRESSURE input line must be immediately followed by the space/time variation.

Examples of Use -
1. Stress Analysis - Define a time dependent pressure inside a hole (element 101).

**BCSET
  GMR  GMR5
  HOLE
  ELEMENT 101
  TIME 0.0 5.0 10.0 2.0
  PRESSURE
  SPLIST ALL
  T 1 0.0
  T 2 50.0
  T 3 500.0
  T 4 1000.0
Definition of Boundary Conditions

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 analysis.
   This card can only be used in a boundary condition data set containing the VALU card.
   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. Define a flux (for all times) at points in element 509.

**BCSET
   GMR REG3
   HOLE
   ELEMENT 509
   POINT 5003 5004
   FLUX
   SPLIST 5003 5004
   T 1 5.0 6.0
Definition of Boundary Conditions

**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 analysis.

This card can only be used in a boundary condition data set containing the VALU card.

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. Heat Transfer - Define a time-dependent temperature inside a hole.

    **BCSET**
    GMR EXCHANGER
    HOLE
    ELEMENT 201
    TIME 0.0 5.0 10.0 3.0
    TEMPERATURE
    SPLIST
    T 1 0.0
    T 2 20.0
    T 3 50.0
    T 4 100.0
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 BEST3D 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 definition of surface for application of Boundary Conditions (section 5.5.4).
Definition of Boundary Conditions

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 card defining the values at time point IT must begin with T. The input for each new time point must begin on a new card.

If N1 = ALL or SAME on the SPLI card, then only a single value of the variable is input for each time point.

If the card "LOAD COMP" is specified in the case input, i.e. the boundary conditions are complex values, then the real part and imaginary part of nodal values are input in the form V1R V1I V2R V2I ...... VNR VNI. More cards can be added, if one card is not enough to specify the needed boundary conditions.

The results of various uses of the SPLI and T cards are shown in the figure on the following page.
Definition of Boundary Conditions

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

Figure for BCSET: TIME and SPLIST cards
TIME-SPACE variation
Definition of Boundary Conditions

5.5.9 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

Spring constant (for steady-state forced vibration problems, it should consist of both real and imaginary parts of the spring constant).

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 BEST3D, 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 z 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 (T₀)

Additional Information -

- CONV is only valid for HEAT 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:

\[ Q = -H \cdot (T_a - T) \]

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.
Definition of Boundary Conditions

RELATION
GMR GMR1
SURFACE SURF1
ELEMENTS 1 2 3 4
CONV 1.26 -100.0
**BCSET
ID BCS2
This section describes the input for body forces.

The following body forces are included in BEST3D: thermal, hot spot, centrifugal and inertial loads. The input cards required to define these loads are described below.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.6.1</td>
<td><strong>BODY</strong></td>
<td>start of body force input</td>
</tr>
<tr>
<td>5.6.2</td>
<td>CENT</td>
<td>centrifugal load input</td>
</tr>
<tr>
<td></td>
<td>DIRE</td>
<td>direction of axis of rotation</td>
</tr>
<tr>
<td></td>
<td>POINT</td>
<td>point on the axis of rotation</td>
</tr>
<tr>
<td></td>
<td>TIME</td>
<td>time for input</td>
</tr>
<tr>
<td></td>
<td>SPEE</td>
<td>speed input</td>
</tr>
<tr>
<td>5.6.3</td>
<td>INER</td>
<td>inertial body force input</td>
</tr>
<tr>
<td></td>
<td>DIRE</td>
<td>direction of acceleration</td>
</tr>
<tr>
<td></td>
<td>TIME</td>
<td>time of input</td>
</tr>
<tr>
<td></td>
<td>ACCE</td>
<td>acceleration input</td>
</tr>
<tr>
<td>5.6.4</td>
<td>THER</td>
<td>thermal body force input</td>
</tr>
<tr>
<td></td>
<td>TIME</td>
<td>times for input</td>
</tr>
<tr>
<td></td>
<td>GMR</td>
<td>Identifies GMR</td>
</tr>
<tr>
<td></td>
<td>TEMP</td>
<td>start of temperature input for GMR</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(node temperatures)</td>
</tr>
<tr>
<td>5.6.5</td>
<td>HOT</td>
<td>hot spot input</td>
</tr>
<tr>
<td></td>
<td>TIME</td>
<td>time for input</td>
</tr>
<tr>
<td></td>
<td>GMR</td>
<td>Identifies GMR</td>
</tr>
<tr>
<td></td>
<td>TEMP</td>
<td>start of hot spot temperature input for GMR</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(node temperatures)</td>
</tr>
</tbody>
</table>
5.6.1 BODY FORCE INPUT CARD

**BODY

Status - OPTIONAL

Full Keyword - BODY FORCE

Function - Identifies the beginning of body force input.

Input Variables - NONE

Additional Information -

If more than type of body force is present, a separate block starting with **BODY should be defined for each type.

Examples of Use -

1. Request a three-dimensional centrifugal and thermal input.

**BODY FORCE
CENT
DIRE 0.0 0.0 1.0
POINT 0.0 0.0 0.0
TIME 1. 2. 3. 4.
SPEED 45. 80. 100. 120.

**BODY FORCE
THER
TIME 0. 5.
GMR REG1
TEMP
1 0.0 500.
2 0.0 500.
3 0.0 300.
Example: **BODY FORCE
CENTRIFUGAL
POINT 0.0 0.0
TIME 1.0 2.0 3.0 3.5 4.0 5.0
SPEED 100.0 200.0 200.0 100.0 100.0 300.0

Figure for **BODY: (all body force values)
VALUE vs. TIME for body force input
CENT

Status - OPTIONAL
Full Keyword - CENTRIFUGAL
Function - Indicates that a centrifugal load will be applied.
Input Variables - NONE
Additional Information -

Only one (time dependent) centrifugal load condition may be defined for an analysis. It is applied to the entire part.

Examples of Use -

1. Request a three-dimensional centrifugal input.

```
**BODY FORCE
CENT
DIRE  0.0  0.0  1.0
POINT 0.0  0.0  0.0
TIME  1.0  2.0  3.0
SPEED  50.0  150.0  200.0
```
DIRE  X Y Z

Status - OPTIONAL
Full Keyword - DIRECTION
Function - Defines a vector parallel to the axis of rotation.
Input Variables -
  X,Y,Z  (Real) - REQUIRED
  Cartesian components of a vector parallel to the axis of rotation.
Additional Information -
  Only one direction can be defined in an analysis
  If this card is omitted, the axis of rotation is assumed to be parallel to the z-axis of the global system.

Examples of Use -

1. Request a three-dimensional centrifugal input with the direction of rotation parallel to the X-axis.

**BODY FORCE
CENT
DIRE  1.0  0.0  0.0
POINT  0.0  0.0  0.0
TIME   1.0
SPEED  100.
**POIN** XP YP ZP

Status - OPTIONAL

Full Keyword - POINT

Function - Defines a reference point on the axis of rotation.

Input Variables -

XP,YP,ZP (Real) - REQUIRED

Cartesian coordinates of a point on the axis of rotation.

Additional Information -

If this card is omitted, the reference point is taken to be the origin (0,0,0) in the global system.

Examples of Use -

1. Request a three-dimensional centrifugal input for rotation about an axis centered between the points (1,1,0) and (5,5,1)

```plaintext
**BODY FORCE
CENT
DIRE 4. 4. 1.
POINT 1. 1. 0.
TIME 1.0
SPEED 50.0
```
TIME  T1 T2 ... TN

Status - REQUIRED (if CENT is input)

Full Keyword - TIMES

Function - Defines the times at which the speed of rotation of the part will be defined.

Input Variables -
  T1,T2,...,TN (Real) - REQUIRED
  Times at which speed of rotation will be defined.

Additional Information -
  This card may be input as many times as required. Each card begins with the keyword TIME.
  A maximum of 20 time values may be specified.

Examples of Use -
  1. Request an input for three-dimensional centrifugal input at 5 times.

**BODY FORCE
  CENT
  DIRE  0.0 0.0 1.0
  POINT  0.0 0.0 0.0
  TIME  0.0 2.0 4.0 6.0 10.0
  SPEED  0.0 400. 500. 400. 0.
SPEE  OMEGA1 OMEGA2 .... OMEGAN

Status - REQUIRED
Full Keyword - SPEED
Function - Defines the speed of rotation of the part.
Input Variables -
  OMEGA1, OMEGA2, ..., OMEGAN (Real) - REQUIRED
  Speed of rotation (RPM) at times specified on TIME card.
Additional Information -
  This card may be input as often as required. Each card begins with the keyword SPEE.

Examples of Use -

1. Request an input for three-dimensional centrifugal case

   **BODY FORCE
   CENT
   DIRE 0.0 0.0 1.0
   POINT 0.0 0.0 0.0
   TIME 1.0 2.0 3.0
   SPEED 100. 200. 400.

   ...
5.6.3  INERTIAL BODY FORCE

INER

Status - OPTIONAL
Full Keyword - INERTIA FORCE
Function - Indicates that an inertia force will be applied.
Input Variables - NONE
Additional Information -
Only one (time dependent) inertia load condition may be defined for an analysis.
It is applied to the entire body.

Examples of Use -
1. Request inertial input

**BODY FORCE
INER
DIRE 1.0 0.0 0.0
TIME 1.0
ACCE 300.
**DIRE**  X Y Z

Status - OPTIONAL

Full Keyword - DIRECTION

Function - Defines a vector parallel to the direction of inertia force.

Input Variables -

X,Y,Z (Real) - REQUIRED

Cartesian components of a vector parallel to the inertia force.

Additional Information -

Only one direction can be defined in an analysis.

If this card is omitted, the inertia force is assumed to be parallel to the z-axis of the global system in the negative direction (i.e. gravity loading).

Examples of Use -

1. Defines an inertial force in the positive Z-direction for a three-dimensional analysis.

```plaintext
**BODY FORCE
INER
DIRE 0. 0. 1.
TIME 1.0
ACCE 10.0
```
TIME  T1  T2  ...  TN

Status - REQUIRED (if INER is input)

Full Keyword - TIMES

Function - Defines the times at which the acceleration of the body will be defined.

Input Variables -

T1, T2, ..., TN (Real) - REQUIRED

Times at which acceleration will be defined.

Additional Information -

This card may be input as many times as required. Each card begins with the keyword TIME.

A maximum of 20 time values may be specified.

Examples of Use -

1. Specifies accelerations at three times.

   **BODY FORCE
   INER
   DIRE  0.  0.  1.
   TIME  1.0  2.0  3.0
   ACCE  10.0 15.0 20.0
ACCE  ACC1 ACC2 .... ACCN

Status - OPTIONAL

Full Keyword - ACCELERATION

Function - Defines the acceleration of the body.

Input Variables -

   ACC1,ACC2,...,ACCN (Real) - REQUIRED
   Acceleration at times specified on TIME card.

Additional Information -

   This card may be input as often as required. Each card begins with the keyword ACCE.

   DEFAULT: Gravity loading of 386.4 in/sec/sec.

Examples of Use -

   1. Specifies an acceleration of 100.0 units in an inertial body force loading at a single time step.

   **BODY FORCE
      IBER
      DIRE 0.  1.
      TIME 1.
      ACCE 100.
5.6.4 THERMAL BODY FORCE

THER

Status - OPTIONAL
Full Keyword - THERMAL
Function - Indicates the start of temperature input for thermal body force.
Input Variables - NONE
Additional Information -

All thermal input for an analysis should be input in a single **BODY FORCE data block, even though more than one GMR may be involved.

This flag is not relevant to heat transfer analysis.

Examples of Use -

1. Thermal input for a two region problem at three time points.

```plaintext
**BODY FORCE
THERM
TIME 2.0 4.0 6.0
GMR REG1
TEMP
  1 100.0 200.0 500.0
  6 100.0 200.0 550.0
  5 100.0 200.0 625.0
  .  .  .  .
  27 100.0 200.0 475.0
GMR REG2
TEMP
  40 100.0 300.0 500.0
  41 100.0 300.0 475.0
  .  .  .  .
```
TIME   T1 T2 .... TN

Status - REQUIRED (if THER is input)
Full Keyword - TIMES
Function - Defines the times at which the nodal point temperatures will be defined.
Input Variables -
   T1,T2,...,TN (Real) - REQUIRED
   Times at which nodal point temperatures will be defined.
Additional Information -
   If all times do not fit on one card, TIME may be continued on a second card immediately following the first time card, starting with the keyword TIME. Only one time definition is allowed for thermal input, and therefore temperatures in each GMR should be defined according to this one definition.
   A maximum of 20 time values may be specified.
Examples of Use -
   1. Specifies three times at which temperatures are input.

**BODY FORCE
**THERM
**TIME  1.0  2.0  3.0
**GMR   GMR1
**TEMP
  1    100.0  200.0  400.0
  2    150.0  300.0  600.0
  3    200.0  400.0  800.0
GMR IDGMR

Status - REQUIRED (if THER is input)

Full Keyword - GMR

Function - Identifies the GMR in which temperatures will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometric definition (NAME on ID card in **GMR input.)

Additional Information -

The GMR card is repeated for each GMR. All GMR's (for which thermal input is desired) should be contained under one **BODY FORCE input.

Examples of Use -

1. Identifies GMR name at which temperatures are to be specified.

**BODY FORCE
  THERM
  TIME 1.0
  GMR REG1
TEMP

Status - REQUIRED (if THER is input)

Full Keyword - TEMPERATURES

Function - Signals the beginning of input for this GMR of temperatures at nodal points as a function of time.

Input Variables - None.

Additional Information - None.

Examples of Use -

1. Defines temperature distribution at previously defined nodes.

```
**BODY FORCE
 THERM
 TIME 1.0
 GMR GMR1
 TEMP
 101 1.0
 103 2.0
 105 25.0
 107 30.0
 109 35.0
```
(NONE)  NNODE TEM1 TEM2 ... TEMN

Status - REQUIRED (if THER is input)
Full Keyword - NO KEYWORD REQUIRED
Function - Defines nodal point temperatures at times specified on TIME card.
Input Variables -
  NNODE (Integer) - REQUIRED
  User nodal point number for which temperatures are being specified on this card.
  TEM1,TEM2,...,TEMN (Real) - REQUIRED
  The temperatures at node NNODE at the times specified on the TIME card.
Additional Information -
  This card is input as many times as required for each nodal point. Each new card begins with the nodal point number.
  Temperature input for each nodal point must begin on a new card.

Examples of Use -
  1. Defines discrete temperature distribution at previously defined nodes.

**BODY FORCE
THERM
TIME 1.0
GMR GMR1
TEMP
  101 10.0
  103 20.0
  105 25.0
  107 30.0
  109 35.0
5.6.5 EMBEDDED HOT SPOTS

HOT

Status - OPTIONAL

Full Keyword - HOT SPOT

Function - Signals that input will be provided defining the temperature history at buried hot spots.

Input Variables - NONE

Additional Information -

The times at which hot spot temperatures will be specified and the temperature histories are defined using TIME and TEMP cards.

Examples of Use -

1. Hot spot input for two regions with two hot spots per region (two time points).

   **BODY FORCE
   HOT SPOT
   TIME 1.0 10.0
   GM R REG1
   TEMP
   1001 0.1 100.0 1000.0
   2001 0.5 100.0 1500.0
   GM R REG2
   TEMP
   4005 0.5 100.0 3000.0
   4006 1.2 100.0 1500.0
**TIME** T1 T2 .... TN

Status - REQUIRED (if HOT is input)

Full Keyword - TIMES

Function - Defines the times at which hot spot temperatures will be defined.

Input Variables -

\[ T1, T2, ..., TN \] (Real) - REQUIRED

Times at which hot spot temperatures will be defined.

Additional Information -

If all times do not fit on one card, Times may be continued on a second card immediately following the first time card, starting with the keyword TIME. Only one time definition is allowed for thermal input, and therefore temperatures in each GMR should be defined according to this one definition.

A maximum of 20 time values may be specified.

Examples of Use -

1. Defines time 1.0 and 3.0 at which temperatures are input.

```
**BODY FORCE**
HOT SPOT
TIME 1.0 3.0
GMR GMR1
TEMP
  1001  0.2  100.0 1000.0
  2001  0.4  100.0 1500.0
GMR GMR2
TEMP
  4005  0.1  100.0 3000.0
  4006  0.3  100.0 1500.0
```
**GMR**

**IDGMR**

Status - REQUIRED (if HOT is input)

Full Keyword - GMR

Function - Identifies the GMR in which temperatures will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometric definition (NAME on ID card in **GMR input.)

Additional Information -

The GMR card is repeated for each GMR. All GMR's (for which thermal input is desired) should be contained under one **BODY FORCE input.

Examples of Use -

1. Defines GMR at which temperatures are input.

```plaintext
**BODY FORCE
HOT SPOT
TIME 1.0
GMR REG1
TEMP
101 0.1 150.0
105 0.1 120.0
108 0.2 135.0
111 0.3 142.0
```
**TEMP**

Status - REQUIRED (if HOT is input)

Full Keyword - TEMPERATURES

Function - Signals the beginning of input of temperatures at hot spots as a function of time.

Input Variables - NONE

Additional Information -

This card is followed by data cards defining hot spot temperatures. Each hot spot is characterized by its centroid and volume.

**(NONE)** NNODE VOL TEM1 TEM2 ... TEMN

Status - REQUIRED (if HOT is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines hot spot temperatures at times specified on TIME card.

Input Variables -

- **NNODE** (Integer) - REQUIRED
  User nodal point number of the hot spot centroid.

- **VOL** (Real) - REQUIRED
  Volume of hot spot.

- **TEM1,TEM2,...,TEMN** (Real) - REQUIRED
  The temperatures at node NNODE at the times specified on the TIME card.

Additional Information -

This card is input as many times as required for each hot spot. Each new card begins with the nodal point number.

Temperature input for each hot spot must begin on a new card.

A maximum of 10 hot spots are permitted per GMR.

Examples of Use -

1. Defines the list of temperature entries.
<table>
<thead>
<tr>
<th>GMR</th>
<th>GMR1</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>0.1</td>
<td>150.0</td>
</tr>
<tr>
<td>105</td>
<td>0.1</td>
<td>120.0</td>
</tr>
<tr>
<td>108</td>
<td>0.2</td>
<td>135.0</td>
</tr>
<tr>
<td>111</td>
<td>0.3</td>
<td>142.0</td>
</tr>
</tbody>
</table>
BEST3D allows for a change of an essential boundary condition to a natural boundary condition and vice-versa during a single analysis. This means that a boundary condition for a component of a particular node may be specified at the outset of the analysis, and then be changed to an unknown boundary condition during the course of the run. Similarly an unknown boundary condition may be changed during an analysis to a specified boundary condition and then changed back again to an unknown, as many times as necessary. This is useful, for instance, in a plastic metal forming analysis when a body is loaded in one direction and then changed to a loading in the other direction. Similarly, in a heat transfer analysis, a body can be initially insulated on a side and then changed later in the run to allow a temperature flow across that same face.

Essentially, the first definition of the boundary conditions for the initial time, is input in the usual manner. This first definition may be time dependent and may be used for any number of solution times defined in the case input. An entire new definition of boundary conditions (with changes in the specified knowns and unknowns) are input, preceded by the **BCCH and TIME card, after the initial definition.

Included on the TIME card is the time at which the changeover takes place. This next set likewise be time-dependent. This procedure may be repeated as many times as necessary.

<table>
<thead>
<tr>
<th>SECTION</th>
<th>KEYWORD</th>
<th>PURPOSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.7.1</td>
<td>Changes in Boundary Condition Knowns and Unknowns</td>
<td>**BCCH start of the new B.C. definition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TIME defines the time the change will occur</td>
</tr>
</tbody>
</table>

BEST3D User Manual
April, 1990
Page 5.199
5.7.1  BOUNDARY CONDITION CHANGE CARD

**BCCH

Status - OPTIONAL (Not available for all analysis types)

Full Keyword - **BCCHANGE

Function - Identifies an entirely new boundary condition definition in which the specified known and unknown boundary conditions will differ from the previously specified set.

Input Variables - NONE

Additional Information -

A TIME card must immediately follow the **BCCH card.

New boundary conditions for the entire body must be input corresponding to each **BCCH card. Even if certain **BCSETS do not change during the entire analysis, they must be repeated (redefined) for each **BCCH card.

All boundary condition sets preceding the **BCCH card are forgotten once the time of solution has passed over to the next set of boundary conditions after the **BCCH card.

Examples of Use -

1. Analysis with three unique sets of known and unknown boundary conditions (**BCCH card is used twice)

```
**BCCHANGE
TIME 10.0
ALL THE BCSETS AFTER FIRST CHANGE
**BCSET21
**BCSET22
.
**BCSET2n
```

```
**BCCH
FULL KEYWORD - **BCCHANGE

Status - OPTIONAL (Not available for all analysis types)

Full Keyword - **BCCHANGE

Function - Identifies an entirely new boundary condition definition in which the specified known and unknown boundary conditions will differ from the previously specified set.

Input Variables - NONE

Additional Information -

A TIME card must immediately follow the **BCCH card.

New boundary conditions for the entire body must be input corresponding to each **BCCH card. Even if certain **BCSETS do not change during the entire analysis, they must be repeated (redefined) for each **BCCH card.

All boundary condition sets preceding the **BCCH card are forgotten once the time of solution has passed over to the next set of boundary conditions after the **BCCH card.

Examples of Use -

1. Analysis with three unique sets of known and unknown boundary conditions (**BCCH card is used twice)

```
**BCCHANGE
TIME 10.0
ALL THE BCSETS AFTER FIRST CHANGE
**BCSET21
**BCSET22
.
**BCSET2n
```
```
Definition of Boundary Condition Changes

**BCCHANGE
TIME 20.0
ALL THE BCSETS AFTER SECOND CHANGES
**BCSET31
**BCSET32
.
.
**BCSET3n

As a particular case:

**BCSET
ID U1FIX
VALUE
LOCAL
GMR REG2
SURFACE SIDE1
DISP 1
SPLIT ALL
T 1 0.0

**BCCH
TIME 10.0

**BCSET
ID U1FIX
VALUE
LOCAL
GMR REG2
SURFACE SIDE1
TRAC 1
SPLIT ALL
T 1 0.0

**BCCH
TIME 20.0

**BCSET
ID U1FIX
VALUE
LOCAL
GMR REG2
SURFACE SIDE1
Definition of Boundary Condition Changes

DISP 1
SPLIST ALL
T 1 0.0
Definition of Boundary Condition Changes

**TIME** **VALUE**

Status - REQUIRED (if **BCCH is input)

Full Keyword - TIME

Function - Identifies the time at which the boundary condition change will occur.

Input Variables -

VALUE (Real) - REQUIRED
The time value at which the change occurs

Additional Information -

A new boundary condition definition (**BCSETS) must be given for the entire body immediately following the TIME card.

The change occurs for the first solution time (specified in the case input) greater than the VALUE specified on the TIME card of the **BCCH and less than the VALUE specified on the TIME cards of the subsequent **BCCH changes.

If more than one **BCCH card is present, the VALUE of time on the subsequent cards should be in ascending order.

The times specified on the TIME card within a BCSET may unnecessarily extend over the range allowed by the TIME card of the **BCCH change. In any case, the TIME card for **BCCH has priority.

Examples of Use -

1. Specifies the time at which the boundary condition change will occur.

```plaintext
**BCCH
TIME 10.0

**BCSET
ID U1FIX
VALUE
LOCAL
```

BEST3D User Manual  April, 1990  Page 5.203
6.0 EXAMPLE PROBLEMS

In this chapter example problems are presented to illustrate data preparation for BEST3D. An attempt has been made to keep the problem geometry as simple as possible so the user is not burdened with undue complexity. It is hoped that an analyst who is using an analysis procedure for the first time will find these example problems invaluable in the learning process.

The example problems are divided into the following categories based on BEST3D analysis type:

ELASTICITY
FORCED-VIBRATION
FREE-VIBRATION
HEAT TRANSFER
PLASTICITY
TRANSIENT DYNAMICS
STRESS ANALYSIS WITH HOLES AND INSERTS

Each problem includes the following items:

1) A Brief Problem Description
2) Geometry and Boundary Element Model
3) Input Data for running the problem in BEST3D
4) Selected Output from BEST3D

It should be noted that since the boundary element models illustrated utilize coarse meshes, the BEST3D results may differ somewhat from the theoretical values. However, with a finer mesh, the theoretical values should be obtained. Also, the results may vary somewhat depending on the computer system being used to run BEST3D.

An estimated RUN TIME is cited for each problem to give the user a feeling for the computer time needed to run the problem. All RUN TIMES are related to problem ELAS605, a simple elastic cube in tension, which will be considered to have a run time of 1 unit. A different problem which has a RUN TIME of 8 would take approximately eight times longer to run that the ELAS605 problem. However, these times will vary somewhat depending on the computer system being used to run BEST3D.
EXAMPLE PROBLEM: ELAS602

ANALYSIS TYPE:
THERMAELASTICITY,
3-D, STATIC, ELASTIC ANALYSIS WITH THERMAL BODY FORCE

PROBLEM DESCRIPTION:
TWO REGION, FREE EXPANSION OF A RECTANGULAR CUBE SUBJECTED TO A
UNIFORM AND A LINEAR TEMPERATURE DISTRIBUTION

BOUNDARY ELEMENT MODEL:
TWO SQUARE REGIONS, WITH SIX QUADRATIC BOUNDARY ELEMENTS PER
REGION. VOLUME INTEGRATION DISCRETIZATION APPROACH.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM
AND LINEAR TEMPERATURE DISTRIBUTIONS

SOLUTION POINTS TO VERIFY:

TIME 1: UNIFORM TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE LINEAR,
AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

<table>
<thead>
<tr>
<th>NODE</th>
<th>X-DISPLACEMENT</th>
<th>XX-STRESS</th>
<th>X-DISPLACEMENT</th>
<th>XX-STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ANAL.</td>
<td>BEST</td>
<td>ANAL.</td>
<td>BEST</td>
</tr>
<tr>
<td>8007</td>
<td>1.0</td>
<td>.99998</td>
<td>0</td>
<td>.0004</td>
</tr>
<tr>
<td>8009</td>
<td>2.0</td>
<td>1.99998</td>
<td>0</td>
<td>-.0005</td>
</tr>
</tbody>
</table>

TIME 2: LINEAR TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE QUADRATIC
AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

<table>
<thead>
<tr>
<th>NODE</th>
<th>X-DISPLACEMENT</th>
<th>XX-STRESS</th>
<th>X-DISPLACEMENT</th>
<th>XX-STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ANAL.</td>
<td>BEST</td>
<td>ANAL.</td>
<td>BEST</td>
</tr>
<tr>
<td>8007</td>
<td>0.5</td>
<td>.499998</td>
<td>0</td>
<td>.0025</td>
</tr>
<tr>
<td>8009</td>
<td>2.0</td>
<td>2.000000</td>
<td>0</td>
<td>-.0025</td>
</tr>
</tbody>
</table>

RUN TIME:
7 X BASE PROBLEM

MISCELLANEOUS:
STRESSES ARE ZERO RELATIVE TO (E*ALPHA*T).
THIS EXAMPLE CAN ALSO BE RUN USING THE GLOBAL SHAPE FUNCTION
APPROACH.
**CASE CONTROL**

**TITLE** FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE

**TIME** 1.0 2.0

**MATERIAL INPUT**

<table>
<thead>
<tr>
<th>ID</th>
<th>MAT 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEMP</td>
<td>70.</td>
</tr>
<tr>
<td>EMOD</td>
<td>100.</td>
</tr>
<tr>
<td>POIS</td>
<td>0.3</td>
</tr>
<tr>
<td>ALPHA</td>
<td>1.0</td>
</tr>
</tbody>
</table>

$  

$ DEFINE GEOMETRY OF REGION 1

$  

**GMR**

<table>
<thead>
<tr>
<th>ID</th>
<th>MAT 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAT 1</td>
<td></td>
</tr>
<tr>
<td>TREF</td>
<td>70.</td>
</tr>
</tbody>
</table>

$ TEMPERATURE AT WHICH MATERIAL CONSTANTS ARE EVALUATED FOR INTEGRATION

$ CHANGE IN TEMPERATURE DATUM

**TIMI 0.0**

**POINTS**

<table>
<thead>
<tr>
<th>POINT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.500</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>1.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>4</td>
<td>1.000</td>
<td>0.500</td>
<td>0.000</td>
</tr>
<tr>
<td>5</td>
<td>1.000</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>6</td>
<td>0.500</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>7</td>
<td>0.000</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>8</td>
<td>0.000</td>
<td>0.500</td>
<td>0.000</td>
</tr>
<tr>
<td>1001</td>
<td>0.000</td>
<td>0.000</td>
<td>0.500</td>
</tr>
<tr>
<td>1003</td>
<td>1.000</td>
<td>0.000</td>
<td>0.500</td>
</tr>
<tr>
<td>1005</td>
<td>1.000</td>
<td>1.000</td>
<td>0.500</td>
</tr>
<tr>
<td>1007</td>
<td>0.000</td>
<td>1.000</td>
<td>0.500</td>
</tr>
<tr>
<td>2001</td>
<td>0.000</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2002</td>
<td>0.500</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2003</td>
<td>1.000</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2004</td>
<td>1.000</td>
<td>0.500</td>
<td>1.000</td>
</tr>
<tr>
<td>2005</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2006</td>
<td>0.500</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2007</td>
<td>0.000</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2008</td>
<td>0.000</td>
<td>0.500</td>
<td>1.000</td>
</tr>
<tr>
<td>4001</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
</tr>
</tbody>
</table>

**SURFACE SURF11**

**TYPE QUAD**

**ELEMENTS**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>101</th>
<th>102</th>
<th>103</th>
<th>104</th>
<th>1</th>
<th>1</th>
<th>201</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>1</td>
<td>3</td>
<td>5</td>
<td>7</td>
<td>1</td>
<td>1</td>
<td>2001</td>
</tr>
<tr>
<td>Y</td>
<td>2</td>
<td>4</td>
<td>6</td>
<td>8</td>
<td>2</td>
<td>2</td>
<td>2002</td>
</tr>
<tr>
<td>Z</td>
<td>3</td>
<td>5</td>
<td>7</td>
<td>1</td>
<td>3</td>
<td>3</td>
<td>2003</td>
</tr>
<tr>
<td>1003</td>
<td>2003</td>
<td>2005</td>
<td>2007</td>
<td>2001</td>
<td>4</td>
<td>4</td>
<td>2004</td>
</tr>
<tr>
<td>2003</td>
<td>2005</td>
<td>2007</td>
<td>2001</td>
<td>2001</td>
<td>5</td>
<td>5</td>
<td>2006</td>
</tr>
</tbody>
</table>

**NORMAL 201**
ELASTICITY EXAMPLE PROBLEM ELAS602 / Input Data

VOLUME
TYPE QUAD

CELLS

4001

FULL

SAMPLING POINTS

4007 1.0 .5 .5

$ DEFINE GEOMETRY OF REGION 2 $

**GMR**

ID GMR2

MAT MAT1

TREF 70.

$ TEMPERATURE AT WHICH MATERIAL CONSTANTS $ $ ARE EVALUATED FOR INTEGRATION 

$ CHANGE IN TEMPERATURE DATUM 

POINTS

5001 1.0000 .0000 .0000
5002 1.5000 .0000 .0000
5003 2.0000 .0000 .0000
5004 2.0000 .5000 .0000
5005 2.0000 1.0000 .0000
5006 1.5000 1.0000 .0000
5007 1.0000 1.0000 .0000
5008 1.0000 .5000 .0000
6001 1.0000 .0000 .5000
6003 2.0000 .0000 .5000
6005 2.0000 1.0000 .5000
6007 1.0000 1.0000 .5000
7001 1.0000 .0000 1.0000
7002 1.5000 .0000 1.0000
7003 2.0000 .0000 1.0000
7004 2.0000 .5000 1.0000
7005 2.0000 1.0000 1.0000
7006 1.5000 1.0000 1.0000
7007 1.0000 1.0000 1.0000
7008 1.0000 .5000 1.0000
8001 1.5 .5 .5

SURFACE SURF21

TYPE QUAD

ELEMENTS

2101 5001 5002 5003 6003 7003 7002 7001 6001
2102 5003 5004 5005 6005 7005 7004 7003 6003
2103 5005 5006 5007 6007 7007 7006 7005 6005
2104 5007 5008 5001 6001 7001 7008 7007 6007
2001 5001 5002 5003 5004 5005 5006 5007 5008
2201 7001 7002 7003 7004 7005 7006 7007 7008

NORMAL 2201 +

VOLUME

TYPE QUAD

CELLS

2 5001 5002 5003 5004 5005 5006 5007 5008 6001 6003
2 6005 6007 7001 7002 7003 7004 7005 7006 7007 7008

BEST3D User Manual April, 1989 Page 6.5
FULL
SAMPLING POINTS
0002 1.25 .25 .25
0007 1.0 .5 .5
0009 2.0 .5 .5

$ DEFINE INTERFACE CONNECTION
$
**INTERFACE
GMR GMRI
SURFACE SURF11
ELEMENT 102
GMR GMRI2
SURFACE SURF21
ELEMENT 2104

$ FIX BODY TO PREVENT RIGID BODY DISPLACEMENT BUT ALLOW FREE EXPANSION
$
**BCSET
ID BC1
GMR GMRI
SURFACE SURF11
ELEMENTS 104
POINT 1
DISP 1
SPLIST 1
 T 1 0.0
DISP 2
SPLIST 1
 T 1 0.0
DISP 3
SPLIST 1
 T 1 0.0

**BCSET
ID BC2
GMR GMRI
SURFACE SURF11
ELEMENTS 104
POINTS 7
DISP 3
SPLIST 7
 T 1 0.0

**BCSET
ID BC3
GMR GMRI
SURFACE SURF11
ELEMENTS 104
POINTS 2001
DISP 1
SPLIST 2001
 T 1 0.0
**BCSET**
ID  BC4
GMR  GMR2
SURFACE  SURF21
ELEMENTS  2101
POINTS  5003
DISP  2
SPLIT  5003
T  1  0.0

**DEFINE TEMPERATURE DISTRIBUTION FOR ALL VOLUME SOURCE POINTS**
(NODAL POINTS OF THE VOLUME CELL)

**BODY FORCE**
THERMAL
TIME 1. 2.
GMR GMR1
TEMP
1  1.  .0000
2  1.  .5000
3  1.  1.0000
4  1.  1.0000
5  1.  1.0000
6  1.  .5000
7  1.  .0000
8  1.  .0000
1001  1.  .0000
1003  1.  1.0000
1005  1.  1.0000
1007  1.  .0000
2001  1.  .0000
2002  1.  .5000
2003  1.  1.0000
2004  1.  1.0000
2005  1.  1.0000
2006  1.  .5000
2007  1.  .0000
2008  1.  .0000
4001  1.  .5

GMR GMR2
TEMP
5001  1.  1.0000
5002  1.  1.5000
5003  1.  2.0000
5004  1.  2.0000
5005  1.  2.0000
5006  1.  1.5000
5007  1.  1.0000
5008  1.  1.0000
6001  1.  1.0000
6003  1.  2.0000
6005  1.  2.0000
6007  1.  1.0000
7001  1.  1.0000
7002  1.  1.5000
7003  1.  2.0000
ELASTICITY EXAMPLE PROBLEM ELAS602 / Input Data

7005  1.  2.0000
7006  1.  1.5000
7007  1.  1.0000
7008  1.  1.0000
7004  1.  2.0000
8001  1.  1.5

$ END OF DATA
### ELASTICITY EXAMPLE PROBLEM ELAS602 / Selected Output

**JOB TITLE:** FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE (PLANE STRESS)

**LOAD BALANCE**

| LOAD BALANCE | -0.13899E-05 | -0.45050E-05 | 0.70616E-05 |

**LOADS FOR REGION GNR1**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>102</td>
<td>-0.29030E-05</td>
<td>-0.58303E-06</td>
<td>0.20266E-04</td>
</tr>
<tr>
<td>103</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>104</td>
<td>0.16131E-05</td>
<td>0.13363E-06</td>
<td>-0.13164E-04</td>
</tr>
<tr>
<td>1</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>201</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>

**LOADS FOR REGION GNR2**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2101</td>
<td>0.00000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2102</td>
<td>0.00000E+00</td>
<td>0.00000E-08</td>
<td></td>
</tr>
<tr>
<td>2103</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2104</td>
<td>0.29030E-05</td>
<td>-0.58303E-05</td>
<td>-0.20266E-04</td>
</tr>
<tr>
<td>2001</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2201</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>

**LOAD BALANCE**

| LOAD BALANCE | 0.29030E-05 | -0.26833E-05 | -0.20266E-04 |

**JOB TITLE:** FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE (PLANE STRESS)

**INTERIOR STRESS AT TIME = 1.00000 FOR REGION = GNR2**

<table>
<thead>
<tr>
<th>NODE</th>
<th>SIGMA-XX</th>
<th>SIGMA-YY</th>
<th>SIGMA-ZZ</th>
<th>TAU-XY</th>
<th>TAU-XZ</th>
<th>TAU-YZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>5001</td>
<td>0.11234E-03</td>
<td>-0.13546E-04</td>
<td>0.26979E-04</td>
<td>0.29923E-05</td>
<td>0.32973E-06</td>
<td>0.21791E-04</td>
</tr>
<tr>
<td>5002</td>
<td>0.22409E-04</td>
<td>-0.20760E-05</td>
<td>0.13773E-04</td>
<td>0.64317E-05</td>
<td>0.00000E+00</td>
<td>0.79850E-06</td>
</tr>
<tr>
<td>5003</td>
<td>-0.32392E-04</td>
<td>-0.99707E-05</td>
<td>0.23063E-04</td>
<td>-0.20800E-05</td>
<td>0.20450E-06</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5004</td>
<td>-0.13599E-04</td>
<td>-0.56465E-05</td>
<td>0.50825E-05</td>
<td>0.14400E-04</td>
<td>0.00000E+00</td>
<td>0.79850E-06</td>
</tr>
<tr>
<td>5005</td>
<td>0.40448E-05</td>
<td>-0.22881E-05</td>
<td>0.54667E-05</td>
<td>-0.17270E-05</td>
<td>0.20212E-06</td>
<td>0.15994E-05</td>
</tr>
<tr>
<td>5006</td>
<td>0.24340E-05</td>
<td>0.30397E-05</td>
<td>0.42068E-05</td>
<td>-0.52063E-06</td>
<td>0.86865E-06</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5007</td>
<td>-0.43249E-05</td>
<td>-0.18667E-05</td>
<td>0.14896E-05</td>
<td>-0.28891E-05</td>
<td>0.60072E-05</td>
<td>-0.13327E-04</td>
</tr>
<tr>
<td>5008</td>
<td>0.16774E-04</td>
<td>-0.38389E-04</td>
<td>0.18250E-04</td>
<td>-0.18920E-05</td>
<td>0.12250E-05</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>6001</td>
<td>-0.29077E-04</td>
<td>-0.14919E-05</td>
<td>-0.48689E-05</td>
<td>-0.64101E-05</td>
<td>0.94679E-05</td>
<td>0.20337E-04</td>
</tr>
<tr>
<td>6002</td>
<td>0.11310E-04</td>
<td>-0.11012E-04</td>
<td>0.86535E-06</td>
<td>0.00000E+00</td>
<td>-0.19223E-05</td>
<td>0.22080E-06</td>
</tr>
<tr>
<td>6003</td>
<td>0.27619E-04</td>
<td>-0.20447E-04</td>
<td>0.28601E-05</td>
<td>0.00000E+00</td>
<td>-0.12947E-04</td>
<td>-0.31716E-04</td>
</tr>
<tr>
<td>6004</td>
<td>-0.88567E-04</td>
<td>-0.33779E-04</td>
<td>0.39831E-04</td>
<td>-0.80560E-05</td>
<td>0.37108E-05</td>
<td>-0.18665E-04</td>
</tr>
<tr>
<td>7001</td>
<td>-0.17966E-04</td>
<td>-0.51466E-05</td>
<td>-0.81414E-05</td>
<td>0.83196E-05</td>
<td>0.11060E-04</td>
<td>0.37159E-05</td>
</tr>
<tr>
<td>7002</td>
<td>-0.45146E-05</td>
<td>-0.30281E-04</td>
<td>-0.71980E-05</td>
<td>0.12581E-04</td>
<td>0.66848E-05</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>7003</td>
<td>0.68553E-05</td>
<td>0.60460E-05</td>
<td>-0.82096E-05</td>
<td>-0.32684E-05</td>
<td>-0.10965E-05</td>
<td>0.92943E-05</td>
</tr>
<tr>
<td>7004</td>
<td>0.53863E-05</td>
<td>0.32507E-05</td>
<td>-0.67515E-05</td>
<td>-0.48702E-05</td>
<td>0.00000E+00</td>
<td>-0.74293E-05</td>
</tr>
<tr>
<td>7005</td>
<td>0.84396E-01</td>
<td>-0.33167E-05</td>
<td>-0.85268E-05</td>
<td>0.29963E-05</td>
<td>-0.40047E-07</td>
<td>0.23147E-04</td>
</tr>
</tbody>
</table>

**BEST3D User Manual**

April, 1989

Page 6.9
## ELASTICITY EXAMPLE PROBLEM ELAS802 / Selected Output

<table>
<thead>
<tr>
<th>Page</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>7006</td>
<td>-0.183853E-04</td>
<td>-0.341219E-04</td>
<td>0.375223E-04</td>
<td>0.16671E-04</td>
<td>0.116924E-04</td>
</tr>
<tr>
<td>7007</td>
<td>-0.140968E-03</td>
<td>-0.474873E-04</td>
<td>0.650414E-04</td>
<td>-0.827377E-05</td>
<td>0.276637E-04</td>
</tr>
<tr>
<td>7008</td>
<td>-0.114998E-03</td>
<td>0.183996E-04</td>
<td>-0.299066E-04</td>
<td>-0.100648E-04</td>
<td>-0.164112E-04</td>
</tr>
<tr>
<td>8001</td>
<td>-0.313929E-04</td>
<td>-0.235858E-04</td>
<td>-0.225379E-04</td>
<td>0.364919E-06</td>
<td>0.13501E-04</td>
</tr>
<tr>
<td>8002</td>
<td>-0.190756E-05</td>
<td>-0.211490E-04</td>
<td>-0.875142E-05</td>
<td>0.569322E-05</td>
<td>0.316275E-04</td>
</tr>
<tr>
<td>8007</td>
<td>-0.352388E-04</td>
<td>-0.305789E-04</td>
<td>-0.237930E-04</td>
<td>-0.281723E-05</td>
<td>0.339229E-04</td>
</tr>
<tr>
<td>8009</td>
<td>-0.176482E-06</td>
<td>-0.117582E-04</td>
<td>-0.133781E-04</td>
<td>0.806576E-07</td>
<td>-0.343067E-05</td>
</tr>
</tbody>
</table>

*BEST3D User Manual*  
April, 1989  
Page 6.10
EXAMPLE PROBLEM: ELAS605

ANALYSIS TYPE: 3-D LINEAR ELASTICITY

PROBLEM DESCRIPTION:
SIMPLE CUBE IN TENSION IN X-DIRECTION.

BOUNDARY ELEMENT MODEL:
SINGLE CUBE COMPOSED OF SIX BOUNDARY ELEMENTS.
SAMPLING POINTS INCLUDED TO MONITOR INTERIOR BEHAVIOR.

REFERENCE FOR ANALYTICAL SOLUTION:
CRANDALL AND DAHL, AN INTRODUCTION TO THE MECHANICS OF SOLIDS,
SECOND EDITION (1972), PG. 81-84. X-DISPLACEMENT = FL/AE.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>NODE</th>
<th>X-DISPLACEMENT</th>
<th>Y-DISPLACEMENT</th>
<th>X-STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>0.0001</td>
<td>-0.00003000</td>
<td>1000.00</td>
</tr>
</tbody>
</table>

RUN TIME:
1.0 X BASE PROBLEM ( THIS IS THE BASE PROBLEM! )

MISCELLANEOUS:
ELASTICITY EXAMPLE PROBLEM ELAS805 / Geometry

NODES AND ELEMENTS

BEST3D User Manual
April, 1989
Page 6.12
**CASE**
**TITLE**  A UNIT CUBE IN SIMPLE TENSION
**ELASTICITY**

**MATE**
ID MAT1
TEMP 70.0
EMOD 10.E6
POIS 0.3

**GMR**
ID GMR1
MAT MAT1
TREF 70.0
POINTS
1  0.0  0.0  0.0
2  0.0  0.5  0.0
3  0.0  1.0  0.0
4  0.5  0.0  0.0
5  0.5  1.0  0.0
6  1.0  0.0  0.0
7  1.0  0.5  0.0
8  1.0  1.0  0.0
9  0.0  0.0  0.5
10 0.0  1.0  0.5
11 1.0  0.0  0.5
12 1.0  1.0  0.5
13 0.0  0.0  1.0
14 0.0  0.5  1.0
15 0.0  1.0  1.0
16 0.5  0.0  1.0
17 0.5  1.0  1.0
18 1.0  0.0  1.0
19 1.0  0.5  1.0
20 1.0  1.0  1.0

SURFACE SURF1
TYPE QUAD
ELEMENTS
1 1 2 3 5 8 7 6 4
2 13 14 15 17 20 19 18 16
3 1 2 3 10 15 14 13 9
4 6 7 8 12 20 19 18 11
5 1 4 6 11 18 16 13 9
6 3 5 8 12 20 17 15 10
NORMAL 1 +
SAMPLING POINTS
21 0.5  0.0  0.5
22 0.5  0.25 0.5
23 0.5  0.5  0.5
24 0.5  0.75 0.5
25 0.5  1.0  0.5
**BCSET
ID DISP1
VALUE
GMR GMR1
SURFACE SURF1
  ELEMENTS 3
  DISP 1
  SPLIST ALL
  T 1  0.0

**BCSET
ID DISP2
VALUE
GMR GMR1
SURFACE SURF1
  ELEMENTS 5
  DISP 2
  SPLIST ALL
  T 1  0.0

**BCSET
ID DISP3
VALUE
GMR GMR1
SURFACE SURF1
  ELEMENTS 1
  DISP 3
  SPLIST ALL
  T 1  0.0

**BCSET
ID TRAC1
VALUE
GMR GMR1
SURFACE SURF1
  ELEMENTS 4
  TRAC 1
  SPLIST ALL
  T 1  1000.0

$ END OF DATA
**** CASE CONTROL INPUT ****

JOB TITLE   A UNIT CUBE IN SIMPLE TENSION

PLANE STRESS FLAG: 0  PLANE STRAIN FLAG: 0
ASYMMETRY FLAG: 0  DIMENSIONALITY FLAG: 3
PLASTICITY FLAG: 0  THERMAL FLAG: 0
CENTRIFUGAL FLAG: 0  INERTIAL LOAD FLAG: 0
COUPLED FLAG: 0  COUPLED THERMAL FLAG: 0
CONSOLIDATION FLAG: 0  DIFFUSION FLAG: 0
BOUNDARY RESTART: 0  GLOBAL SHAPE FUNCTION: 0
DOMAIN RESTART: 0  STEADY-STATE FLAG: 0

THERMAL INHOMOGENEITY FLAG: 0
MATERIAL INHOMOGENEITY FLAG: 0
NON-ITERATIVE PLASTICITY FLAG: 0
TRANSIENT DYNAMICS FLAG: 0
STEADY STATE DYNAMICS FLAG: 0
FLUID DYNAMICS FLAG: 0

NUMBER OF DEGREES OF FREEDOM: 3

BOUNDARY INTEGRATION EPSILON: 0.00100000
INTERIOR INTEGRATION EPSILON: 0.00100000

PRINTING FLAGS FOR BOUNDARY VALUES:
  BOUNDARY DISP. AND TRAC.  1
  MODAL DISP.,STRESS AND STRAIN  1
  LOAD CALCULATION  1

**** MATERIAL INPUT ****

MATERIAL NAME: MAT1

ELASTIC
ISOTROPIC MATERIAL

DENSITY: 0.0000E+00
DAMPING COEFF: 0.0000
POISSON'S RATIO: 0.3000

TEMP  ALPHA  E
0.70000E+02  0.00000E+00  0.10000E+08

**** OTR INPUT ****
ELASTICITY EXAMPLE PROBLEM ELAS805 / Selected Output

REGION 1
NAME: GN1
MATERIAL MAT1

REFERENCE TEMPERATURE 70.0000
INITIAL TEMPERATURE OF GN1 0.0000

NODES 25 ELEMENTS 6 SURFACES 1
SOURCE POINTS 20 CELLS 0 INFINITE ELEMENTS 0
ADDITIONAL JUMP TERM 0.00000 ENCLOSED ELEMENTS 0
GLOBAL SHAPE FUNCTION NODES 0 HOLE ELEMENTS 0
NUMBER OF INSERTS 0 INSERT ELEMENTS 0

INFORMATIONAL MESSAGE 1001
NOT ALL POINTS ARE DEFINED AS BOUNDARY SOURCE POINTS

COORDINATE LIST

<table>
<thead>
<tr>
<th>NODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.5000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>1.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.5000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>0.5000</td>
<td>1.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>1.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>7</td>
<td>1.0000</td>
<td>0.5000</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>1.0000</td>
<td>1.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.5000</td>
</tr>
<tr>
<td>10</td>
<td>0.0000</td>
<td>1.0000</td>
<td>0.5000</td>
</tr>
<tr>
<td>11</td>
<td>1.0000</td>
<td>0.0000</td>
<td>0.5000</td>
</tr>
<tr>
<td>12</td>
<td>1.0000</td>
<td>1.0000</td>
<td>0.5000</td>
</tr>
<tr>
<td>13</td>
<td>0.0000</td>
<td>0.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>14</td>
<td>0.0000</td>
<td>0.5000</td>
<td>1.0000</td>
</tr>
<tr>
<td>15</td>
<td>0.0000</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>16</td>
<td>0.5000</td>
<td>0.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>17</td>
<td>0.5000</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>18</td>
<td>1.0000</td>
<td>0.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>19</td>
<td>1.0000</td>
<td>0.5000</td>
<td>1.0000</td>
</tr>
<tr>
<td>20</td>
<td>1.0000</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>21</td>
<td>0.5000</td>
<td>0.0000</td>
<td>0.5000</td>
</tr>
<tr>
<td>22</td>
<td>0.5000</td>
<td>0.2500</td>
<td>0.5000</td>
</tr>
<tr>
<td>23</td>
<td>0.5000</td>
<td>0.6000</td>
<td>0.5000</td>
</tr>
<tr>
<td>24</td>
<td>0.5000</td>
<td>0.7500</td>
<td>0.5000</td>
</tr>
<tr>
<td>25</td>
<td>0.5000</td>
<td>1.0000</td>
<td>0.5000</td>
</tr>
</tbody>
</table>

SURFACE SURF1 QUADRATIC VARIATION

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>NODES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1, 2, 3, 5, 8, 7, 6, 4</td>
</tr>
<tr>
<td>2</td>
<td>13, 16, 15, 19, 20, 17, 15, 14</td>
</tr>
<tr>
<td>3</td>
<td>1, 9, 13, 14, 15, 10, 3, 2</td>
</tr>
<tr>
<td>4</td>
<td>6, 7, 8, 12, 20, 19, 18, 11</td>
</tr>
<tr>
<td>5</td>
<td>1, 4, 6, 11, 18, 16, 13, 9</td>
</tr>
<tr>
<td>6</td>
<td>3, 10, 15, 17, 20, 12, 8, 5</td>
</tr>
</tbody>
</table>

SOURCE POINT LIST

BEST3D User Manual April, 1989 Page 6.16
ELASTICITY EXAMPLE PROBLEM ELAS605 / Selected Output

18 19 20

** Boundary Condition Input **

Boundary Condition Set Name DISP1 Type: Value
GNE GNE1 SURFACE SURF1

Element List
3

Source Point List
1 9 13 14 15 10 3 2

Component 1 Displacement Input

Data Values:
0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00

** Boundary Condition Input **

Boundary Condition Set Name DISP2 Type: Value
GNE GNE1 SURFACE SURF1

Element List
3

Source Point List
1 9 13 14 15 10 3 2

Component 2 Displacement Input

Data Values:
0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00

** Boundary Condition Input **

Boundary Condition Set Name DISP3 Type: Value
GNE GNE1 SURFACE SURF1

Element List
1

Source Point List
1 2 3 5 8 7 6 4

Component 3 Displacement Input

Data Values:
0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
ELASTICITY EXAMPLE PROBLEM ELAS605 / Selected Output

***** BOUNDARY CONDITION INPUT *****

BOUNDARY CONDITION SET NAME TRAC1

TYPE: VALUE

ELEMENT LIST

4

SOURCE POINT LIST

6 7 8 12 20 19 18 11

COMPONENT 1 TRACTION INPUT

DATA VALUES:

0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04

** ERROR SUMMARY AFTER INPUT PHASE **

FATAL ERRORS: 0
WARNING MESSAGES: 0
INFORMATIONAL MESSAGES: 1

INFORMATIONAL MESSAGES LISTED ABOVE:

1001

BEGIN SURFACE INTEGRATION OF GMR GMRI

HIGH PRECISION

SOURCE POINTS

INTERIOR POINTS

END SURFACE INTEGRATION OF GMR GMRI

MATRIX DECOMPOSITION - DIAGONAL BLOCK 1

CONDITION NUMBER 0.10394E+03

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION

BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GMRI

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>NODE NO.</th>
<th>X-DISPL.</th>
<th>Y-DISPL.</th>
<th>Z-DISPL.</th>
<th>X-TRAC.</th>
<th>Y TRAC.</th>
<th>Z TRAC.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>0.00000E+00</td>
<td>-0.15000E-04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>0.00000E+00</td>
<td>-0.30000E-04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>5</td>
<td>0.50000E-04</td>
<td>-0.50000E-04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>8</td>
<td>0.99999E-04</td>
<td>-0.29999E-04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>7</td>
<td>0.10000E-03</td>
<td>-0.15000E-03</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>6</td>
<td>0.10001E-03</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>1</td>
<td>4</td>
<td>0.60000E-04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>
ELASTICITY EXAMPLE PROBLEM ELAS605 / Selected Output

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>13</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>16</td>
<td>0.50000E-04</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>18</td>
<td>0.99994E-04</td>
<td>0.0000E+00</td>
<td>-0.29991E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>19</td>
<td>0.10000E-03</td>
<td>-0.15000E-04</td>
<td>-0.29997E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0.99987E-04</td>
<td>-0.29994E-04</td>
<td>-0.29994E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>0.50000E-04</td>
<td>-0.3000E-04</td>
<td>-0.30010E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>16</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>-0.30005E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>14</td>
<td>0.0000E+00</td>
<td>-0.15005E-04</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>-0.15005E-04</td>
<td>-0.10001E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>13</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>-0.30004E-04</td>
<td>-0.10006E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>14</td>
<td>0.0000E+00</td>
<td>-0.15005E-04</td>
<td>-0.30005E-04</td>
<td>-0.10002E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>0.0000E+00</td>
<td>-0.30005E-04</td>
<td>-0.30005E-04</td>
<td>-0.10004E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>0.0000E+00</td>
<td>-0.30005E-04</td>
<td>-0.15005E-04</td>
<td>-0.10002E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>0.0000E+00</td>
<td>-0.30004E-04</td>
<td>0.0000E+00</td>
<td>-0.10005E+04</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0.0000E+00</td>
<td>-0.15005E-04</td>
<td>0.0000E+00</td>
<td>-0.10001E+04</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>6</td>
<td>0.10001E-03</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>7</td>
<td>0.10001E-03</td>
<td>-0.15000E-04</td>
<td>0.0000E+00</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>0.99994E-04</td>
<td>-0.29991E-04</td>
<td>0.0000E+00</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>12</td>
<td>0.10000E-03</td>
<td>-0.29997E-04</td>
<td>-0.15000E-04</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>20</td>
<td>0.99987E-04</td>
<td>-0.29994E-04</td>
<td>-0.29994E-04</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>19</td>
<td>0.10001E-03</td>
<td>-0.15000E-04</td>
<td>-0.29997E-04</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>18</td>
<td>0.99994E-04</td>
<td>0.0000E+00</td>
<td>-0.29991E-04</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>11</td>
<td>0.10001E-03</td>
<td>0.0000E+00</td>
<td>-0.15000E-04</td>
<td>0.10000E+00</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GNA1

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>NODE NO.</th>
<th>X-DISPL.</th>
<th>Y-DISPL.</th>
<th>Z-DISPL.</th>
<th>X-TRAC.</th>
<th>Y TRAC.</th>
<th>Z TRAC.</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>3</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>-0.1500E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>6</td>
<td>15</td>
<td>0.0000E+00</td>
<td>-0.3000E-04</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>6</td>
<td>17</td>
<td>0.5000E-04</td>
<td>-0.3000E-04</td>
<td>-0.3000E-04</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>
Elasticity Example Problem ELAS05 / Selected Output

<table>
<thead>
<tr>
<th>Element</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.95296E-02</td>
</tr>
<tr>
<td>2</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>-0.10000E+04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>4</td>
<td>0.10000E+04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.00000E+00</td>
<td>0.95156E-02</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>

Load Balance: -0.20203E-01 0.95156E-02 0.95296E-02

Job Title: A Unit Cube in Simple Tension

Load Calculation at Time = 0.000000

Loads for Region UNR1

<table>
<thead>
<tr>
<th>Node</th>
<th>Displacement</th>
<th>Stress</th>
<th>Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I/Y/Z</td>
<td>IX/YY/ZZ</td>
<td>IX/IX/YZ</td>
</tr>
<tr>
<td>1</td>
<td>0.00000E+00</td>
<td>0.10000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.12547E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.12547E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.00000E+00</td>
<td>0.10000E+00</td>
<td>-0.42452E-01</td>
</tr>
<tr>
<td></td>
<td>-0.15000E+04</td>
<td>-0.36594E-01</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.19430E+00</td>
<td>-0.96197E-04</td>
</tr>
<tr>
<td>3</td>
<td>0.00000E+00</td>
<td>0.10000E+00</td>
<td>-0.53161E-01</td>
</tr>
<tr>
<td></td>
<td>-0.30004E+04</td>
<td>0.11022E-00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.35271E-01</td>
<td>-0.13619E-02</td>
</tr>
<tr>
<td>4</td>
<td>0.50000E+04</td>
<td>0.10001E+04</td>
<td>0.46646E-01</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.66680E-02</td>
<td>0.46653E-01</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.66680E-02</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.50000E+04</td>
<td>0.99992E+03</td>
<td>-0.41151E-01</td>
</tr>
<tr>
<td></td>
<td>-0.30011E+04</td>
<td>-0.13870E-01</td>
<td>0.49563E-01</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.90094E-01</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.10001E-03</td>
<td>0.99975E-03</td>
<td>0.60773E-01</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.25114E+00</td>
<td>0.60773E-01</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>-0.25113E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>
### Elasticity Example Problem

#### Selected Output

<table>
<thead>
<tr>
<th>Node</th>
<th>Displacement</th>
<th>Stress</th>
<th>Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I/Y/Z</td>
<td>II/YY/ZZ</td>
<td>II/ZZ/YZ</td>
</tr>
<tr>
<td>7</td>
<td>0.1000E-03</td>
<td>0.99991E+03</td>
<td>0.40438E-01</td>
</tr>
<tr>
<td></td>
<td>-0.1500E+00</td>
<td>0.17284E+01</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.0000E+00</td>
<td>-0.14902E+00</td>
<td>0.16646E-01</td>
</tr>
<tr>
<td>8</td>
<td>0.99994E-04</td>
<td>0.99736E+03</td>
<td>-0.33964E-02</td>
</tr>
<tr>
<td></td>
<td>-0.29991E+00</td>
<td>0.10892E+00</td>
<td>0.66612E-01</td>
</tr>
<tr>
<td></td>
<td>0.0000E+00</td>
<td>-0.12321E+00</td>
<td>0.24402E-01</td>
</tr>
<tr>
<td>9</td>
<td>0.0000E+00</td>
<td>0.10000E+04</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.0000E+00</td>
<td>-0.19434E+00</td>
<td>-0.42641E-01</td>
</tr>
<tr>
<td></td>
<td>-0.15005E+04</td>
<td>0.36622E-01</td>
<td>-0.10666E-03</td>
</tr>
<tr>
<td>10</td>
<td>0.0000E+00</td>
<td>0.10002E+04</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>-0.30006E+04</td>
<td>0.52651E-01</td>
<td>-0.29666E-01</td>
</tr>
<tr>
<td></td>
<td>-0.15006E+04</td>
<td>0.36850E-01</td>
<td>-0.20378E-02</td>
</tr>
<tr>
<td>11</td>
<td>0.10001E-03</td>
<td>0.99991E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.0000E+00</td>
<td>-0.14898E+00</td>
<td>0.40439E-01</td>
</tr>
<tr>
<td></td>
<td>-0.15004E+04</td>
<td>0.17348E-01</td>
<td>0.15598E-01</td>
</tr>
<tr>
<td>12</td>
<td>0.10001E-03</td>
<td>0.99990E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>-0.29978E+00</td>
<td>0.10396E+00</td>
<td>0.32741E-01</td>
</tr>
<tr>
<td></td>
<td>-0.15005E+04</td>
<td>0.59256E-01</td>
<td>0.89568E-02</td>
</tr>
<tr>
<td>13</td>
<td>0.0000E+00</td>
<td>0.10003E+04</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.0000E+00</td>
<td>-0.35310E-01</td>
<td>-0.53181E-01</td>
</tr>
<tr>
<td></td>
<td>-0.30004E+04</td>
<td>0.11018E+00</td>
<td>-0.13642E-02</td>
</tr>
</tbody>
</table>

**Job Title:** A UNIT CUBE IN SIMPLE TENSION

**Modal Output at Time:** 0.000000 FOR REGION = G841

**Node** | **Displacement** | **Stress** | **Strain**
---|------------------|------------|--------|
| I/Y/Z | II/YY/ZZ | II/ZZ/YZ | II/YY/ZZ | II/ZZ/YZ |
| 14    | 0.0000E+00   | 0.10002E+04 | -0.29635E-01 | 0.10002E-03 | -0.38526E-08 |
|      | -0.15006E-04 | 0.36623E-01 | 0.00000E+00 | -0.30006E-04 | 0.00000E+00 |
|      | -0.30006E-04 | 0.52651E-01 | -0.20378E-02 | -0.30005E-04 | 0.26489E-09 |
| 15    | 0.0000E+00   | 0.10002E+04 | -0.37327E-01 | 0.10002E-03 | -0.48525E-08 |
|      | -0.30006E-04 | 0.10503E+00 | 0.37365E-01 | -0.30004E-04 | 0.48576E-08 |
|      | -0.30004E-04 | 0.10514E+00 | 0.25862E-02 | -0.30004E-04 | 0.33621E-09 |
| 16    | 0.50004E-04  | 0.99992E+03 | 0.49639E-01 | 0.99994E-04 | 0.64401E-08 |
|      | 0.0000E+00   | -0.89941E-01 | -0.41141E-01 | -0.30007E-04 | -0.53483E-08 |
|      | -0.30011E-04 | 0.13901E-01 | 0.00000E+00 | -0.29994E-04 | 0.00000E+00 |
| 17    | 0.50004E-04  | 0.99986E+03 | -0.43996E-01 | 0.99987E-04 | -0.57194E-08 |
|      | -0.30010E-04 | 0.18402E-01 | -0.43992E-01 | -0.29974E-04 | 0.57189E-08 |
|      | -0.30010E-04 | 0.18250E-01 | 0.00000E+00 | -0.29974E-04 | 0.00000E+00 |
| 18    | 0.99994E-04  | 0.99976E+03 | 0.66575E-01 | 0.99976E-04 | 0.85483E-08 |
|      | 0.0000E+00   | -0.12329E+00 | -0.33613E-02 | -0.30005E-04 | 0.44020E-09 |

**Best3D User Manual**

April, 1989 Page 6.21
### ELASTICITY EXAMPLE PROBLEM ELAS605 / Selected Output

- $-0.29991E-04$  $0.10891E+00$  $-0.24587E-01$  $-0.29978E-04$  $-0.31963E-06$

<table>
<thead>
<tr>
<th>Node</th>
<th>I Displacement</th>
<th>Y Displacement</th>
<th>Z Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>$0.10000E+03$</td>
<td>$0.99990E+03$</td>
<td>$0.53983E-01$</td>
</tr>
<tr>
<td></td>
<td>$0.29997E-04$</td>
<td>$0.10406E+00$</td>
<td>$0.89480E-02$</td>
</tr>
<tr>
<td>20</td>
<td>$0.99987E-04$</td>
<td>$0.99796E+03$</td>
<td>$-0.20816E-01$</td>
</tr>
<tr>
<td></td>
<td>$-0.29994E-04$</td>
<td>$0.10793E+00$</td>
<td>$-0.20753E-01$</td>
</tr>
</tbody>
</table>

### JOB TITLE: A UNIT CUBE IN SIMPLE TENSION

**INTERIOR DISPLACEMENT AT TIME = 0.0000 FOR REGION = GR1**

<table>
<thead>
<tr>
<th>Node</th>
<th>I Displacement</th>
<th>Y Displacement</th>
<th>Z Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>$0.500135E-04$</td>
<td>$0.000000E+00$</td>
<td>$-0.150089E-04$</td>
</tr>
<tr>
<td>22</td>
<td>$0.500090E-04$</td>
<td>$-0.750441E-05$</td>
<td>$-0.150052E-04$</td>
</tr>
<tr>
<td>23</td>
<td>$0.500105E-04$</td>
<td>$-0.150052E-04$</td>
<td>$-0.150045E-04$</td>
</tr>
<tr>
<td>24</td>
<td>$0.500067E-04$</td>
<td>$-0.225076E-04$</td>
<td>$-0.150045E-04$</td>
</tr>
<tr>
<td>26</td>
<td>$0.500091E-04$</td>
<td>$-0.300126E-04$</td>
<td>$-0.150074E-04$</td>
</tr>
</tbody>
</table>

### JOB TITLE: A UNIT CUBE IN SIMPLE TENSION

**INTERIOR STRESS AT TIME = 0.0000 FOR REGION = GR1**

<table>
<thead>
<tr>
<th>Node</th>
<th>Sigma-XX</th>
<th>Sigma-YY</th>
<th>Sigma-ZZ</th>
<th>Tau-XY</th>
<th>Tau-EZ</th>
<th>Tau-YZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>$0.100017E+04$</td>
<td>$0.981304E-01$</td>
<td>$-0.322382E-01$</td>
<td>$0.000000E+00$</td>
<td>$0.546609E-03$</td>
<td>$0.000000E+00$</td>
</tr>
<tr>
<td>22</td>
<td>$0.999768E+03$</td>
<td>$0.227315E+00$</td>
<td>$-0.385985E-01$</td>
<td>$-0.166191E+00$</td>
<td>$0.128456E-02$</td>
<td>$0.643814E-01$</td>
</tr>
<tr>
<td>23</td>
<td>$0.100012E+04$</td>
<td>$-0.556403E+00$</td>
<td>$-0.556858E+01$</td>
<td>$-0.231984E-02$</td>
<td>$-0.232729E-02$</td>
<td>$0.584437E-02$</td>
</tr>
<tr>
<td>24</td>
<td>$0.999633E+03$</td>
<td>$0.353963E+00$</td>
<td>$-0.154854E+00$</td>
<td>$0.160049E+00$</td>
<td>$0.271529E-03$</td>
<td>$0.514198E+01$</td>
</tr>
<tr>
<td>26</td>
<td>$0.100003E+04$</td>
<td>$0.000000E+00$</td>
<td>$-0.982690E-01$</td>
<td>$0.000000E+00$</td>
<td>$0.226028E-02$</td>
<td>$0.000000E+00$</td>
</tr>
</tbody>
</table>

**END OF ANALYSIS**
VERIFICATION PROBLEM: D.402

ANALYSIS TYPE:
THermoelasTictY, 
3-D, StAtic, ElAsTic anaLYsis WITh a THERmal HoT sPoT

PROBLEM DESCRIPTION:
FREE EXPANSION OF A RECTANGULAR CUBE SUBJECTED TO A THERMAL
HoT sPoT OF UNIT VOLUME

BOUNDARY ELEMENT MODEL:
OCTAL SYMMETRY MODEL CONSISTING OF TWO SQUARE REGIONS, WITH THREE
QUADRATIC BOUNDARY ELEMENTS IN REGION ONE AND FOUR IN REGION TWO.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM
AND LINEAR TEMPERATURE DISTRIBUTIONS

SOLUTION POINTS TO VERIFY:

TIME 1: UNIFORM TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE LINEAR,
AND STRESSES ARE ZERO (\(<\<100, E\*\alpha T\)

<table>
<thead>
<tr>
<th>GMR</th>
<th>NODE</th>
<th>X-DISP.</th>
<th>Y-DISP.</th>
<th>XX-STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>GMR1</td>
<td>3000</td>
<td>.25000</td>
<td>.25000</td>
<td>.0010</td>
</tr>
<tr>
<td>GMR2</td>
<td>3005</td>
<td>.74997</td>
<td>.24998</td>
<td>-.0044</td>
</tr>
</tbody>
</table>

RUN TIME: 
1.3 X BASE PROBLEM

MISCELLANEOUS:
STRESSES ARE ZERO RELATIVE TO \(E\*\alpha T\).
**CASE**

**TITLE - D.402 - 3-D 2-REGION HOTSPOT**

TIMES 1.
SYMM OCTA
ECHO

**MATE**

ID MAT1
TEMP 0.0
EMOD 100.
POIS 0.3
ALPHA 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
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

102 3 4 5 1005 2005 2004 2003 1003
103 5 6 7 1007 2007 2006 2005 1005
201 2001 2002 2003 2004 2005 2006 2007 2008

NORMAL 201 +
SAMPLING POINT
3000 .5 .5 .5

**GMR**

ID GMR2
MAT MAT1
TREF 70.0

POINTS

1 1.0000 .0000 .0000

*BEST3D User Manual*

April, 1989
```
2 1.5000   1.0000   0.0000    0.0000
3 2.0000   0.0000   0.0000    0.0000
4 2.0000   1.0000   0.0000    0.0000
5 2.0000   0.5000   0.0000   0.0000
6 1.5000   0.0000   0.0000    0.0000
7 1.0000   0.0000   0.0000    0.0000
8 1.0000   0.0000   0.0000    0.0000
1001 1.0000 0.0000   0.0000  1.0000
1003 2.0000 0.0000   0.0000  1.0000
1005 2.0000 1.0000   0.0000  1.0000
1007 1.0000 1.0000   0.0000  1.0000
2001 1.0000 0.0000   1.0000  1.0000
2002 1.5000 0.0000   0.0000  1.0000
2003 2.0000 0.0000   0.0000  1.0000
2004 2.0000 0.5000   0.0000  1.0000
2005 2.0000 1.0000   0.0000  1.0000
2006 1.5000 1.0000   0.0000  1.0000
2007 1.0000 1.0000   0.0000  1.0000
2008 1.0000 0.5000   1.0000  1.0000

SURFACE SURF21
TYPE QUAD
ELEMENTS
  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
NORMA L  201 +
SAMPLING POINT
  3005  1.5  .5  .5

**INTERFACE
  GMR GMR1
  SURFACE SURF11
  ELEMENT 102
  GMR GMR2
  SURFACE SURF21
  ELEMENT 104

$  INPUT FOR HOT SPOT

**BODY FORCE
  HOT SPOT
  TIME 1.  3.
  GMR GMR1
  TEMP
  3000 1.  0.5  1.5
  GMR GMR2
  TEMP
  3005 1.  0.5  1.5

$  END OF DATA
```
EXAMPLE PROBLEM: FORC601

ANALYSIS TYPE: DYNAMIC ANALYSIS
            3-D, STEADY-STATE

PROBLEM DESCRIPTION:
DYNAMIC STIFFNESS OF RIGID SURFACE SQUARE FOOTING RESTING
ON ELASTIC HALF-SPACE (SOIL).

BOUNDARY ELEMENT MODEL:
BECAUSE OF QUADRATIC SYMMETRY IN GEOMETRY AND LOAD, ONLY
ONE QUARTER OF THE GEOMETRY IS MODELED. 4 ELEMENTS ARE
USED TO DISCRETIZE THE SOIL-FOUNDATION INTERFACE WHILE THE
FREE-SURFACE IS MODELED WITH 6 ELEMENTS.

THE FOUNDATION IS SUBJECTED TO UNIT VERTICAL DISPLACEMENT
(DIRECTION 3) WHILE TRACTIONS ON THE OTHER TWO DIRECTIONS
ARE ASSUMED TO BE ZERO. THIS CORRESPONDS TO THE SO-CALLED
'RELAXED' CONDITION. THE FREE-SURFACE IS ASSUMED TO BE
TRACTION FREE.

SOLUTION POINTS TO VERIFY:

| EXCITING CIRCULAR | ELEMENT | VERTICAL LOAD (BEST) |
| FREQUENCY | | (DIRECTION 3) | REAL PART | COMPLEX PART |
|-----------|---------|---------------------|
| 0.001     | 1       | 0.3553E+00          | 0.2878E-03 |
| 0.001     | 3       | 0.5434E+00          | 0.4398E-03 |
| 1.5       | 1       | 0.1242E+00          | 0.5871E+00 |
| 1.5       | 3       | 0.4411E+00          | 0.8158E+00 |

RUN TIME:
20 X BASE PROBLEM

MISCELLANEOUS:
**CASE CONTROL**

**TITLE**  SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)

**SYMMETRY**  QUAD

FORCED 0.001  1.5

PRINT LOAD

**MATE**

ID MAT1

TEMP 70.

EMOD 2.66666

POIS 0.33333

DENS 1.0

DAMP 0.0

**GMR**

ID GMR1

MAT MAT1

TREF 70.

HALF

$ SINCE THE PROBLEM INVOLVES HALF-SPACE,

$ THE CARD 'HALF' IS USED IN THE GMR INPUT.

$ THIS ELIMINATES THE NEED FOR 'ENCLOSING ELEMENTS'.

**POINTS**

1 0.000000  0.000000  0.000000
2 0.700000  0.700000  0.000000
3 0.700000  0.000000  0.000000
4 0.000000  0.700000  0.000000
5 1.000000  0.000000  0.000000
6 1.000000  1.000000  0.000000
7 0.000000  1.000000  0.000000
8 1.500000  0.000000  0.000000
9 1.500000  1.500000  0.000000
10 0.000000  1.500000  0.000000
11 3.000000  0.000000  0.000000
12 3.000000  3.000000  0.000000
13 0.000000  3.000000  0.000000
14 6.000000  0.000000  0.000000
15 6.000000  6.000000  0.000000
16 0.000000  6.000000  0.000000
17 0.350000  0.350000  0.000000
18 0.350000  0.000000  0.000000
19 0.700000  0.350000  0.000000
20 0.000000  0.350000  0.000000
21 0.350000  0.700000  0.000000
22 0.850000  0.000000  0.000000
23 1.000000  0.500000  0.000000
24 0.850000  0.850000  0.000000
25 0.500000  1.000000  0.000000
26 0.000000  0.850000  0.000000
27 1.250000  0.000000  0.000000
28 1.500000  0.750000  0.000000
29 1.250000  1.250000  0.000000
30 0.750000  1.500000  0.000000
FORCED VIBRATION EXAMPLE PROBLEM FORC601 / Input Data

<table>
<thead>
<tr>
<th></th>
<th>0.000000</th>
<th>1.250000</th>
<th>0.000000</th>
</tr>
</thead>
<tbody>
<tr>
<td>32</td>
<td>2.250000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>33</td>
<td>3.000000</td>
<td>1.500000</td>
<td>0.000000</td>
</tr>
<tr>
<td>34</td>
<td>2.250000</td>
<td>2.250000</td>
<td>0.000000</td>
</tr>
<tr>
<td>35</td>
<td>1.500000</td>
<td>3.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>36</td>
<td>0.000000</td>
<td>2.250000</td>
<td>0.000000</td>
</tr>
<tr>
<td>37</td>
<td>4.500000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>38</td>
<td>6.000000</td>
<td>3.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>39</td>
<td>4.500000</td>
<td>4.500000</td>
<td>0.000000</td>
</tr>
<tr>
<td>40</td>
<td>3.000000</td>
<td>6.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>41</td>
<td>0.000000</td>
<td>4.500000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>

SURFACE SURF
TYPE QUAD
ELEMENTS

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>19</td>
<td>2</td>
<td>17</td>
<td>1</td>
<td>18</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>21</td>
<td>4</td>
<td>20</td>
<td>1</td>
<td>17</td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>5</td>
<td>23</td>
<td>6</td>
<td>24</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>24</td>
<td>6</td>
<td>25</td>
<td>7</td>
<td>26</td>
<td>4</td>
</tr>
<tr>
<td>5</td>
<td>27</td>
<td>8</td>
<td>28</td>
<td>9</td>
<td>29</td>
<td>6</td>
</tr>
<tr>
<td>6</td>
<td>29</td>
<td>9</td>
<td>30</td>
<td>10</td>
<td>31</td>
<td>7</td>
</tr>
<tr>
<td>7</td>
<td>32</td>
<td>11</td>
<td>33</td>
<td>12</td>
<td>34</td>
<td>9</td>
</tr>
<tr>
<td>8</td>
<td>34</td>
<td>12</td>
<td>35</td>
<td>13</td>
<td>36</td>
<td>10</td>
</tr>
<tr>
<td>9</td>
<td>37</td>
<td>14</td>
<td>38</td>
<td>15</td>
<td>39</td>
<td>12</td>
</tr>
<tr>
<td>10</td>
<td>39</td>
<td>15</td>
<td>40</td>
<td>16</td>
<td>41</td>
<td>13</td>
</tr>
</tbody>
</table>

NORMAL 1 +

**BCSET
ID DISP3
VALUE
GMR GMR1
SURFACE SURF
ELEMENTS 1 2 3 4
DISP 3
SPLIST ALL
T 1 1.

$ END OF DATA
FORCED VIBRATION EXAMPLE PROBLEM FORC601 / Selected Output

**JOB TITLE:** SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)

**LOAD CALCULATION AT EXCITING FREQUENCY =** 0.001000

**LOADS FOR REGION GMR1**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>7</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>8</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>9</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>10</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

**LOAD BALANCE** 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.1797E+01 0.1455E-02

**JOB TITLE:** SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)

**LOAD CALCULATION AT EXCITING FREQUENCY =** 1.500000

**LOADS FOR REGION GMR1**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>4</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>7</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>8</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>9</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>10</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

**LOAD BALANCE** 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.1131E+01 0.2806E+01

BEST3D User Manual

April, 1989

Page 6.31
EXAMPLE PROBLEM: FREE601

ANALYSIS TYPE: 3-D FREE VIBRATION

PROBLEM DESCRIPTION:
DETERMINATION OF THE NATURAL FREQUENCY OF A SQUARE CANTILEVER
(1*1*6.5), YOUNG'S MODULUS=10000, DENSITY=1.0, POISSON'S RATIO=0.

BOUNDARY ELEMENT MODEL:
TEN 8-NODED QUADRATIC SURFACE ELEMENTS, ONE ON EACH END AND
TWO ON EACH SIDE. GLOBAL BOUNDARY CONDITIONS SPECIFIED EVERYWHERE.

REFERENCE FOR COMPARED SOLUTION:
S. AHMAD AND P.K. BANERJEE, FREE VIBRATION ANALYSIS BY BEM USING
PARTICULAR INTEGRALS, JOURNAL OF ENGINEERING MECHANICS, ASCE,

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY BY PRESENT MESH</th>
<th>BEST(HZ)</th>
<th>FEM RESULT(HZ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>.39055</td>
<td>0.368</td>
<td>0.378</td>
</tr>
<tr>
<td>2</td>
<td>.39055</td>
<td>0.368</td>
<td>0.378</td>
</tr>
<tr>
<td>3</td>
<td>2.3424</td>
<td>2.214</td>
<td>2.188</td>
</tr>
</tbody>
</table>

RUN TIME:
13 X BASE PROBLEM

MISCELLANEOUS:
FREE VIBRATION EXAMPLE PROBLEM FREE601 / Geometry

BEST3D User Manual
April, 1989
Page 6.33
**CASE
TITLE FREE VIBRATION OF A 3-D CANTILEVER (1*1*6.5) BY 2 REGION MESH.
FREE 4

**MATE
ID MAT1
TEMP 70.
POIS 0.000001
EMOD 10000.
DENSITY 1.0

**GMR
ID GMK1
MAT MAT1
POINTS
1 1.0 0.0 1.0
2 1.0 0.0 0.5
3 1.0 0.0 0.0
4 0.5 0.0 0.0
5 0.0 0.0 0.0
6 0.0 0.0 0.5
7 0.0 0.0 1.0
8 0.5 0.0 1.0
9 1.0 0.8125 1.0
10 1.0 0.8125 0.0
11 0.0 0.8125 0.0
12 0.0 0.8125 1.0
13 1.0 1.625 1.0
14 1.0 1.625 0.5
15 1.0 1.625 0.0
16 0.5 1.625 0.0
17 0.0 1.625 0.0
18 0.0 1.625 0.5
19 0.0 1.625 1.0
20 0.5 1.625 1.0
21 1.0 2.4375 1.0
22 1.0 2.4375 0.0
23 0.0 2.4375 0.0
24 0.0 2.4375 1.0
25 1.0 3.25 1.0
26 1.0 3.25 0.5
27 1.0 3.25 0.0
28 0.5 3.25 0.0
29 0.0 3.25 0.0
30 0.0 3.25 0.5
31 0.0 3.25 1.0
32 0.5 3.25 1.0

SURFACE SURF1
TYPE QUAD
ELEMENTS
1 1 2 3 4 5 6 7 8
2 25 26 27 28 29 30 31 32
**FREE VIBRATION EXAMPLE PROBLEM FREE801 / Input Data**

<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>10</td>
<td>15</td>
<td>14</td>
<td>13</td>
</tr>
<tr>
<td>4</td>
<td>13</td>
<td>14</td>
<td>15</td>
<td>22</td>
<td>27</td>
<td>26</td>
<td>25</td>
</tr>
<tr>
<td>5</td>
<td>8</td>
<td>1</td>
<td>9</td>
<td>13</td>
<td>20</td>
<td>19</td>
<td>12</td>
</tr>
<tr>
<td>6</td>
<td>19</td>
<td>20</td>
<td>13</td>
<td>21</td>
<td>28</td>
<td>32</td>
<td>31</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>6</td>
<td>1</td>
<td>11</td>
<td>17</td>
<td>18</td>
<td>19</td>
</tr>
<tr>
<td>8</td>
<td>19</td>
<td>18</td>
<td>17</td>
<td>23</td>
<td>29</td>
<td>30</td>
<td>31</td>
</tr>
<tr>
<td>9</td>
<td>5</td>
<td>4</td>
<td>3</td>
<td>10</td>
<td>15</td>
<td>16</td>
<td>17</td>
</tr>
<tr>
<td>10</td>
<td>17</td>
<td>16</td>
<td>15</td>
<td>22</td>
<td>27</td>
<td>28</td>
<td>29</td>
</tr>
</tbody>
</table>

NORMAL 1

**GMR**

**ID GMR2**

**MAT MAT1**

**POINTS**

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>201</td>
<td>1.0</td>
<td>3.25</td>
<td>1.0</td>
</tr>
<tr>
<td>202</td>
<td>1.0</td>
<td>3.25</td>
<td>0.5</td>
</tr>
<tr>
<td>203</td>
<td>1.0</td>
<td>3.25</td>
<td>0.0</td>
</tr>
<tr>
<td>204</td>
<td>0.5</td>
<td>3.25</td>
<td>0.0</td>
</tr>
<tr>
<td>205</td>
<td>0.0</td>
<td>3.25</td>
<td>0.0</td>
</tr>
<tr>
<td>206</td>
<td>0.0</td>
<td>3.25</td>
<td>0.5</td>
</tr>
<tr>
<td>207</td>
<td>0.0</td>
<td>3.25</td>
<td>1.0</td>
</tr>
<tr>
<td>208</td>
<td>0.5</td>
<td>3.25</td>
<td>1.0</td>
</tr>
<tr>
<td>209</td>
<td>1.0</td>
<td>4.0625</td>
<td>1.0</td>
</tr>
<tr>
<td>210</td>
<td>1.0</td>
<td>4.0625</td>
<td>0.0</td>
</tr>
<tr>
<td>211</td>
<td>0.0</td>
<td>4.0625</td>
<td>0.0</td>
</tr>
<tr>
<td>212</td>
<td>0.0</td>
<td>4.0625</td>
<td>1.0</td>
</tr>
<tr>
<td>213</td>
<td>1.0</td>
<td>4.875</td>
<td>1.0</td>
</tr>
<tr>
<td>214</td>
<td>1.0</td>
<td>4.875</td>
<td>0.5</td>
</tr>
<tr>
<td>215</td>
<td>1.0</td>
<td>4.875</td>
<td>0.0</td>
</tr>
<tr>
<td>216</td>
<td>0.5</td>
<td>4.875</td>
<td>0.0</td>
</tr>
<tr>
<td>217</td>
<td>0.0</td>
<td>4.875</td>
<td>0.0</td>
</tr>
<tr>
<td>218</td>
<td>0.0</td>
<td>4.875</td>
<td>0.5</td>
</tr>
<tr>
<td>219</td>
<td>0.0</td>
<td>4.875</td>
<td>1.0</td>
</tr>
<tr>
<td>220</td>
<td>0.5</td>
<td>4.875</td>
<td>1.0</td>
</tr>
<tr>
<td>221</td>
<td>1.0</td>
<td>5.6875</td>
<td>1.0</td>
</tr>
<tr>
<td>222</td>
<td>1.0</td>
<td>5.6875</td>
<td>0.0</td>
</tr>
<tr>
<td>223</td>
<td>0.0</td>
<td>5.6875</td>
<td>0.0</td>
</tr>
<tr>
<td>224</td>
<td>0.0</td>
<td>5.6875</td>
<td>1.0</td>
</tr>
<tr>
<td>225</td>
<td>1.0</td>
<td>6.6</td>
<td>1.0</td>
</tr>
<tr>
<td>226</td>
<td>1.0</td>
<td>6.6</td>
<td>0.5</td>
</tr>
<tr>
<td>227</td>
<td>1.0</td>
<td>6.6</td>
<td>0.0</td>
</tr>
<tr>
<td>228</td>
<td>0.5</td>
<td>6.6</td>
<td>0.0</td>
</tr>
<tr>
<td>229</td>
<td>0.0</td>
<td>6.6</td>
<td>0.0</td>
</tr>
<tr>
<td>230</td>
<td>0.0</td>
<td>6.6</td>
<td>0.5</td>
</tr>
<tr>
<td>231</td>
<td>0.0</td>
<td>6.6</td>
<td>1.0</td>
</tr>
<tr>
<td>232</td>
<td>0.5</td>
<td>6.6</td>
<td>1.0</td>
</tr>
</tbody>
</table>

**SURFACE SURF1**

**TYPE QUAD**

**ELEMENTS**

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>201</td>
<td>201</td>
<td>202</td>
<td>203</td>
<td>204</td>
<td>205</td>
<td>206</td>
</tr>
<tr>
<td>202</td>
<td>225</td>
<td>226</td>
<td>227</td>
<td>228</td>
<td>229</td>
<td>230</td>
</tr>
<tr>
<td>203</td>
<td>201</td>
<td>202</td>
<td>203</td>
<td>210</td>
<td>215</td>
<td>214</td>
</tr>
<tr>
<td>204</td>
<td>213</td>
<td>214</td>
<td>215</td>
<td>222</td>
<td>227</td>
<td>226</td>
</tr>
<tr>
<td>205</td>
<td>207</td>
<td>208</td>
<td>201</td>
<td>209</td>
<td>213</td>
<td>220</td>
</tr>
<tr>
<td>206</td>
<td>219</td>
<td>220</td>
<td>213</td>
<td>221</td>
<td>225</td>
<td>232</td>
</tr>
<tr>
<td>207</td>
<td>207</td>
<td>206</td>
<td>205</td>
<td>211</td>
<td>217</td>
<td>218</td>
</tr>
<tr>
<td>208</td>
<td>219</td>
<td>218</td>
<td>217</td>
<td>223</td>
<td>229</td>
<td>230</td>
</tr>
</tbody>
</table>
**INTERFACE
$ INTERFACE CONDITION
$ FIRST REGION
  GMR GMRI
  SURFACE SURF1
  ELEMENT 2
$ SECOND REGION
  GMR GMRI
  SURFACE SURF1
  ELEMENT 201

**BCSET
  ID  BCS1
  VALUE
  GMR GMRI
  SURFACE SURF1
  ELEMENT 1
  DISP 1
  SPLIST ALL
  T 1 0.0
  DISP 2
  SPLIST ALL
  T 1 0.0
  DISP 3
  SPLIST ALL
  T 1 0.0

$  
$ END OF DATA
FREE VIBRATION EXAMPLE PROBLEM FREE801 / Selected Output

**BEST NATURAL FREQUENCY ANALYSIS**

**JOB TITLE:** FREE VIBRATION OF A 3-D CANTILEVER (1x1x6.6) BY 2 REGION MESH.

4 CONVERGED MODE SHAPES IN 9 ITERATIONS

<table>
<thead>
<tr>
<th>NODE NUMBER</th>
<th>FREQUENCY HERTZ</th>
<th>FREQUENCY RAD/SEC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.39056E+00</td>
<td>0.24639E+01</td>
</tr>
<tr>
<td>2</td>
<td>0.39056E+00</td>
<td>0.24639E+01</td>
</tr>
<tr>
<td>3</td>
<td>0.23424E+01</td>
<td>0.14718E+02</td>
</tr>
<tr>
<td>4</td>
<td>0.61734E+01</td>
<td>0.38789E+02</td>
</tr>
</tbody>
</table>

**JOB TITLE:** FREE VIBRATION OF A 3-D CANTILEVER (1x1x6.6) BY 2 REGION MESH.

DEFLECTED SHAPE FOR NODE 1 AT EIGEN FREQUENCY 3.90656E-01 HERTZ OF REGION = K61

<table>
<thead>
<tr>
<th>ELEMENT NO.</th>
<th>X-DISPL.</th>
<th>Y-DISPL.</th>
<th>Z-DISPL.</th>
<th>X-TRAC.</th>
<th>Y TRAC.</th>
<th>Z TRAC.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.10331E+01</td>
<td>0.86608E+02</td>
<td>-0.57112E+00</td>
</tr>
<tr>
<td>1 8</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.42060E+01</td>
<td>-0.14965E+02</td>
<td>0.34536E+00</td>
</tr>
<tr>
<td>1 7</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.12496E+01</td>
<td>-0.11635E+02</td>
<td>0.90665E+00</td>
</tr>
<tr>
<td>1 6</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.23586E+01</td>
<td>-0.10224E+02</td>
<td>0.61640E+00</td>
</tr>
<tr>
<td>1 5</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.10331E+01</td>
<td>-0.86608E+02</td>
<td>-0.57126E+00</td>
</tr>
<tr>
<td>1 4</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.42060E+01</td>
<td>0.14965E+02</td>
<td>0.34536E+00</td>
</tr>
<tr>
<td>1 3</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.12496E+01</td>
<td>0.11635E+02</td>
<td>0.90665E+00</td>
</tr>
<tr>
<td>1 2</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>-0.23586E+01</td>
<td>0.10224E+02</td>
<td>0.61640E+00</td>
</tr>
</tbody>
</table>
EXAMPLE PROBLEM: HEAT602

ANALYSIS TYPE: STEADY-STATE HEAT CONDUCTION
3-D ANALYSIS

PROBLEM DESCRIPTION:
LINEAR VARIATION OF TEMPERATURE IN A CUBE. INTERIOR SAMPLING
POINTS INCLUDED FOR OUTPUT OF TEMPERATURE AND FLUX.

BOUNDARY ELEMENT MODEL:
SINGLE GMR, SIX SURFACE ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
CARSLAW AND JAEGGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:
GMR  NODE       (TEMPERATURE)     BEST
      22           25.00          25.00

GMR  NODE       (FLUX-Y)         BEST
      22          -2500.0        -2500.2

RUN TIME:
0.6 X BASE PROBLEM

MISCELLANEOUS:
INTEGRATION FILES ARE SAVED FOR RESTART.
BINARY RESULTS FILE IS WRITTEN.
**CASE**
**TITLE** ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
HEAT
RESTART WRITE
BINARY

**MATE**
ID MAT1
TEMP 70.0
COND 25.

**GHR**
ID GMR1
MAT MAT1
TREF 70.0

POINTS
1 0.0 0.0 0.0
2 0.0 0.5 0.0
3 0.0 1.0 0.0
4 0.5 0.0 0.0
5 0.5 1.0 0.0
6 1.0 0.0 0.0
7 1.0 0.5 0.0
8 1.0 1.0 0.0
9 0.0 0.0 0.0
10 0.0 1.0 0.0
11 1.0 0.0 0.0
12 1.0 1.0 0.0
13 0.0 0.0 1.0
14 0.0 0.5 1.0
15 0.0 1.0 1.0
16 0.5 0.0 1.0
17 0.5 1.0 1.0
18 1.0 0.0 1.0
19 1.0 0.5 1.0
20 1.0 1.0 1.0

SURFACE SURF1
TYPE QUAD

ELEMENTS
1 1 2 3 5 8 7 6 4
2 13 14 15 17 20 19 18 16
3 1 2 3 10 15 14 13 9
4 6 7 8 12 20 19 18 11
5 1 4 6 11 18 16 13 9
6 3 5 8 12 20 17 15 10
NORMAL 1 +

SAMPLING POINTS
21 0.5 0.0 0.5
22 0.5 0.25 0.5
23 0.5 0.5 0.5
24 0.5 0.75 0.5
25 0.5 1.0 0.5
**BCSET
  ID TRAC15
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 5
  TEMP
  SPLIST ALL
  T 1  0.0
**BCSET
  ID TRAC16
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 6
  TEMP
  SPLIST ALL
  T 1  100.0

$  ZERO FLUX BOUNDARY CONDITION ASSUMED ON ELEMENTS  1 THRU 4
$  END OF DATA
HEAT TRANSFER EXAMPLE PROBLEM HEAT802 / Selected Output

**JOB TITLE:** ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS

**HEAT RATE CALCULATION AT TIME = 1.000000**

**HEAT RATE FOR REGION GNR1**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>HEAT RATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>4</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.26000E+04</td>
</tr>
<tr>
<td>6</td>
<td>-0.26000E+04</td>
</tr>
</tbody>
</table>

TOTAL HEAT RATE 0.00000E+00

**JOB TITLE:** ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS

**INTERIOR TEMPERATURE AT TIME = 1.000000 FOR REGION = GNR1**

<table>
<thead>
<tr>
<th>NODE</th>
<th>TEMPERATURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>22</td>
<td>0.249999E+02</td>
</tr>
<tr>
<td>23</td>
<td>0.560000E+02</td>
</tr>
<tr>
<td>24</td>
<td>0.750000E+02</td>
</tr>
<tr>
<td>25</td>
<td>0.100000E+03</td>
</tr>
</tbody>
</table>

**JOB TITLE:** ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS

**INTERIOR FLUX AT TIME = 1.000000 FOR REGION = GNR1**

<table>
<thead>
<tr>
<th>NODE</th>
<th>FLUX-X</th>
<th>FLUX-Y</th>
<th>FLUX-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.00000E+00</td>
<td>-0.249994E+04</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>22</td>
<td>0.142300E-05</td>
<td>-0.250018E+04</td>
<td>0.143300E-05</td>
</tr>
<tr>
<td>23</td>
<td>0.150488E-10</td>
<td>-0.250005E+04</td>
<td>0.149831E-10</td>
</tr>
<tr>
<td>24</td>
<td>-0.143297E-05</td>
<td>-0.250013E+04</td>
<td>-0.143297E-05</td>
</tr>
<tr>
<td>25</td>
<td>0.000000E+00</td>
<td>-0.249994E+04</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>

END OF ANALYSIS
EXAMPLE PROBLEM: HEAT604

ANALYSIS TYPE: STEADY-STATE HEAT CONDUCTION
3-D ANALYSIS, MIXTURE OF LINEAR AND QUADRATIC ELEMENTS

PROBLEM DESCRIPTION:
LINEAR VARIATION OF TEMPERATURE IN A CUBE. INTERIOR SAMPLING POINTS INCLUDED FOR OUTPUT OF TEMPERATURE AND FLUX.

BOUNDARY ELEMENT MODEL:
INCLUDED ARE EIGHT AND NINE NODED SURFACE ELEMENTS, WITH BOTH LINEAR AND QUADRATIC FUNCTIONAL VARIATION OF THE FIELD VARIABLES.

REFERENCE FOR ANALYTICAL SOLUTION:
CARSLAW AND JAEGGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>GMR</th>
<th>NODE</th>
<th>(TEMPERATURE)</th>
<th>ANALYTICAL</th>
<th>BEST</th>
</tr>
</thead>
<tbody>
<tr>
<td>GMR1</td>
<td>101</td>
<td></td>
<td>0.500</td>
<td>0.500</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>GMR</th>
<th>NODE</th>
<th>(FLUX-Z)</th>
<th>ANALYTICAL</th>
<th>BEST</th>
</tr>
</thead>
<tbody>
<tr>
<td>GMR1</td>
<td>102</td>
<td></td>
<td>-2.000</td>
<td>-2.000</td>
</tr>
</tbody>
</table>

RUN TIME:
0.5 X BASE PROBLEM

MISCELLANEOUS:
SEPARATE SURFACES MUST BE USED FOR THE LINEAR AND QUADRATIC ELEMENTS. EVEN THOUGH TWO SURFACES ARE PRESENT, THERE IS ONLY ONE 'DISJOINT BOUNDARY'. CONSEQUENTLY, THERE IS ONLY ONE ELEMENT-SIGN PAIR ON THE 'NORMAL' INPUT LINE.

MATERIAL MAT2 IS NOT UTILIZED.
INTEGRATION FILES ARE SAVED FOR RESTART.
NODES AND ELEMENTS
**CASE**

**TITLE** HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS

$ $ THREE-DIMENSIONAL GEOMETRY (DEFAULT)
$ STEADY-STATE HEAT TRANSFER
$ HEAT
$ SAVE INTEGRATION FT FILES FOR LATER USE
$ RESTART WRITE
$ PRINT INTERIOR TEMP AND FLUX

**MATE**

ID MAT1
$ REFERENCE TEMPERATURE
$ TEMP 0.0
$ THERMAL CONDUCTIVITY
$ COND 1.

**MATE**

ID MAT2
$ REFERENCE TEMPERATURE
$ TEMP 0.0
$ THERMAL CONDUCTIVITY
$ COND 1.

**GMR**

ID GMR1
$ USE MATERIAL MAT1 FOR REGION
$ MAT MAT1
$ REFERENCE TEMPERATURE FOR REGION
$ TREF 0.0

$ POINTS
$ ID X Y Z
1 0.0 0.0 0.0
2 0.0 .25000 0.0
3 0.0 .50000 0.0
4 .25000 0.0 0.0
5 .25000 .50000 0.0
6 .50000 0.0 0.0
7 .50000 .25000 0.0
8 .50000 .50000 0.0
9 0.0 0.0 .25000
10 0.0 .50000 .25000
11 .50000 0.0 .25000
12 .50000 .50000 .25000
13 0.0 0.0 .50000
14 0.0 .25000 .50000
15 0.0 .50000 .50000
16 .25000 0.0 .50000
17 .25000 .50000 .50000
18 .50000 0.0 .50000
HEAT TRANSFER EXAMPLE PROBLEM HEAT604 / Input Data

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>.50000</td>
<td>.25000</td>
</tr>
<tr>
<td>20</td>
<td>.50000</td>
<td>.50000</td>
</tr>
<tr>
<td>21</td>
<td>.25000</td>
<td>.50000</td>
</tr>
<tr>
<td>22</td>
<td>.12500</td>
<td>.50000</td>
</tr>
<tr>
<td>23</td>
<td>.37500</td>
<td>.50000</td>
</tr>
<tr>
<td>24</td>
<td>.37500</td>
<td>.50000</td>
</tr>
<tr>
<td>25</td>
<td>.12500</td>
<td>.50000</td>
</tr>
<tr>
<td>92</td>
<td>.25000</td>
<td>.25000</td>
</tr>
<tr>
<td>94</td>
<td>.50000</td>
<td>.25000</td>
</tr>
</tbody>
</table>

$ SURFACE SURF1
$ USE LINEAR FUNCTIONAL VARIATION
TYPE LINEAR
$ ELEMENTS
$ ID G1 G2 G3 G4 G5 G6 G7 G8 G9
  2  13 14 15 17 20 19 18 16 92
$ SURFACE SURF2
$ USE QUADRATIC FUNCTIONAL VARIATION
TYPE QUADRATIC
$ ELEMENTS
$ ID G1 G2 G3 G4 G5 G6 G7 G8 G9
  1  1  2  3  5  8  7  6  4
  3  1  2  3 10 15 14 13  9
  4  6  7  8 12 20 19 18 11 94
  5  1  4  6 11 18 16 13  9
  6  3  5  8 12 20 17 15 10

$ OUTWARD (RIGHT-HAND) NORMAL OF ELEMENT 1 IS POSITIVE
$ (ONLY ONE DISJOINT BOUNDARY)
NORMAL 1 +

SAMPLING POINTS
101 0.25 0.25 0.25
102 0.25 0.25 0.50
103 0.25 0.10 0.50

$ SPECIFY TEMPERATURE = 0.0 AT Z=0.0 (ELEMENT 1, GMR1)
$ **BCSET
  ID TEMP11
  VALUE
  GMR GMR1
  SURFACE SURF2
  ELEMENTS 1
  TEMP
  SPLIST ALL
  T 1 0.0

$ SPECIFY TEMPERATURE = 1.0 AT Z=0.5 (ELEMENT 2, GMR1)
$ **BCSET
  ID TEMP12
VALUE
GHR GHR1
SURFACE SURF1
ELEMENTS 2
TEMP
  SPLIST ALL
  T  1   1.0

$  REMAINING ELEMENTS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)

$  END OF DATA
HEAT TRANSFER EXAMPLE PROBLEM HEAT604 / Selected Output

JOB TITLE: HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
INTERIOR TEMPERATURE AT TIME = 1.000000 FOR REGION = GR1

<table>
<thead>
<tr>
<th>NODE</th>
<th>TEMPERATURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>0.500000E+00</td>
</tr>
<tr>
<td>102</td>
<td>0.100000E+01</td>
</tr>
<tr>
<td>103</td>
<td>0.100000E+01</td>
</tr>
</tbody>
</table>

JOB TITLE: HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
INTERIOR FLUX AT TIME = 1.000000 FOR REGION = GR1

<table>
<thead>
<tr>
<th>NODE</th>
<th>FLUX-X</th>
<th>FLUX-Y</th>
<th>FLUX-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>0.870418E-07</td>
<td>0.187338E-08</td>
<td>-0.200004E+01</td>
</tr>
<tr>
<td>102</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>-0.200004E+01</td>
</tr>
<tr>
<td>103</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>-0.200004E+01</td>
</tr>
</tbody>
</table>

END OF ANALYSIS
EXAMPLE PROBLEM: HEAT605

ANALYSIS TYPE: TRANSIENT HEAT CONDUCTION
3-D ANALYSIS

PROBLEM DESCRIPTION:
TRANSIENT HEAT FLOW BETWEEN TWO CONCENTRIC CIRCULAR CYLINDERS.
AT TIME ZERO, THE TEMPERATURE OF THE INNER CYLINDER IS ELEVATED
FROM ZERO TO 1.0. THE RESPONSE IS MONITORED AT A NUMBER OF
INTERIOR SAMPLING POINTS.

BOUNDARY ELEMENT MODEL:
The outer cylinder has radius 8.0 and is modeled with 3-D
surface elements. The inner cylinder (radius=1.0) is represented
by hole elements.

REFERENCE FOR ANALYTICAL SOLUTION:
CARSLOW AND JAEGGER (1959), CONDUCTION OF HEAT IN SOLIDS,
PP205-207.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>GMR</th>
<th>NODE</th>
<th>TIME</th>
<th>TEMP (TEMPERATURE)</th>
<th>ANALYTICAL</th>
<th>BEST</th>
</tr>
</thead>
<tbody>
<tr>
<td>GMR1</td>
<td>5005</td>
<td>4.00</td>
<td>0.1105</td>
<td>0.1067</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>8.00</td>
<td>0.1971</td>
<td>0.1948</td>
<td></td>
</tr>
</tbody>
</table>

RUN TIME:
29 X BASE PROBLEM

MISCELLANEOUS:
The cylinders have a length of 40.0 units. Consequently, nearly
planar conditions exist at z=20.0, where the sampling points
are located.
Total heat rate calculation does not include contribution
from hole elements.
A binary output file is saved.
**CASE**
**TITLE** TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT

$ THREE-DIMENSIONAL GEOMETRY (DEFAULT)
$ TRANSIENT HEAT TRANSFER
HEAT TRANSIENT 8 1.0

**MATE**
ID MAT1
$ REFERENCE TEMPERATURE
TEMP 0.0
$ THERMAL CONDUCTIVITY
COND 1.0
$ DENSITY
DENS 1.0
$ SPECIFIC HEAT
SPEC 1.0

**GMR**
ID GMR1
$ USE MATERIAL MAT1 FOR REGION
MAT MAT1
$ REFERENCE TEMPERATURE FOR REGION
TREF 0.0

$ POINTS

<table>
<thead>
<tr>
<th>ID</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>101</td>
<td>4.00000</td>
<td>0.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>103</td>
<td>2.82843</td>
<td>2.82843</td>
<td>0.0</td>
</tr>
<tr>
<td>105</td>
<td>0.00000</td>
<td>4.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>107</td>
<td>-2.82843</td>
<td>2.82843</td>
<td>0.0</td>
</tr>
<tr>
<td>109</td>
<td>-4.00000</td>
<td>0.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>111</td>
<td>-2.82843</td>
<td>-2.82843</td>
<td>0.0</td>
</tr>
<tr>
<td>113</td>
<td>0.00000</td>
<td>-4.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>115</td>
<td>2.82843</td>
<td>-2.82843</td>
<td>0.0</td>
</tr>
<tr>
<td>201</td>
<td>8.00000</td>
<td>0.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>202</td>
<td>7.39104</td>
<td>3.06147</td>
<td>0.0</td>
</tr>
<tr>
<td>203</td>
<td>5.65685</td>
<td>5.65685</td>
<td>0.0</td>
</tr>
<tr>
<td>204</td>
<td>3.06147</td>
<td>7.39104</td>
<td>0.0</td>
</tr>
<tr>
<td>205</td>
<td>0.00000</td>
<td>8.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>206</td>
<td>-3.06147</td>
<td>7.39104</td>
<td>0.0</td>
</tr>
<tr>
<td>207</td>
<td>-5.65685</td>
<td>5.65685</td>
<td>0.0</td>
</tr>
<tr>
<td>208</td>
<td>-7.39104</td>
<td>3.06147</td>
<td>0.0</td>
</tr>
<tr>
<td>209</td>
<td>-8.00000</td>
<td>0.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>210</td>
<td>-7.39104</td>
<td>-3.06147</td>
<td>0.0</td>
</tr>
<tr>
<td>211</td>
<td>-5.65685</td>
<td>-5.65685</td>
<td>0.0</td>
</tr>
<tr>
<td>212</td>
<td>-3.06147</td>
<td>-7.39104</td>
<td>0.0</td>
</tr>
<tr>
<td>213</td>
<td>0.00000</td>
<td>-8.00000</td>
<td>0.0</td>
</tr>
<tr>
<td>214</td>
<td>3.06147</td>
<td>-7.39104</td>
<td>0.0</td>
</tr>
</tbody>
</table>
HEAT TRANSFER EXAMPLE PROBLEM HEAT005 / Input Data

<table>
<thead>
<tr>
<th>ID</th>
<th>G1</th>
<th>G2</th>
<th>G3</th>
<th>G4</th>
<th>G5</th>
<th>G6</th>
<th>G7</th>
<th>G8</th>
</tr>
</thead>
<tbody>
<tr>
<td>215</td>
<td>5.65685</td>
<td>-5.65685</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>216</td>
<td>7.39104</td>
<td>-3.06147</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1201</td>
<td>8.00000</td>
<td>0.00000</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1203</td>
<td>5.65685</td>
<td>5.65685</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1205</td>
<td>0.00000</td>
<td>8.00000</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1207</td>
<td>-5.65685</td>
<td>5.65685</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1209</td>
<td>-8.00000</td>
<td>0.00000</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1211</td>
<td>-5.65685</td>
<td>-5.65685</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1213</td>
<td>0.00000</td>
<td>-8.00000</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1215</td>
<td>5.65685</td>
<td>-5.65685</td>
<td>20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2101</td>
<td>4.00000</td>
<td>0.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2103</td>
<td>2.82843</td>
<td>2.82843</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2105</td>
<td>0.00000</td>
<td>4.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2107</td>
<td>-2.82843</td>
<td>2.82843</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2109</td>
<td>-4.00000</td>
<td>0.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2111</td>
<td>-2.82843</td>
<td>-2.82843</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2113</td>
<td>0.00000</td>
<td>-4.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2115</td>
<td>2.82843</td>
<td>-2.82843</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2201</td>
<td>8.00000</td>
<td>0.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2202</td>
<td>7.39104</td>
<td>3.06147</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2203</td>
<td>5.65685</td>
<td>5.65685</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2204</td>
<td>3.06147</td>
<td>7.39104</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2205</td>
<td>0.00000</td>
<td>8.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2206</td>
<td>-3.06147</td>
<td>7.39104</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2207</td>
<td>-5.65685</td>
<td>5.65685</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2208</td>
<td>-7.39104</td>
<td>3.06147</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2209</td>
<td>-8.00000</td>
<td>0.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2210</td>
<td>-7.39104</td>
<td>-3.06147</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2211</td>
<td>-5.65685</td>
<td>-5.65685</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2212</td>
<td>-3.06147</td>
<td>-7.39104</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2213</td>
<td>0.00000</td>
<td>-8.00000</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2214</td>
<td>3.06147</td>
<td>-7.39104</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2215</td>
<td>5.65685</td>
<td>-5.65685</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2216</td>
<td>7.39104</td>
<td>-3.06147</td>
<td>40.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

$ SURFACE SURF1
$ USE QUADRATIC FUNCTIONAL VARIATION
$ TYPE QUAD
$ DISC AT NEAR END OF CYLINDER
$ ELEMENTS
$ID  G1  G2  G3  G4  G5  G6  G7  G8
 1  1  101  201  202  203  103
 2  1  103  203  204  205  105
 3  1  105  205  206  207  107
 4  1  107  207  208  209  109
 5  1  109  209  210  211  111
 6  1  111  211  212  213  113
 7  1  113  213  214  215  115
 8  1  115  215  216  201  101
$ SURFACE OF OUTER CYLINDER
$
### Input Data

<table>
<thead>
<tr>
<th>ID</th>
<th>1001</th>
<th>1002</th>
<th>1003</th>
<th>1004</th>
<th>1005</th>
<th>1006</th>
<th>1007</th>
<th>1008</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>201</td>
<td>202</td>
<td>203</td>
<td>204</td>
<td>205</td>
<td>206</td>
<td>207</td>
<td>208</td>
</tr>
<tr>
<td></td>
<td>209</td>
<td>210</td>
<td>211</td>
<td>212</td>
<td>213</td>
<td>214</td>
<td>215</td>
<td>216</td>
</tr>
<tr>
<td></td>
<td>201</td>
<td>202</td>
<td>203</td>
<td>204</td>
<td>205</td>
<td>206</td>
<td>207</td>
<td>208</td>
</tr>
</tbody>
</table>

#### Disc at Far End of Cylinder

<table>
<thead>
<tr>
<th>ID</th>
<th>2001</th>
<th>2002</th>
<th>2003</th>
<th>2004</th>
<th>2005</th>
<th>2006</th>
<th>2007</th>
<th>2008</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2001</td>
<td>2101</td>
<td>2201</td>
<td>2202</td>
<td>2203</td>
<td>2103</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2002</td>
<td>2103</td>
<td>2203</td>
<td>2204</td>
<td>2205</td>
<td>2105</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2003</td>
<td>2105</td>
<td>2205</td>
<td>2206</td>
<td>2207</td>
<td>2107</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2004</td>
<td>2107</td>
<td>2207</td>
<td>2208</td>
<td>2209</td>
<td>2109</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2005</td>
<td>2109</td>
<td>2209</td>
<td>2210</td>
<td>2211</td>
<td>2111</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2006</td>
<td>2111</td>
<td>2211</td>
<td>2212</td>
<td>2213</td>
<td>2113</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2007</td>
<td>2113</td>
<td>2213</td>
<td>2214</td>
<td>2215</td>
<td>2115</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2008</td>
<td>2115</td>
<td>2215</td>
<td>2216</td>
<td>2201</td>
<td>2101</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Outward (Right-Hand) Normal of Element

1 is Negative Normal

#### Surface of Inner Cylinder

<table>
<thead>
<tr>
<th>ID</th>
<th>9001</th>
<th>9002</th>
<th>9003</th>
<th>9004</th>
<th>9005</th>
<th>9006</th>
<th>9007</th>
<th>9008</th>
<th>9009</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

#### Use Quadratic Functional Variation for 3-Noded Hole Elements

**Type Quad**

<table>
<thead>
<tr>
<th>ID</th>
<th>9001</th>
<th>9002</th>
<th>9003</th>
<th>9004</th>
<th>9005</th>
<th>9006</th>
<th>9007</th>
<th>9008</th>
<th>9009</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td></td>
<td>9001</td>
<td>9002</td>
<td>9003</td>
<td>9004</td>
<td>9005</td>
<td>9006</td>
<td>9007</td>
<td>9008</td>
<td>9009</td>
</tr>
</tbody>
</table>

#### Sampling Points

<table>
<thead>
<tr>
<th>ID</th>
<th>5001</th>
<th>5002</th>
<th>5003</th>
<th>5004</th>
<th>5005</th>
<th>5006</th>
<th>5007</th>
<th>5008</th>
<th>5009</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.7</td>
<td>2.4</td>
<td>3.1</td>
<td>3.8</td>
<td>4.5</td>
<td>5.2</td>
<td>5.9</td>
<td>6.6</td>
<td>7.3</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
<td>20.0</td>
</tr>
</tbody>
</table>
HEAT TRANSFER EXAMPLE PROBLEM HEAT805 / Input Data

$**BCSET
ID TEMP1
$  VALUE BOUNDARY CONDITION SET
  VALUE
  GMR GMRI
  HOLE
    ELEMENTS 9001 9002 9003 9004
  TEMP
    SPLIST ALL
    T 1  1.0

$**BCSET
ID TEMP2
$  VALUE BOUNDARY CONDITION SET
  VALUE
  GMR GMRI
  SURFACE SURFI
    ELEMENTS 1001 1002 1003 1004
    ELEMENTS 1005 1006 1007 1008
  TEMP
    SPLIST ALL
    T 1  0.0

$ ENDS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)

$ END OF DATA
HEAT TRANSFER EXAMPLE PROBLEM HEAT605 / Selected Output

JOB TITLE: TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT

HOLE ELEMENT SOLUTION AT TIME = 8.000000 FOR REGION = ONR1

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>NODE NO.</th>
<th>TEMPERATURE</th>
<th>FLUX</th>
</tr>
</thead>
<tbody>
<tr>
<td>9001</td>
<td>9001</td>
<td>0.10000E+01</td>
<td>-0.85688E+00</td>
</tr>
<tr>
<td>9001</td>
<td>9002</td>
<td>0.10000E+01</td>
<td>-0.64049E+00</td>
</tr>
<tr>
<td>9001</td>
<td>9003</td>
<td>0.10000E+01</td>
<td>-0.58113E+00</td>
</tr>
<tr>
<td>9002</td>
<td>9003</td>
<td>0.10000E+01</td>
<td>-0.58113E+00</td>
</tr>
<tr>
<td>9002</td>
<td>9004</td>
<td>0.10000E+01</td>
<td>-0.67267E+00</td>
</tr>
<tr>
<td>9002</td>
<td>9005</td>
<td>0.10000E+01</td>
<td>-0.67366E+00</td>
</tr>
<tr>
<td>9003</td>
<td>9005</td>
<td>0.10000E+01</td>
<td>-0.57366E+00</td>
</tr>
<tr>
<td>9003</td>
<td>9006</td>
<td>0.10000E+01</td>
<td>-0.67267E+00</td>
</tr>
<tr>
<td>9003</td>
<td>9007</td>
<td>0.10000E+01</td>
<td>-0.58113E+00</td>
</tr>
<tr>
<td>9004</td>
<td>9007</td>
<td>0.10000E+01</td>
<td>-0.58113E+00</td>
</tr>
<tr>
<td>9004</td>
<td>9008</td>
<td>0.10000E+01</td>
<td>-0.64049E+00</td>
</tr>
<tr>
<td>9004</td>
<td>9009</td>
<td>0.10000E+01</td>
<td>-0.58113E+00</td>
</tr>
</tbody>
</table>

JOB TITLE: TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT

INTERIOR TEMPERATURE AT TIME = 8.000000 FOR REGION = ONR1

<table>
<thead>
<tr>
<th>NODE</th>
<th>TEMPERATURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>5001</td>
<td>0.700617E+00</td>
</tr>
<tr>
<td>5002</td>
<td>0.609723E+00</td>
</tr>
<tr>
<td>5003</td>
<td>0.373628E+00</td>
</tr>
<tr>
<td>5004</td>
<td>0.272058E+00</td>
</tr>
<tr>
<td>5005</td>
<td>0.194838E+00</td>
</tr>
<tr>
<td>5006</td>
<td>0.135317E+00</td>
</tr>
<tr>
<td>5007</td>
<td>0.890075E-01</td>
</tr>
<tr>
<td>5008</td>
<td>0.638336E-01</td>
</tr>
<tr>
<td>5009</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>

END OF ANALYSIS

BEST3D User Manual April, 1989 Page 6.55
EXAMPLE PROBLEM: PLAS602

ANALYSIS TYPE: STRESS ANALYSIS
3-D, ELASTOPLASTIC ANALYSIS, VON MISES MATERIAL MODEL WITH
PERFECTLY PLASTIC BEHAVIOR, VARIABLE STIFFNESS ALGORITHM.

PROBLEM DESCRIPTION:
THICK CYLINDER SUBJECTED TO INCREASING INTERNAL PRESSURE,
UNDER PLANE STRAIN CONDITIONS.

BOUNDARY ELEMENT MODEL:
15-DEGREE SEGMENT, WITH 1:2 RATIO, 1-GMR, 18 BOUNDARY ELEMENTS, 4 (TWENTY-NODED) ISOPARAMETRIC CELLS.

REFERENCE FOR ANALYTICAL SOLUTION:
HILL,R.(1950) THE MATHEMATICAL THEORY OF PLASTICITY, PG.106-111
PROVIDES ANALYTICAL SOLUTION TO THIS PROBLEM.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>TIME</th>
<th>NODE</th>
<th>BEST SOLN.</th>
<th>ANALYTICAL SOLN.</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>10</td>
<td>1.2925</td>
<td>1.2921</td>
</tr>
<tr>
<td>4.0</td>
<td>10</td>
<td>1.5905</td>
<td>1.5855</td>
</tr>
<tr>
<td>10.0</td>
<td>10</td>
<td>2.7196</td>
<td>2.7279</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TIME</th>
<th>NODE</th>
<th>BEST SOLN.</th>
<th>ANALYTICAL SOLN.</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>20</td>
<td>428.1</td>
<td>433.0</td>
</tr>
<tr>
<td>2.0</td>
<td>10</td>
<td>184.6</td>
<td>183.2</td>
</tr>
<tr>
<td>2.0</td>
<td>22</td>
<td>257.5</td>
<td>250.6</td>
</tr>
<tr>
<td>10.0</td>
<td>20</td>
<td>284.7</td>
<td>280.4</td>
</tr>
<tr>
<td>10.0</td>
<td>10</td>
<td>388.9</td>
<td>389.7</td>
</tr>
<tr>
<td>10.0</td>
<td>22</td>
<td>542.2</td>
<td>541.3</td>
</tr>
</tbody>
</table>

RUN TIME:
31 X BASE PROBLEM

MISCELLANEOUS:
LOAD-DISPLACEMENT RESPONSE (NORMALISED RADIAL DISPLACEMENT AS)
(A FUNCTION OF INTERNAL PRESSURE) AND DISTRIBUTION OF
CIRCUMFERENTIAL STRESS THROUGH THE THICKNESS OF THE CYLINDER
ARE THE PRIMARY INTERESTS OF THIS ANALYSIS.
**CASE**

**TITLE THICK CYLINDER UNDER INTERNAL PRESSURE**

PLASTICITY DIRECT $\$ SPECIFIES VARIABLE STIFFNESS PLASTICITY TIMES 1. 2. 3. 4. 5. 6. 7. 8. 9. 10.

**MATE**

ID MAT1

TEMP 70.0
EMOD 2600.0
POIS 0.3
INELASTIC
VON MISES
YIELD 600.0
CURVE 600.0 10.0

$\$ (PERFECTLY PLASTIC MATERIAL WITH ZERO STRAIN HARDENING)

**GMR**

ID GMR1
MAT MAT1
TREF 70.0
POINTS

<table>
<thead>
<tr>
<th>POINT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.9144</td>
<td>-1.3053</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>11.1538</td>
<td>-1.4684</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>12.3931</td>
<td>-1.6316</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>13.6324</td>
<td>-1.7947</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>14.8717</td>
<td>-1.9579</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>16.1110</td>
<td>-2.1211</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>17.3503</td>
<td>-2.2842</td>
<td>0.0</td>
</tr>
<tr>
<td>8</td>
<td>18.5896</td>
<td>-2.4474</td>
<td>0.0</td>
</tr>
<tr>
<td>9</td>
<td>19.8289</td>
<td>-2.6105</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>20.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>11</td>
<td>19.8289</td>
<td>2.6105</td>
<td>0.0</td>
</tr>
<tr>
<td>12</td>
<td>18.5896</td>
<td>2.4474</td>
<td>0.0</td>
</tr>
<tr>
<td>13</td>
<td>17.3503</td>
<td>2.2842</td>
<td>0.0</td>
</tr>
<tr>
<td>14</td>
<td>16.1110</td>
<td>2.1211</td>
<td>0.0</td>
</tr>
<tr>
<td>15</td>
<td>14.8717</td>
<td>1.9579</td>
<td>0.0</td>
</tr>
<tr>
<td>16</td>
<td>13.6324</td>
<td>1.7947</td>
<td>0.0</td>
</tr>
<tr>
<td>17</td>
<td>12.3931</td>
<td>1.6316</td>
<td>0.0</td>
</tr>
<tr>
<td>18</td>
<td>11.1538</td>
<td>1.4684</td>
<td>0.0</td>
</tr>
<tr>
<td>19</td>
<td>9.9144</td>
<td>1.3053</td>
<td>0.0</td>
</tr>
<tr>
<td>20</td>
<td>10.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>21</td>
<td>12.5</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>22</td>
<td>15.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>23</td>
<td>17.5</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>101</td>
<td>9.9144</td>
<td>-1.3053</td>
<td>1.0</td>
</tr>
<tr>
<td>102</td>
<td>11.1538</td>
<td>-1.4684</td>
<td>1.0</td>
</tr>
<tr>
<td>103</td>
<td>12.3931</td>
<td>-1.6316</td>
<td>1.0</td>
</tr>
<tr>
<td>104</td>
<td>13.6324</td>
<td>-1.7947</td>
<td>1.0</td>
</tr>
<tr>
<td>105</td>
<td>14.8717</td>
<td>-1.9579</td>
<td>1.0</td>
</tr>
<tr>
<td>106</td>
<td>16.1110</td>
<td>-2.1211</td>
<td>1.0</td>
</tr>
<tr>
<td>107</td>
<td>17.3503</td>
<td>-2.2842</td>
<td>1.0</td>
</tr>
</tbody>
</table>
PLASTICITY EXAMPLE PROBLEM PLAS602 / Input Data

**BCSET
   ID DISP13
   VALUE
   GMR GMR1
   SURFACE SURF1
   ELEMENTS 1 2 3 4 5 6 7 8
     DISP 3
     SPLIST ALL
     T 1 0.0

**BCSET
   ID DISP73
   VALUE
   LOCAL
   GMR GMR1
   SURFACE SURF1
   ELEMENTS 9 10 11 12 13 14 15 16
     DISP 1
     SPLIST ALL
     T 1 0.0

**BCSET
   ID TRAC12
   VALUE
   LOCAL
   GMR GMR1
   SURFACE SURF1
   ELEMENTS 17
     TIMES 1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12.
   TRAC 1
   SPLIST ALL
     T 1 -250.0
     T 2 -275.0
     T 3 -300.0
     T 4 -325.0
     T 5 -350.0
     T 6 -375.0
     T 7 -400.0
     T 8 -410.0
     T 9 -420.0
     T 10 -430.0
     T 11 -440.0
     T 12 -450.0
     T 13 -455.0
     T 14 -460.0
     T 15 -465.0
     T 16 -470.0
     T 17 -475.0
     T 18 -480.0
     T 19 -485.0
     T 20 -495.0
$
$ END OF DATA
### PLASTICITY EXAMPLE PROBLEM PLAS602 / Selected Output

#### JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE

**INTERIOR STRESS AT TIME = 10.0000 FOR REGION = GM1**

<table>
<thead>
<tr>
<th>NODE</th>
<th>SIGMA-XX</th>
<th>SIGMA-YY</th>
<th>SIGMA-ZZ</th>
<th>TAU-XY</th>
<th>TAU-XZ</th>
<th>TAU-YZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>-0.397758E+03</td>
<td>0.270349E+03</td>
<td>-0.437602E+02</td>
<td>0.909878E+02</td>
<td>-0.231789E+01</td>
<td>0.306473E+00</td>
</tr>
<tr>
<td>201</td>
<td>-0.384075E+03</td>
<td>0.285010E+03</td>
<td>-0.330210E+02</td>
<td>0.894281E+02</td>
<td>0.106619E+01</td>
<td>-0.191894E+00</td>
</tr>
<tr>
<td>1</td>
<td>-0.384248E+03</td>
<td>0.278167E+03</td>
<td>-0.363330E+02</td>
<td>0.918806E+02</td>
<td>0.412484E+01</td>
<td>0.543625E+00</td>
</tr>
<tr>
<td>20</td>
<td>-0.407711E+03</td>
<td>0.284736E+03</td>
<td>-0.409720E+02</td>
<td>0.414813E+00</td>
<td>0.000000E+00</td>
<td>-0.127245E+01</td>
</tr>
<tr>
<td>19</td>
<td>-0.395655E+03</td>
<td>0.272460E+03</td>
<td>-0.406581E+02</td>
<td>-0.910414E+02</td>
<td>0.243674E+01</td>
<td>0.321178E+00</td>
</tr>
<tr>
<td>219</td>
<td>-0.365773E+03</td>
<td>0.283308E+03</td>
<td>-0.344644E+02</td>
<td>-0.894288E+02</td>
<td>0.159988E+01</td>
<td>-0.314662E+00</td>
</tr>
<tr>
<td>119</td>
<td>-0.393616E+03</td>
<td>0.274866E+03</td>
<td>-0.387867E+02</td>
<td>-0.908113E+02</td>
<td>-0.217849E+01</td>
<td>-0.287120E+00</td>
</tr>
<tr>
<td>120</td>
<td>-0.411829E+03</td>
<td>0.280681E+03</td>
<td>-0.441993E+02</td>
<td>0.121263E+00</td>
<td>0.000000E+00</td>
<td>0.133678E+00</td>
</tr>
</tbody>
</table>

#### JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE

**INTERIOR STRESS AT TIME = 10.0000 FOR REGION = GM1**

<table>
<thead>
<tr>
<th>NODE</th>
<th>SIGMA-XX</th>
<th>SIGMA-YY</th>
<th>SIGMA-ZZ</th>
<th>TAU-XY</th>
<th>TAU-XZ</th>
<th>TAU-YZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>109</td>
<td>0.114488E+02</td>
<td>0.367736E+03</td>
<td>0.122995E+03</td>
<td>0.604143E+02</td>
<td>0.406951E+00</td>
<td>-0.534817E+01</td>
</tr>
<tr>
<td>209</td>
<td>0.610522E+01</td>
<td>0.377527E+03</td>
<td>0.114790E+03</td>
<td>0.497064E+02</td>
<td>0.102944E+01</td>
<td>-0.131649E+01</td>
</tr>
<tr>
<td>9</td>
<td>0.187399E+02</td>
<td>0.409735E+03</td>
<td>0.123835E+03</td>
<td>0.523250E+02</td>
<td>0.264109E+01</td>
<td>-0.341142E+02</td>
</tr>
<tr>
<td>10</td>
<td>0.291634E+02</td>
<td>0.388943E+03</td>
<td>0.115917E+03</td>
<td>0.586843E+02</td>
<td>0.000000E+00</td>
<td>-0.674978E+02</td>
</tr>
<tr>
<td>11</td>
<td>0.205498E+02</td>
<td>0.411270E+03</td>
<td>0.124728E+03</td>
<td>0.524186E+02</td>
<td>-0.231390E+01</td>
<td>-0.299907E+02</td>
</tr>
<tr>
<td>211</td>
<td>0.527742E+01</td>
<td>0.378152E+03</td>
<td>0.115029E+03</td>
<td>0.498270E+02</td>
<td>0.988978E+01</td>
<td>0.987533E+02</td>
</tr>
<tr>
<td>111</td>
<td>0.115394E+02</td>
<td>0.388292E+03</td>
<td>0.128647E+03</td>
<td>0.504860E+02</td>
<td>0.423199E+00</td>
<td>0.658628E+00</td>
</tr>
<tr>
<td>110</td>
<td>0.192720E+01</td>
<td>0.391864E+03</td>
<td>0.128319E+03</td>
<td>-0.110063E+01</td>
<td>0.000000E+00</td>
<td>-0.852908E+03</td>
</tr>
</tbody>
</table>

#### JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE

**INELASTIC SOLUTION AT TIME = 10.0000**

**NODE TYPE: E = ELASTIC NODE, P = PLASTIC NODE**
## PLASTICITY EXAMPLE PROBLEM PLAS602 / Selected Output

<table>
<thead>
<tr>
<th>NODE</th>
<th>TYPE</th>
<th>EQUIVALENT PLASTIC STRAIN</th>
<th>EQUIVALENT STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>P</td>
<td>3.38406E-01</td>
<td>6.00002E+02</td>
</tr>
<tr>
<td>201</td>
<td>P</td>
<td>3.41856E-01</td>
<td>6.00001E+02</td>
</tr>
<tr>
<td>1</td>
<td>P</td>
<td>3.33060E-01</td>
<td>6.00002E+02</td>
</tr>
<tr>
<td>20</td>
<td>P</td>
<td>3.41273E-01</td>
<td>6.00002E+02</td>
</tr>
<tr>
<td>19</td>
<td>P</td>
<td>3.39623E-01</td>
<td>6.00002E+02</td>
</tr>
<tr>
<td>219</td>
<td>P</td>
<td>3.46894E-01</td>
<td>6.00001E+02</td>
</tr>
<tr>
<td>119</td>
<td>P</td>
<td>3.41131E-01</td>
<td>6.00002E+02</td>
</tr>
<tr>
<td>120</td>
<td>P</td>
<td>3.42517E-01</td>
<td>6.00002E+02</td>
</tr>
</tbody>
</table>

### JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE

**INELASTIC SOLUTION AT TIME = 10.0000**

**NODE TYPE:** E = ELASTIC Node, P = PLASTIC NODE

<table>
<thead>
<tr>
<th>NODE</th>
<th>TYPE</th>
<th>EQUIVALENT PLASTIC STRAIN</th>
<th>EQUIVALENT STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.99694E+02</td>
</tr>
<tr>
<td>12</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.98760E+02</td>
</tr>
<tr>
<td>112</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.94051E+02</td>
</tr>
<tr>
<td>109</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.44585E+02</td>
</tr>
<tr>
<td>209</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.42506E+02</td>
</tr>
<tr>
<td>9</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.81315E+02</td>
</tr>
<tr>
<td>10</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.43756E+02</td>
</tr>
<tr>
<td>11</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.42019E+02</td>
</tr>
<tr>
<td>211</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.42937E+02</td>
</tr>
<tr>
<td>111</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.45060E+02</td>
</tr>
<tr>
<td>110</td>
<td>E</td>
<td>0.00000E+00</td>
<td>3.44982E+02</td>
</tr>
</tbody>
</table>
EXAMPLE PROBLEM: PLAS603

ANALYSIS TYPE: STRESS ANALYSIS
3-D, ELASTOPLASTIC ANALYSIS, TWO-SURFACE CYCLIC
PLASTICITY MATERIAL MODEL, ITERATIVE ALGORITHM.

PROBLEM DESCRIPTION:
UNIT CUBE SUBJECTED TO CYCLIC LOADING, SIMPLY
SUPPORTED BOUNDARY CONDITIONS, AXIAL TENSION UP TO
1.75 TIMES YIELD STRESS AND AXIAL COMPRESSION UP TO 1.25
TIMES YIELD STRESS APPLIED FOR ONE COMPLETE CYCLE.

BOUNDARY ELEMENT MODEL:
6 (8-NODED) ELEMENTS AROUND BOUNDARY, 1 ON EACH SIDE OF
THE UNIT CUBE, 1 (20-NODED) VOLUME CELL FILLING UP THE
COMPLETE VOLUME OF THE BODY, GLOBAL BOUNDARY CONDITIONS
SPECIFIED EVERYWHERE.

REFERENCE FOR EXPERIMENTAL SOLUTION:
PRESENT EXPERIMENTAL RESULTS FOR TESTS CONDUCTED PRIMARILY
ON A BENCHMARK NOTCH SPECIMEN BUT ALSO PRESENT RESULTS
COMPARED HERE FOR A SPECIMEN WITHOUT A NOTCH.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>TIME</th>
<th>NODE</th>
<th>STRAIN (EPS-ZZ)</th>
<th>BEST SOLN.</th>
<th>EXPERIMENTAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.0</td>
<td>1</td>
<td>0.100268E-01</td>
<td>0.11200E-01</td>
<td></td>
</tr>
<tr>
<td>30.0</td>
<td>1</td>
<td>-0.588888E-02</td>
<td>-0.62200E-02</td>
<td></td>
</tr>
</tbody>
</table>

RUN TIME:
6 X BASE PROBLEM

MISCELLANEOUS:
TWO-SURFACE PLASTICITY MODEL DICTATES THAT THE MATERIAL
BECOMES PLASTIC AT LOWER STRESS LEVEL ALLOWING FOR A SMOOTH
TRANSITION FROM AN ELASTIC TO PLASTIC STATE, IT ALSO HELPS
IN DESCRIBING CYCLIC BEHAVIOR BY TAKING INTO ACCOUNT THE
REDUCED YIELD STRESS DURING UNLOADING AS SHOWN BY THE
BAUSCHINGER EFFECT, THE HYSTERESIS LOOP OF STRESS VS. STRAIN
IS OF MAIN INTEREST IN THE RESULTS.
PLASTICITY EXAMPLE PROBLEM PLAS03 / Geometry

NODES AND ELEMENTS
**CASE**
TITLE CYCLIC LOADING OF UNIT CUBE
PLASTICITY ITERATIVE $ \text{ITERATIVE PLASTICITY ALGORITHM SELECTED}$
TIMES 4.0 6.0 7.0 8.0 9.0 10.0 12.0 14.0 16.0 18.0 20.0 22.0
TIMES 24.0 26.0 28.0 30.0 33.0 36.0 38.0 40.0

**MATE**
ID MAT1
$ (\text{ALL UNITS ARE IN KSI})$
TEMP 70.0
EMOD 23.8E+03
POIS 0.3
INELASTIC
TWO SURFACE
YIELD 100.0 400.0 $ \text{SPECIFIES INNER AND OUTER YIELD}$
$ \text{STRESS VALUES, RESPECTIVELY}$
HARD 35.7E+03 11.9E+03 $ \text{SPECIFIES INNER AND OUTER}$
$ \text{HARDENING VALUES, RESPECTIVELY}$
TIME 7 17 $ \text{SPECIFIES THE TIME STEP NUMBERS AT}$
$ \text{WHICH A REVERSAL OF LOADING TAKES PLACE,}$
$ \text{THIS TURNS OFF THE ACCELERATED ALGORITHM}$

**GMR**
ID GMR1
MAT MAT1
TREF 70.0
POINTS
1 0.0 0.0 0.0
2 0.0 0.5 0.0
3 0.0 1.0 0.0
4 0.5 1.0 0.0
5 1.0 1.0 0.0
6 1.0 0.5 0.0
7 1.0 0.0 0.0
8 0.5 0.0 0.0
101 0.0 0.0 1.0
102 0.0 0.5 1.0
103 0.0 1.0 1.0
104 0.5 1.0 1.0
105 1.0 1.0 1.0
106 1.0 0.5 1.0
107 1.0 0.0 1.0
108 0.5 0.0 1.0
201 0.0 0.0 0.5
203 0.0 1.0 0.5
205 1.0 1.0 0.5
207 1.0 0.0 0.5
SURFACE SURF1
TYPE QUAD
ELEMENTS
1 1 2 3 4 5 6 7 8
2 101 108 107 106 105 104 103 102
PLASTICITY EXAMPLE PROBLEM PLAS603 / Input Data

3 1 201 101 102 103 203 3 2
4 7 6 5 205 105 106 107 207
5 1 8 7 207 107 108 101 201
6 3 203 103 104 105 205 5 4
NORMAL 1 +
VOLUME
CELL
51 1 2 3 4 5 6 7 8 201 203 205 207 101 102 103 104 105 106 107 108
FULL
$(FULL$ DEFINES$ THAT$ THE$ VOLUME$ IS$ COMPLETELY$ FILLED$ WITH$ CELLS)$
$(AND$ INITIAL$ STRESS$ EXPANSION$ IS$ USED$ TO$ COMPUTE$ JUMP$ TERMS.)$

**BCSET
ID DISP13
VALUE
GMR GMRI
SURFACE SURF1
ELEMENTS 1
DISP 3
SPLIST ALL
T 1 0.0

**BCSET
ID DISP31
VALUE
GMR GMRI
SURFACE SURF1
ELEMENTS 3
DISP 1
SPLIST ALL
T 1 0.0

**BCSET
ID DISP52
VALUE
GMR GMRI
SURFACE SURF1
ELEMENTS 5
DISP 2
SPLIST ALL
T 1 0.0

**BCSET
ID TRAC12
VALUE
GMR GMRI
SURFACE SURF1
ELEMENTS 2
TIMES 0.0 10.0 30.0 40.0
TRAC 3
SPLIST ALL
T 1 0.0
T 2 175.0
T 3 -125.0
T 4 0.0
$(TRACTION$ BOUNDARY$ COND.$ SPECIFYING$ INITIAL$ LOADING$ UPTO)$
$(TIME$ 10.0$ TO$ STRESS$ OF$ 175.0$ KSI,$ FOLLOWED$ BY$ UNLOADING)$
PLASTICITY EXAMPLE PROBLEM PLAS603 / Input Data

$ (TO -125.0 KSI AT TIME 30.0 AND FINALLY RELOADED TO 0.0)
$ (AT TIME 40.0.)
$ END OF DATA
PLASTICITY EXAMPLE PROBLEM PLAS603 / Selected Output

**JOB TITLE:** CYCLIC LOADING OF UNIT CUBE

**NUMBER OF SUB-INCREMENT = 7 FOR TIME = 30.0000**

<table>
<thead>
<tr>
<th>NODE</th>
<th>EPS-XX</th>
<th>EPS-YY</th>
<th>EPS-EZ</th>
<th>EPS-XY</th>
<th>EPS-EZ</th>
<th>EPS-YZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.189353E-02</td>
<td>0.189436E-02</td>
<td>-0.588856E-02</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.189409E-02</td>
<td>0.189337E-02</td>
<td>-0.588818E-02</td>
<td>-0.201641E-07</td>
<td>0.000000E+00</td>
<td>0.352786E-06</td>
</tr>
<tr>
<td>3</td>
<td>0.189377E-02</td>
<td>0.189264E-02</td>
<td>-0.588792E-02</td>
<td>-0.144105E-07</td>
<td>0.000000E+00</td>
<td>0.548864E-06</td>
</tr>
<tr>
<td>4</td>
<td>0.189320E-02</td>
<td>0.189202E-02</td>
<td>-0.588814E-02</td>
<td>0.337308E-07</td>
<td>0.348338E-06</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.189312E-02</td>
<td>0.189295E-02</td>
<td>-0.588803E-02</td>
<td>0.294308E-07</td>
<td>0.460641E-06</td>
<td>0.380160E-06</td>
</tr>
<tr>
<td>6</td>
<td>0.189342E-02</td>
<td>0.189333E-02</td>
<td>-0.588843E-02</td>
<td>0.290599E-07</td>
<td>0.000000E+00</td>
<td>0.268764E-06</td>
</tr>
<tr>
<td>7</td>
<td>0.189295E-02</td>
<td>0.189407E-02</td>
<td>-0.588852E-02</td>
<td>0.312888E-07</td>
<td>0.555922E-06</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>8</td>
<td>0.189312E-02</td>
<td>0.189465E-02</td>
<td>-0.588844E-02</td>
<td>0.461918E-07</td>
<td>0.480918E-06</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>201</td>
<td>0.189331E-02</td>
<td>0.189353E-02</td>
<td>-0.588765E-02</td>
<td>0.000000E+00</td>
<td>-0.409453E-06</td>
<td>-0.327060E-06</td>
</tr>
<tr>
<td>203</td>
<td>0.189369E-02</td>
<td>0.189291E-02</td>
<td>-0.588905E-02</td>
<td>0.000000E+00</td>
<td>-0.431805E-06</td>
<td>-0.494418E-06</td>
</tr>
<tr>
<td>205</td>
<td>0.189294E-02</td>
<td>0.189295E-02</td>
<td>-0.588820E-02</td>
<td>0.000000E+00</td>
<td>0.479064E-06</td>
<td>0.482393E-06</td>
</tr>
<tr>
<td>207</td>
<td>0.189273E-02</td>
<td>0.189377E-02</td>
<td>-0.588878E-02</td>
<td>0.000000E+00</td>
<td>0.394132E-06</td>
<td>0.389418E-06</td>
</tr>
<tr>
<td>101</td>
<td>0.189415E-02</td>
<td>0.189377E-02</td>
<td>-0.588728E-02</td>
<td>0.000000E+00</td>
<td>-0.432277E-06</td>
<td>-0.475820E-06</td>
</tr>
<tr>
<td>102</td>
<td>0.189427E-02</td>
<td>0.189291E-02</td>
<td>-0.588764E-02</td>
<td>0.126702E-06</td>
<td>0.000000E+00</td>
<td>-0.307666E-06</td>
</tr>
<tr>
<td>103</td>
<td>0.189395E-02</td>
<td>0.189238E-02</td>
<td>-0.588668E-02</td>
<td>0.144393E-06</td>
<td>0.469926E-06</td>
<td>0.999745E-07</td>
</tr>
<tr>
<td>104</td>
<td>0.189293E-02</td>
<td>0.189281E-02</td>
<td>-0.588790E-02</td>
<td>-0.165102E-06</td>
<td>-0.684112E-07</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>105</td>
<td>0.189225E-02</td>
<td>0.189242E-02</td>
<td>-0.588644E-02</td>
<td>-0.362705E-06</td>
<td>0.389001E-06</td>
<td>0.206490E-06</td>
</tr>
<tr>
<td>106</td>
<td>0.189240E-02</td>
<td>0.189288E-02</td>
<td>-0.588633E-02</td>
<td>-0.801115E-07</td>
<td>0.000000E+00</td>
<td>-0.262035E-06</td>
</tr>
<tr>
<td>107</td>
<td>0.189171E-02</td>
<td>0.189372E-02</td>
<td>-0.588578E-02</td>
<td>0.207518E-06</td>
<td>0.149027E-06</td>
<td>-0.637781E-06</td>
</tr>
<tr>
<td>108</td>
<td>0.189269E-02</td>
<td>0.189404E-02</td>
<td>-0.588722E-02</td>
<td>0.123943E-06</td>
<td>-0.122994E-06</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>
### PLASTICITY EXAMPLE PROBLEM PLAS03 / Selected Output

#### JOB TITLE:  CYCLIC LOADING OF UNIT CUBE

**INELASTIC SOLUTION AT TIME = 30.0000**

**NODE TYPE:**  
- E = ELASTIC NODE
- P = PLASTIC NODE
- NE = METAKLASTIC NODE

<table>
<thead>
<tr>
<th>NODE</th>
<th>TYPE</th>
<th>EQUIVALENT PLASTIC STRAIN</th>
<th>EQUIVALENT STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>NE</td>
<td>6.47921E-04</td>
<td>1.24692E+02</td>
</tr>
<tr>
<td>2</td>
<td>NE</td>
<td>6.47766E-04</td>
<td>1.24680E+02</td>
</tr>
<tr>
<td>3</td>
<td>NE</td>
<td>6.47537E-04</td>
<td>1.24669E+02</td>
</tr>
<tr>
<td>4</td>
<td>NE</td>
<td>6.48655E-04</td>
<td>1.24640E+02</td>
</tr>
<tr>
<td>5</td>
<td>NE</td>
<td>6.48929E-04</td>
<td>1.24633E+02</td>
</tr>
<tr>
<td>6</td>
<td>NE</td>
<td>6.49105E-04</td>
<td>1.24641E+02</td>
</tr>
<tr>
<td>7</td>
<td>NE</td>
<td>6.48107E-04</td>
<td>1.24673E+02</td>
</tr>
<tr>
<td>8</td>
<td>NE</td>
<td>6.47962E-04</td>
<td>1.24682E+02</td>
</tr>
<tr>
<td>201</td>
<td>NE</td>
<td>6.49083E-04</td>
<td>1.24629E+02</td>
</tr>
<tr>
<td>203</td>
<td>NE</td>
<td>6.48976E-04</td>
<td>1.24616E+02</td>
</tr>
<tr>
<td>205</td>
<td>NE</td>
<td>6.48733E-04</td>
<td>1.24603E+02</td>
</tr>
<tr>
<td>207</td>
<td>NE</td>
<td>6.48906E-04</td>
<td>1.24615E+02</td>
</tr>
<tr>
<td>101</td>
<td>NE</td>
<td>6.49319E-04</td>
<td>1.24625E+02</td>
</tr>
<tr>
<td>102</td>
<td>NE</td>
<td>6.48306E-04</td>
<td>1.24623E+02</td>
</tr>
<tr>
<td>103</td>
<td>NE</td>
<td>6.48369E-04</td>
<td>1.24609E+02</td>
</tr>
<tr>
<td>104</td>
<td>NE</td>
<td>6.48128E-04</td>
<td>1.24602E+02</td>
</tr>
<tr>
<td>105</td>
<td>NE</td>
<td>6.48701E-04</td>
<td>1.24679E+02</td>
</tr>
<tr>
<td>106</td>
<td>NE</td>
<td>6.48796E-04</td>
<td>1.24699E+02</td>
</tr>
<tr>
<td>107</td>
<td>NE</td>
<td>6.48706E-04</td>
<td>1.24692E+02</td>
</tr>
<tr>
<td>108</td>
<td>NE</td>
<td>6.49066E-04</td>
<td>1.24620E+02</td>
</tr>
</tbody>
</table>
EXAMPLE PROBLEM: TRAN601

ANALYSIS TYPE: TRANSIENT ELASTODYNAMICS
3-D, TRANSIENT ELASTODYNAMICS BY CONVOLUTION INTEGRALS AND LINEAR VARIATION.

PROBLEM DESCRIPTION:
DETERMINE THE TIME-DEPENDENT RADIAL DISPLACEMENT AT THE SURFACE OF A SPHERICAL CAVITY (RADIUS=212) IN AN INFINITE MEDIUM (POISSON'S RATIO=0.25, YOUNG'S MODULUS=8993000, DENSITY=0.00025) UNDER A HEAVISIDE INTERNAL PRESSURE 1000\(H(t)\). INITIAL DISPLACEMENT CONDITIONS ARE ZERO.

BOUNDARY ELEMENT MODEL:
AN OCTANT MODEL OF A SPHERICAL SURFACE USING THREE 8-NODED QUADRATIC SURFACE ELEMENTS. GLOBAL BOUNDARY CONDITIONS SPECIFIED EVERYWHERE.

REFERENCE FOR ANALYTICAL SOLUTION:
TIMOSHENKO, THEORY OF ELASTICITY, PG. 509-513.

SOLUTION POINTS TO VERIFY:
(DISPLACEMENT AT POINT 5(212,0,0) IN X-DIRECTION)

<table>
<thead>
<tr>
<th>TIME STEP</th>
<th>TIME STEP</th>
<th>TIME POINT</th>
<th>BEST SOL</th>
<th>ANALYTICAL SOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.5000E-04</td>
<td>5</td>
<td>.63449E-02</td>
<td>6.2534E-03</td>
</tr>
<tr>
<td>2</td>
<td>7.0000E-04</td>
<td>5</td>
<td>.11018E-01</td>
<td>1.1265E-02</td>
</tr>
<tr>
<td>10</td>
<td>3.5000E-03</td>
<td>5</td>
<td>.15500E-01</td>
<td>1.6127E-02</td>
</tr>
</tbody>
</table>

RUN TIME:

MISCELLANEOUS:
THIS PROBLEM IS AN EXTERIOR PROBLEM, SO AN EXTERIOR CARD IS REQUIRED. BY UTILIZING SYMMETRY, ONLY ONE OCTANT GEOMETRY AND BOUNDARY CONDITIONS ARE REQUIRED.
NODES AND ELEMENTS
**CASE
TITL A SPHERICAL CAVITY IN INFINITE SPACE UNDER INTERNAL PRESSURE
TRANSIENT 10 0.00035
TIME VARIATION LINEAR
SYMMETRY OCTAN
PRINT BOUN

**MAT
ID MAT1
EMOD 8993000.0
POISS 0.25
DENS 0.00025

**GMR
ID GMR1
MAT MAT1
EXTERIOR POINTS
1 0.000 0.000 212.000
2 81.129 0.000 195.863
3 149.907 0.000 149.907
4 195.863 0.000 81.129
5 212.000 0.000 0.000
6 195.863 81.129 0.000
7 149.907 149.907 0.000
8 81.129 195.863 0.000
9 0.000 212.000 0.000
10 0.000 195.863 81.129
11 0.000 149.907 149.907
12 0.000 81.129 195.863
13 142.864 64.216 142.864
14 142.864 142.864 64.216
15 64.216 142.864 142.864
16 122.398 122.398 122.398
SURFACE SURF1
TYPE QUAD ELEMENTS
1 1 2 3 13 16 15 11 12
2 3 4 5 6 7 14 16 13
3 16 14 7 8 9 10 11 15
NORMAL 1 -

**BCSET
ID BCS1
VALUE
GMR GMR1
SURFACE SURF1
ELEMENTS 1 2 3
TRAC 1 SPLIST 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16
T 1 0. 382.684 707.109 923.882 1000. 923.882 707.109 382.684 0. 0.

$ NOTE: EXTERIOR PROBLEM
TRANSIENT DYNAMICS EXAMPLE PROBLEM TRAN601 / Input Data

T 1  0.  0.  673.887  673.887  302.906  577.349
TRAC 2
SPLIST  1  2  3  4  5  6  7  8  9  10  11  12  13  14  15  16
T 1  0.  0.  0.  0.  0.  382.684  707.109  923.882  1000.  923.882  707.109
T 1  382.684  302.906  673.887  673.887  577.349
TRAC 3
SPLIST  1  2  3  4  5  6  7  8  9  10  11  12  13  14  15  16
T 1  1000.  923.882  707.109  382.684  0.  0.  0.  0.  0.  382.684  707.109
T 1  923.882  673.887  302.906  673.887  577.349
$
\$
END OF DATA
## Job Title: A Spherical Cavity in Infinite Space Under Internal Pressure

**Boundary Solution at Time = 0.0003S**

For Region = One

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-0.62769E-19</td>
<td>-0.12662E-18</td>
<td>0.63449E-02</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.10000E+04</td>
</tr>
<tr>
<td>1</td>
<td>12</td>
<td>-0.42061E-18</td>
<td>0.23266E-02</td>
<td>0.66283E-02</td>
<td>0.00000E+00</td>
<td>0.38268E+03</td>
<td>0.92388E+03</td>
</tr>
<tr>
<td>1</td>
<td>11</td>
<td>0.17674E-18</td>
<td>0.43689E-02</td>
<td>0.43689E-02</td>
<td>0.00000E+00</td>
<td>0.70711E+03</td>
<td>0.70711E+03</td>
</tr>
<tr>
<td>1</td>
<td>15</td>
<td>0.18508E-02</td>
<td>0.41402E-02</td>
<td>0.41402E-02</td>
<td>0.30291E+03</td>
<td>0.67389E+03</td>
<td>0.67389E+03</td>
</tr>
<tr>
<td>1</td>
<td>16</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
</tr>
<tr>
<td>1</td>
<td>13</td>
<td>0.41402E-02</td>
<td>0.13508E-02</td>
<td>0.41402E-02</td>
<td>0.67389E+03</td>
<td>0.30291E+03</td>
<td>0.67389E+03</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>0.43596E-02</td>
<td>-0.37000E-18</td>
<td>0.43596E-02</td>
<td>0.70711E+03</td>
<td>0.00000E+00</td>
<td>0.70711E+03</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>0.23266E-02</td>
<td>0.18577E-18</td>
<td>0.66283E-02</td>
<td>0.38268E+03</td>
<td>0.92388E+03</td>
<td>0.92388E+03</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.43596E-02</td>
<td>-0.37000E-18</td>
<td>0.43596E-02</td>
<td>0.70711E+03</td>
<td>0.00000E+00</td>
<td>0.70711E+03</td>
</tr>
<tr>
<td>2</td>
<td>13</td>
<td>0.41402E-02</td>
<td>0.13508E-02</td>
<td>0.41402E-02</td>
<td>0.67389E+03</td>
<td>0.30291E+03</td>
<td>0.67389E+03</td>
</tr>
<tr>
<td>2</td>
<td>16</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
</tr>
<tr>
<td>2</td>
<td>14</td>
<td>0.41402E-02</td>
<td>0.13508E-02</td>
<td>0.41402E-02</td>
<td>0.67389E+03</td>
<td>0.30291E+03</td>
<td>0.67389E+03</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>0.43596E-02</td>
<td>0.43596E-02</td>
<td>-0.13614E-18</td>
<td>0.70711E+03</td>
<td>0.70711E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0.56283E-02</td>
<td>0.23266E-02</td>
<td>-0.60646E-18</td>
<td>0.92388E+03</td>
<td>0.38268E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>0.64449E-02</td>
<td>0.19266E-18</td>
<td>0.68961E-19</td>
<td>0.10000E+04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>0.56283E-02</td>
<td>-0.29863E-18</td>
<td>0.23266E-02</td>
<td>0.92388E+03</td>
<td>0.38268E+03</td>
<td>0.38268E+03</td>
</tr>
<tr>
<td>3</td>
<td>16</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.35235E-02</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
<td>0.57736E+03</td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>0.18508E-02</td>
<td>0.41402E-02</td>
<td>0.41402E-02</td>
<td>0.30291E+03</td>
<td>0.67389E+03</td>
<td>0.67389E+03</td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>0.17674E-18</td>
<td>0.43689E-02</td>
<td>0.43689E-02</td>
<td>0.00000E+00</td>
<td>0.70711E+03</td>
<td>0.70711E+03</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-0.53732E-20</td>
<td>0.56283E-02</td>
<td>0.23266E-02</td>
<td>0.00000E+00</td>
<td>0.92388E+03</td>
<td>0.38268E+03</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>0.90690E-19</td>
<td>0.64449E-02</td>
<td>-0.56866E-18</td>
<td>0.00000E+00</td>
<td>0.10000E+04</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>0.23266E-02</td>
<td>0.56283E-02</td>
<td>0.23266E-02</td>
<td>0.38268E+03</td>
<td>0.92388E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>7</td>
<td>0.43596E-02</td>
<td>0.43596E-02</td>
<td>-0.15614E-18</td>
<td>0.70711E+03</td>
<td>0.70711E+03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>14</td>
<td>0.41402E-02</td>
<td>0.41402E-02</td>
<td>0.18508E-02</td>
<td>0.67389E+03</td>
<td>0.67389E+03</td>
<td>0.30291E+03</td>
</tr>
</tbody>
</table>
VERIFICATION PROBLEM: D.403

ANALYSIS TYPE: ELASTIC ANALYSIS OF A 3-D CUBE WITH HOLES (USING HOLE ELEMENTS)

PROBLEM DESCRIPTION:
A 3-D UNIT CUBE HAS A TUBULAR HOLE OF RADIUS .1 CENTERED IN THE MIDDLE RUNNING FROM ONE SIDE TO THE OTHER. THE HOLE IS SUBJECTED TO A UNIFORM PRESSURE APPLIED IN THE HOLE.

BOUNDARY ELEMENT MODEL:
3-D UNIT CUBE IS MODELED WITH SIX QUADRATIC ELEMENTS AND THE HOLE IS MODELED WITH A SINGLE QUADRATIC LINE ELEMENT.

REFERENCE FOR SOLUTION:
RESULTS FROM EXACT BEM MODELING OF THE HOLE USING SURFACE PATCHES ARE USED FOR COMPARISON.

SOLUTION POINTS TO VERIFY:

TIME 1:

<table>
<thead>
<tr>
<th>NODE</th>
<th>X-DISPLACEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1004</td>
<td>0.06607</td>
</tr>
</tbody>
</table>

RUN TIME:
0.73 X BASE PROBLEM

MISCELLANEOUS:
NONE
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM D.403 / Geometry
**CASE**
**TITLE** - D.403 - 3-D CUBE WITH HOLE SUBJECTED TO INTERNAL PRESSURE
**TIMES** 1.
**ECHO**

**MATE**
**ID** MAT1
**TEMP** 70.0
**EMOD** 100.
**POIS** 0.3
**DENSITY** 1.
**ALPHA** 1.

**GMR**
**ID** GMRI
**MAT** MAT1
**TREF** 70.0
**TINI** 0.0

**POINTS**

1  0.000  0.000  0.000
2  0.500  0.000  0.000
3  1.000  0.000  0.000
4  1.000  0.500  0.000
5  1.000  1.000  0.000
6  0.500  1.000  0.000
7  0.000  1.000  0.000
8  0.000  0.500  0.000
1001  0.000  0.000  0.500
1003  0.000  0.000  0.500
1004  0.000  0.500  0.500
1005  0.000  1.000  0.500
1007  1.000  0.000  1.000
2001  0.000  0.000  1.000
2002  0.500  0.000  1.000
2003  1.000  0.000  1.000
2004  0.000  0.500  1.000
2005  0.000  1.000  1.000
2006  0.500  1.000  1.000
2007  0.000  1.000  1.000
2008  0.000  0.500  1.000

**SURFACE** SURF11
**TYPE** QUAD
**ELEMENTS**

101  1  2  3  1003  2003  2002  2001  1001
102  3  4  5  1005  2005  2004  2003  1003  1004
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

**NORMAL** 201 +
**HOLE**
**POINTS**

3001  0.5  0.5  0.0
3005  0.5  0.5  0.5
3009  0.5  0.5  1.0
**TYPE** QUAD
**Input Data**

```
ELEMENTS
   301 .1 3001 3005 3009

**BCSET**
   ID BC1
   VALUE
   GMR GMR1
   SURFACE SURF11
   ELEMENTS 104
   DISP 1
   SPLIST ALL
       T 1 0.0

**BCSET**
   ID BC2
   VALUE
   GMR GMR1
   SURFACE SURF11
   ELEMENTS 104
   POINTS 1001 1007
   DISP 3
   SPLIST 1001 1007
       T 1 0.0 0.0

**BCSET**
   ID BC3
   VALUE
   GMR GMR1
   SURFACE SURF11
   ELEMENTS 104
   POINTS 8 2008
   DISP 2
   SPLIST 8 2008
       T 1 0.0 0.0

**BCSET**
   ID PRESSURE
   VALUE
   GMR GMR1
   HOLE
   ELEMENT 301
   PRESS
   SPLIST ALL
       T 1 100.

$ END OF DATA
```
VERIFICATION PROBLEM: D.404

ANALYSIS TYPE:
THERMOELASTIC ANALYSIS OF A 3-D CUBE WITH A HOLE

PROBLEM DISCRIPTION:
A 3-D UNIT CUBE HAS A TUBULAR HOLE OF RADIUS .1 CENTERED IN THE MIDDLE RUNNING FROM ONE SIDE TO THE OTHER. THE CUBE IS SUBJECTED TO A UNIFORM TRACTION APPLIED ON A FACE OF THE CUBE PERDICULAR TO THE HOLE.

BOUNDARY ELEMENT MODEL:
3-D UNIT CUBE IS MODELED WITH SIX QUADRATIC ELEMENTS AND THE HOLE IS MODELED WITH THREE QUADRATIC LINE ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM AND LINEAR TEMPERATURE DISTRIBUTIONS

SOLUTION POINTS TO VERIFY:

TIME 1:

<table>
<thead>
<tr>
<th>NODE</th>
<th>X-DISPLACEMENT</th>
<th></th>
<th>Y-DISPLACEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ANAL.</td>
<td>BEST</td>
<td>ANAL.</td>
</tr>
<tr>
<td>4</td>
<td>.999</td>
<td>.999</td>
<td>.000</td>
</tr>
<tr>
<td>5</td>
<td>0.996</td>
<td>.996</td>
<td>.499</td>
</tr>
<tr>
<td>5001</td>
<td>.497</td>
<td>.497</td>
<td>.199</td>
</tr>
</tbody>
</table>

RUN TIME:
1.54 X BASE PROBLEM

MISCELLANEOUS:
NONE
**CASE**

**TITLE** - D.404 - 3-D CUBE WITH HOLE SUBJECTED TO A THERMAL BODY FORCE TIMES 1.

**MATE**

ID MAT1
TEMP 70.0
EMOD 100.
POIS 0.
ALPHA 1.

**GMAT**

ID GMAT1
MAT MAT1
TREF 70.0
TINI 0.0

**NGMP**

ID NGMP1
MAT MAT1
TREF 70.0
TINI 0.0

**SURFACE SURF11**

TYPE QUAD

ELEMEANTS

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
11 1 2 3 4 5 6 7 8

NORMAL 201 +

HOLE POINTS

3001 .5 .5 0.
3002 .5 .5 0.25
3003 .5 .5 0.5
3004 .5 .5 1.0
3005 .5 .5 1.5
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM D.404

**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
POINTS 1001 1007
DISP 3
SPLIST 1001 1007
T 1 0.0.

**BCSET**
ID BC3
GMR GMR1
SURFACE SURF11
ELEMENTS 104
POINTS 8 2008
DISP 2
SPLIST 8 2008
T 1 0.0.

**BODY FORCE**
THERMAL
TIME 1.
GMR GMR1
TEMPS
5001 1.

$ END OF DATA
EXAMPLE PROBLEM: ELAS603

ANALYSIS TYPE:
ELASTICITY (USING HOLE ELEMENTS)
3-D, STATIC, ELASTIC ANALYSIS OF A BODY WITH HOLES

PROBLEM DESCRIPTION:
A 3-D, FIXED END THICK CYLINDER WITH HOLES 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 holes 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 hole element is used per hole. Note the hole elements are curvilinear in geometry.

REFERENCE FOR ANALYTICAL SOLUTION:
Results from a axisymmetric BEM analysis are used for comparison with the 3-D hole results from the present data.

SOLUTION POINTS TO VERIFY:

<table>
<thead>
<tr>
<th>NODE</th>
<th>RADIAL POSITION</th>
<th>RADIAL X-DISPLACEMENT</th>
<th>CIRCUMFERENTIAL YY-STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>10.0</td>
<td>.077708</td>
<td>223.5E+03</td>
</tr>
<tr>
<td>21</td>
<td>12.5</td>
<td>.064537</td>
<td>153.5E+03</td>
</tr>
<tr>
<td>22</td>
<td>15.0</td>
<td>.056174</td>
<td>116.6E+03</td>
</tr>
<tr>
<td>23</td>
<td>17.5</td>
<td>.050939</td>
<td>94.9E+03</td>
</tr>
<tr>
<td>10</td>
<td>20.0</td>
<td>.047734</td>
<td>81.0E+03</td>
</tr>
</tbody>
</table>

RUN TIME:
11 X BASE PROBLEM

MISCELLANEOUS:
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM

GEOMETRY

NODES AND ELEMENTS
**CASE CONTROL**

TITL THICK CYLINDER WITH HOLES UNDER INTERNAL PRESSURE (SINGLE REGION)

**MATERIAL INPUT**

| ID | MAT1 | TEMP 70.0 | EMOD 30.0E+06 | POIS 0.3 |

**GMR**

| ID | GMR1 | MAT MAT1 | TREF 70.0 |

<table>
<thead>
<tr>
<th>POINTS</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.9144</td>
<td>-1.3053</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>11.1538</td>
<td>-1.4684</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>12.3931</td>
<td>-1.6316</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>13.6324</td>
<td>-1.7947</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>14.8717</td>
<td>-1.9579</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>16.1110</td>
<td>-2.1211</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>17.3503</td>
<td>-2.2842</td>
<td>0.0</td>
</tr>
<tr>
<td>8</td>
<td>18.5896</td>
<td>-2.4474</td>
<td>0.0</td>
</tr>
<tr>
<td>9</td>
<td>19.8289</td>
<td>-2.6105</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>20.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>11</td>
<td>19.8289</td>
<td>2.6105</td>
<td>0.0</td>
</tr>
<tr>
<td>12</td>
<td>18.5896</td>
<td>2.4474</td>
<td>0.0</td>
</tr>
<tr>
<td>13</td>
<td>17.3503</td>
<td>2.2842</td>
<td>0.0</td>
</tr>
<tr>
<td>14</td>
<td>16.1110</td>
<td>2.1211</td>
<td>0.0</td>
</tr>
<tr>
<td>15</td>
<td>14.8717</td>
<td>1.9579</td>
<td>0.0</td>
</tr>
<tr>
<td>16</td>
<td>13.6324</td>
<td>1.7947</td>
<td>0.0</td>
</tr>
<tr>
<td>17</td>
<td>12.3931</td>
<td>1.6316</td>
<td>0.0</td>
</tr>
<tr>
<td>18</td>
<td>11.1538</td>
<td>1.4684</td>
<td>0.0</td>
</tr>
<tr>
<td>19</td>
<td>9.9144</td>
<td>1.3053</td>
<td>0.0</td>
</tr>
<tr>
<td>20</td>
<td>10.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>21</td>
<td>12.5</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>22</td>
<td>15.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>23</td>
<td>17.5</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>101</td>
<td>9.9144</td>
<td>-1.3053</td>
<td>2.0</td>
</tr>
<tr>
<td>102</td>
<td>11.1538</td>
<td>-1.4684</td>
<td>2.0</td>
</tr>
<tr>
<td>103</td>
<td>12.3931</td>
<td>-1.6316</td>
<td>2.0</td>
</tr>
<tr>
<td>104</td>
<td>13.6324</td>
<td>-1.7947</td>
<td>2.0</td>
</tr>
<tr>
<td>105</td>
<td>14.8717</td>
<td>-1.9579</td>
<td>2.0</td>
</tr>
<tr>
<td>106</td>
<td>16.1110</td>
<td>-2.1211</td>
<td>2.0</td>
</tr>
<tr>
<td>107</td>
<td>17.3503</td>
<td>-2.2842</td>
<td>2.0</td>
</tr>
<tr>
<td>108</td>
<td>18.5896</td>
<td>-2.4474</td>
<td>2.0</td>
</tr>
<tr>
<td>109</td>
<td>19.8289</td>
<td>-2.6105</td>
<td>2.0</td>
</tr>
<tr>
<td>110</td>
<td>20.0</td>
<td>0.0</td>
<td>2.0</td>
</tr>
<tr>
<td>111</td>
<td>19.8289</td>
<td>2.6105</td>
<td>2.0</td>
</tr>
<tr>
<td>112</td>
<td>18.5896</td>
<td>2.4474</td>
<td>2.0</td>
</tr>
<tr>
<td>113</td>
<td>17.3503</td>
<td>2.2842</td>
<td>2.0</td>
</tr>
<tr>
<td>114</td>
<td>16.1110</td>
<td>2.1211</td>
<td>2.0</td>
</tr>
<tr>
<td>115</td>
<td>14.8717</td>
<td>1.9579</td>
<td>2.0</td>
</tr>
</tbody>
</table>
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM ELAS603 / Input Data

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   10.00  0.0  2.0
121   12.50  0.0  2.0
122   15.00  0.0  2.0
123   17.50  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   20.00  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   10.00  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    1     18
 14    17  217  117  116  115  215    1     16
 15    15  215  115  114  113  213    1     14
 16    13  213  113  112  111  211    1     12
 17     1  201  101  120  119  219    1     19  20  220
 20  109  209     9    10     1  211  111  110  210

NORMAL 1 +

$\$ \$\$

DEFINE HOLE POINTS AND ELEMENTS

HOLES
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  0.0  1.
  502  13.75  0.0  1.
  503  16.25  0.0  1.
  504  18.75  0.0  1.

BEST3D User Manual
April, 1989
Page 6.86
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM ELAS603 / Input Data

.601 11.1538 -1.4684 1.
.602 13.6324 -1.7947 1.
.604 18.5896 -2.4474 1.

TYPE QUAD ELEMENTS
701 .5 401 501 601
702 .5 402 502 602
703 .5 403 503 603
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 -100000.

$ $ END OF DATA $

BEST3D User Manual  April, 1989  Page 6.87
### Holes, Inserts, and Body Forces Example Problem Elas603 / Selected Output

#### Job Title: Thick Cylinder with Holes Under Internal Pressure (Single Region)

**Hole Element Solution at Time = 0.0000**

**For Region = Gxg1**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>701</td>
<td>0.68387E-01</td>
<td>0.98921E-02</td>
<td>0.76308E-09</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.70631E-01</td>
<td>0.10212E-01</td>
<td>0.14288E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.70631E-01</td>
<td>0.10212E-01</td>
<td>-0.14288E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.69166E-01</td>
<td>0.15084E-06</td>
<td>-0.78924E-10</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.71396E-01</td>
<td>0.62787E-06</td>
<td>0.48832E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.71396E-01</td>
<td>0.63242E-06</td>
<td>-0.48832E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.68378E-01</td>
<td>-0.99840E-02</td>
<td>-0.14294E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.70631E-01</td>
<td>-0.10212E-01</td>
<td>0.14294E-03</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>701</td>
<td>0.70631E-01</td>
<td>-0.10212E-01</td>
<td>-0.14294E-03</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>

#### Job Title: Thick Cylinder with Holes Under Internal Pressure (Single Region)

**Modal Output at Time = 0.000000**

**For Region = Gxg1**

<table>
<thead>
<tr>
<th>Node</th>
<th>Displacement</th>
<th>Stress</th>
<th>Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X/Y/Z</td>
<td>II/YY/ZZ</td>
<td>IV/IZ/YZ</td>
</tr>
<tr>
<td>19</td>
<td>0.76942E-01</td>
<td>-0.87392E+05</td>
<td>-0.42497E+06</td>
</tr>
<tr>
<td>20</td>
<td>0.10122E-01</td>
<td>0.22517E+00</td>
<td>0.82712E+04</td>
</tr>
<tr>
<td>21</td>
<td>0.00000E+00</td>
<td>0.57699E+05</td>
<td>0.10913E+04</td>
</tr>
<tr>
<td>22</td>
<td>0.77708E-01</td>
<td>-0.93024E+05</td>
<td>-0.18940E+01</td>
</tr>
<tr>
<td>23</td>
<td>-0.93412E-07</td>
<td>0.22351E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td></td>
<td>0.00000E+00</td>
<td>0.62811E+05</td>
<td>-0.29442E-01</td>
</tr>
<tr>
<td>24</td>
<td>0.64577E-01</td>
<td>-0.63572E+05</td>
<td>-0.16888E+00</td>
</tr>
<tr>
<td>25</td>
<td>0.37504E-07</td>
<td>0.15349E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>26</td>
<td>0.00000E+00</td>
<td>0.58967E+05</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>27</td>
<td>0.61714E-01</td>
<td>-0.29188E+05</td>
<td>-0.16664E+00</td>
</tr>
<tr>
<td>28</td>
<td>-0.27236E-07</td>
<td>0.11683E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>29</td>
<td>0.00000E+00</td>
<td>0.43062E+05</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>30</td>
<td>0.50939E-01</td>
<td>-0.98318E+04</td>
<td>-0.78835E+00</td>
</tr>
<tr>
<td>31</td>
<td>-0.16330E-08</td>
<td>0.94951E+05</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>32</td>
<td>0.00000E+00</td>
<td>0.35170E+05</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>
VERIFICATION PROBLEM: D.406

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 INSERTS 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:

<table>
<thead>
<tr>
<th>NODE</th>
<th>RADIAL POSITION</th>
<th>RADIAL X-DISPLACEMENT AXIS.</th>
<th>CIRCUMFERENTIAL YY-STRESS AXIS.</th>
</tr>
</thead>
<tbody>
<tr>
<td>9020</td>
<td>10.0</td>
<td>.913</td>
<td>5.11</td>
</tr>
<tr>
<td>9021</td>
<td>12.5</td>
<td>.712</td>
<td>4.28</td>
</tr>
<tr>
<td>9022</td>
<td>15.0</td>
<td>.602</td>
<td>3.46</td>
</tr>
<tr>
<td>9023</td>
<td>17.5</td>
<td>.537</td>
<td>3.04</td>
</tr>
<tr>
<td>9024</td>
<td>20.0</td>
<td>.500</td>
<td>2.72</td>
</tr>
</tbody>
</table>

RUN TIME:
6.50 X BASE PROBLEM

MISCELLANEOUS:
NONE
**CASE CONTROL**

**CASE CONTROL**
TITLE D.406 - THICK CYLINDER W/ INSERTS UNDER INTERNAL PRESSURE
PRINT BOUND
PRINT LOAD
ECHO

**MATERIAL INPUT**

<table>
<thead>
<tr>
<th>ID</th>
<th>MATERIAL</th>
<th>TEMP</th>
<th>EMOD</th>
<th>POIS</th>
</tr>
</thead>
<tbody>
<tr>
<td>HAT1</td>
<td>70.0</td>
<td>100.0</td>
<td>0.3</td>
<td></td>
</tr>
</tbody>
</table>

**GMR**

<table>
<thead>
<tr>
<th>ID</th>
<th>MATERIAL</th>
<th>TREF</th>
</tr>
</thead>
<tbody>
<tr>
<td>GMR1</td>
<td>70.0</td>
<td></td>
</tr>
</tbody>
</table>

**GMR POINTS**

<table>
<thead>
<tr>
<th>POINT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.9144</td>
<td>-1.3053</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>11.1538</td>
<td>-1.4684</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>12.3931</td>
<td>-1.6316</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>13.6324</td>
<td>-1.7947</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>14.8717</td>
<td>-1.9579</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>16.1110</td>
<td>-2.1211</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>17.3503</td>
<td>-2.2842</td>
<td>0.0</td>
</tr>
<tr>
<td>8</td>
<td>18.5896</td>
<td>-2.4474</td>
<td>0.0</td>
</tr>
<tr>
<td>9</td>
<td>19.8289</td>
<td>-2.6105</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>20.0</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>19.8289</td>
<td>2.6105</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>18.5896</td>
<td>2.4474</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>17.3503</td>
<td>2.2842</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>16.1110</td>
<td>2.1211</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>14.8717</td>
<td>1.9579</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>13.6324</td>
<td>1.7947</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>12.3931</td>
<td>1.6316</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>11.1538</td>
<td>1.4684</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>9.9144</td>
<td>1.3053</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>10.0</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>12.5</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>15.0</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>17.5</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>101</td>
<td>9.9144</td>
<td>-1.3053</td>
<td></td>
</tr>
<tr>
<td>102</td>
<td>11.1538</td>
<td>-1.4684</td>
<td></td>
</tr>
<tr>
<td>103</td>
<td>12.3931</td>
<td>-1.6316</td>
<td></td>
</tr>
<tr>
<td>104</td>
<td>13.6324</td>
<td>-1.7947</td>
<td></td>
</tr>
<tr>
<td>105</td>
<td>14.8717</td>
<td>-1.9579</td>
<td></td>
</tr>
<tr>
<td>106</td>
<td>16.1110</td>
<td>-2.1211</td>
<td></td>
</tr>
<tr>
<td>107</td>
<td>17.3503</td>
<td>-2.2842</td>
<td></td>
</tr>
<tr>
<td>108</td>
<td>18.5896</td>
<td>-2.4474</td>
<td></td>
</tr>
<tr>
<td>109</td>
<td>19.8289</td>
<td>-2.6105</td>
<td></td>
</tr>
<tr>
<td>110</td>
<td>20.0</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>111</td>
<td>19.8289</td>
<td>2.6105</td>
<td></td>
</tr>
<tr>
<td>112</td>
<td>18.5896</td>
<td>2.4474</td>
<td></td>
</tr>
<tr>
<td>113</td>
<td>17.3503</td>
<td>2.2842</td>
<td></td>
</tr>
<tr>
<td>114</td>
<td>16.1110</td>
<td>2.1211</td>
<td></td>
</tr>
<tr>
<td>115</td>
<td>14.8717</td>
<td>1.9579</td>
<td></td>
</tr>
</tbody>
</table>
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM D.408

$\begin{align*}
116 & \quad 13.6324 & \quad 1.7947 & \quad 2.0 \\
117 & \quad 12.3931 & \quad 1.6316 & \quad 2.0 \\
118 & \quad 11.1538 & \quad 1.4684 & \quad 2.0 \\
119 & \quad 9.9144 & \quad 1.3053 & \quad 2.0 \\
120 & \quad 10.0 & \quad 0.0 & \quad 2.0 \\
121 & \quad 12.5 & \quad 0.0 & \quad 2.0 \\
122 & \quad 15.0 & \quad 0.0 & \quad 2.0 \\
123 & \quad 17.5 & \quad 0.0 & \quad 2.0 \\
201 & \quad 9.9144 & \quad -1.3053 & \quad 1.0 \\
203 & \quad 12.3931 & \quad -1.6316 & \quad 1.0 \\
205 & \quad 14.8717 & \quad -1.9579 & \quad 1.0 \\
207 & \quad 17.3503 & \quad -2.2842 & \quad 1.0 \\
209 & \quad 19.8289 & \quad -2.6105 & \quad 1.0 \\
210 & \quad 20.0 & \quad 0.0 & \quad 1.0 \\
211 & \quad 19.8289 & \quad 2.6105 & \quad 1.0 \\
213 & \quad 17.3503 & \quad 2.2842 & \quad 1.0 \\
215 & \quad 14.8717 & \quad 1.9579 & \quad 1.0 \\
217 & \quad 12.3931 & \quad 1.6316 & \quad 1.0 \\
219 & \quad 9.9144 & \quad 1.3053 & \quad 1.0 \\
220 & \quad 10.0 & \quad 0.0 & \quad 1.0 \\
\end{align*}$

$\begin{align*}
\text{DEFINE SURFACE BOUNDARY ELEMENTS}
\end{align*}$

$\begin{align*}
\text{SURFACE SURF1} \\
\text{TYPE QUAD}
\end{align*}$

$\begin{align*}
\text{ELEMENTS}
\end{align*}$

$\begin{align*}
1 & \quad 1 & \quad 20 & \quad 19 & \quad 18 & \quad 17 & \quad 21 & \quad 3 & \quad 2 \\
2 & \quad 3 & \quad 21 & \quad 17 & \quad 16 & \quad 15 & \quad 22 & \quad 5 & \quad 4 \\
3 & \quad 5 & \quad 22 & \quad 15 & \quad 14 & \quad 13 & \quad 23 & \quad 7 & \quad 6 \\
4 & \quad 7 & \quad 23 & \quad 13 & \quad 12 & \quad 11 & \quad 10 & \quad 9 & \quad 8 \\
5 & \quad 101 & \quad 102 & \quad 103 & \quad 121 & \quad 117 & \quad 118 & \quad 119 & \quad 120 \\
6 & \quad 103 & \quad 104 & \quad 105 & \quad 122 & \quad 115 & \quad 116 & \quad 117 & \quad 121 \\
7 & \quad 105 & \quad 106 & \quad 107 & \quad 123 & \quad 113 & \quad 114 & \quad 115 & \quad 122 \\
8 & \quad 107 & \quad 108 & \quad 109 & \quad 110 & \quad 111 & \quad 112 & \quad 113 & \quad 123 \\
9 & \quad 101 & \quad 201 & \quad 1 & \quad 2 & \quad 3 & \quad 203 & \quad 103 & \quad 102 \\
10 & \quad 103 & \quad 203 & \quad 3 & \quad 4 & \quad 5 & \quad 205 & \quad 105 & \quad 104 \\
11 & \quad 105 & \quad 205 & \quad 5 & \quad 6 & \quad 7 & \quad 207 & \quad 107 & \quad 106 \\
12 & \quad 107 & \quad 207 & \quad 7 & \quad 8 & \quad 9 & \quad 209 & \quad 109 & \quad 108 \\
13 & \quad 19 & \quad 219 & \quad 119 & \quad 118 & \quad 117 & \quad 217 & \quad 17 & \quad 18 \\
14 & \quad 17 & \quad 217 & \quad 117 & \quad 116 & \quad 115 & \quad 215 & \quad 15 & \quad 16 \\
15 & \quad 15 & \quad 215 & \quad 115 & \quad 114 & \quad 113 & \quad 213 & \quad 13 & \quad 14 \\
16 & \quad 13 & \quad 213 & \quad 113 & \quad 112 & \quad 111 & \quad 211 & \quad 11 & \quad 12 \\
17 & \quad 1 & \quad 201 & \quad 101 & \quad 120 & \quad 119 & \quad 219 & \quad 19 & \quad 20 & \quad 220 \\
20 & \quad 109 & \quad 209 & \quad 9 & \quad 10 & \quad 11 & \quad 211 & \quad 111 & \quad 110 & \quad 210 \\
\text{NORMAL 1 +}
\end{align*}$

$\begin{align*}
\text{DEFINE INSERT POINTS AND ELEMENTS}
\end{align*}$

$\begin{align*}
\text{INSERT 1000.0}
\end{align*}$

$\begin{align*}
\text{POINTS}
\end{align*}$

$\begin{align*}
401 & \quad 11.1538 & \quad 1.4684 & \quad 1. \\
402 & \quad 13.6324 & \quad 1.7947 & \quad 1. \\
403 & \quad 16.1110 & \quad 2.1211 & \quad 1. \\
404 & \quad 18.5896 & \quad 2.4474 & \quad 1. \\
501 & \quad 11.25 & \quad 0. & \quad 1. \\
502 & \quad 13.75 & \quad 0. & \quad 1. \\
503 & \quad 16.25 & \quad 0. & \quad 1. \\
504 & \quad 18.75 & \quad 0. & \quad 1. \\
\end{align*}$
HOLES, INSERTS, AND BODY FORCES EXAMPLE PROBLEM D.406

Input Data

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

SAMPLING POINTS
9020 10.0 0.0 0.0
9021 12.5 0.0 0.0
9022 15.0 0.0 0.0
9023 17.5 0.0 0.0
9024 20.0 0.0 0.0

$ 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.
7.0 APPLICATIONS

The analyses developed in BEST3D have been applied to a vast array of realistic engineering problems in references listed in Section 8. A few representative examples are described in the following sections:

7.1 Elastic Analysis

Multi-region cylinder
A column attached to an Elastic Half space
Turbine Component Analysis
A Cube with a Hole
Analysis of a thick cylinder with four Holes
Cube with a single Insert
Lateral Behavior of a Cube with Multiple Inserts
Thick Cylinder with Circular Insert Supports
Cube with Multiple Inserts with Random Orientation
Beam with Insert Reinforcement in Bending
Laminated Fiber Composite

7.2 Vibration Analysis

Free Vibration of a Rectangular parallelepiped
Calculation of Natural Frequency of Twisted Plates
Comparison with Finite Element Results for an Automotive component

7.3 Heat transfer Analysis

Tube and Disk Fin Heat Exchanger
Heat Transfer with Hole Elements
Cooling of a Steel Sphere
Injection Mold
Thermal Response of Turbine Blade

7.4 Dynamic Analysis

Dynamic Expansion of a Spherical Cavity
A Column subject to a Transient End Load
Dynamic Analysis of a buried Foundation
7.5 Non-Linear Stress Analysis

Benchmark Notch Specimen
Three-dimensional Analysis of a Notch Plate
Three-dimensional Analysis of a Perforated Plate
Multi-region Cylinder [33]:

In this problem a 22.5 degree sector of a thick cylinder was analyzed. In this case, several features are present. First, only twelve elements are used. Second, the structure is broken into two regions along the 11.25 degree plane. Third, the boundary conditions on the 0 and 22.5 degree planes have a cyclic symmetry between these two surfaces, rather than the roller boundary conditions. In this case only the results for quadratic variation of displacements and tractions are shown. Good agreement between the calculated and analytical results was obtained (Figure 7.1.1).

<table>
<thead>
<tr>
<th>PRESSURE</th>
<th>ID</th>
<th>PRESSURE</th>
<th>PRESSURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
</tr>
<tr>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
</tr>
<tr>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
</tr>
<tr>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
<td>$\delta_R$</td>
</tr>
</tbody>
</table>

$P = 10126$ PSI

$\omega = 1000$ RAD/SEC

$\rho = 0.0007$ LBM/IN$^3$

Fig. 7.1.1 Thick cylinder - Multi-region
A Column Attached to an Elastic Half-Space [33]:

In this problem a long column is attached to an elastic half-space and loaded in tension. Note that the correct solution for the tip deflection of the column is essentially the sum of the extension of the column and the displacement of the half-space under a patch load.

It is of particular interest to note that it was never possible to obtain an acceptable solution to this problem when it was run as a single region, using either linear or quadratic variation. This is due to the fact that, when considered as a single region, much of the column is effectively located at infinity. To obtain accurate results would require a much more extensive mesh on the surface of the half-space, losing all the advantage gained by the use of infinite elements. Assigning the column and the half-space to separate regions eliminates this problem, leading to reasonable results (-5% error) for linear variation and very good results (less than 1% error) for quadratic variation (Figure 7.1.2).
In order to evaluate the capabilities of the analysis of real components, an analysis of a commercial cooled turbine blade geometry was carried out.

The blade analyzed is a cooled high turbine blade presently in service. It is subject to mechanical loads (primarily centrifugal) and thermal loads. Of particular interest for this blade is the location and magnitude of the peak stress under the platform. A surface model was built for this problem. Both full and hidden line views of the model are shown in Figure 7.1.3a. The model consists of five subregions. The interfaces between subregions are generally perpendicular to the radial direction. The characteristics of the model are summarized below.

<table>
<thead>
<tr>
<th>Subregion</th>
<th>Elements</th>
<th>Linear (Nodes)</th>
<th>Quadratic (Nodes)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>60</td>
<td>61</td>
<td>180</td>
</tr>
<tr>
<td>2</td>
<td>86</td>
<td>85</td>
<td>256</td>
</tr>
<tr>
<td>3</td>
<td>107</td>
<td>98</td>
<td>303</td>
</tr>
<tr>
<td>4</td>
<td>106</td>
<td>96</td>
<td>298</td>
</tr>
<tr>
<td>5</td>
<td>80</td>
<td>76</td>
<td>232</td>
</tr>
<tr>
<td>TOTAL</td>
<td>439</td>
<td>416</td>
<td>1269</td>
</tr>
</tbody>
</table>

The system equations for a fully linear analysis contain 1248 equations.
Initially a fully linear analysis under centrifugal load was carried out. The total centrifugal load at various spanwise stations on the blade was compared with the design calculations for the blade. The agreement between the two totally independent calculations was excellent.

Study of blade tip deflections, load distribution over the base of the blade neck and concentrated stresses indicated reasonable qualitative agreement with three-dimensional finite element results. The time required to execute was only 15 cpu min (on an IBM 3081) compared to 45 cpu min for the finite element (MARC) analysis. While the use of stress contour plots demonstrated that the peak stress in the analysis occurred in the correct location (under the trailing edge rail on the concave side of the blade), the value of the peak stress was too low (146 ksi vs. 169 ksi). Improving this result using a full quadratic analysis would have involved over 3700 degrees of freedom and required 45 to 60 minutes of computing time.

In order to improve local stress accuracy in the critical location while retaining computational efficiency, the mixed variation capability was employed. Quadratic variation was used over sixteen elements (in only two subregions) in the immediate vicinity of the critical rail. The extent of the quadratic variation is shown in Figure 7.1.3b Problem size increased only 6% (from 1248 to 1317 degrees of freedom) and computer time only 10% (to 16.5 cpu min.). The peak stress increased to 174 ksi (on the surface), in excellent agreement with the MARC result of 169 ksi (at a slightly subsurface integration point).

Fig. 7.1.3b Local Quadratic Surface Elements for the BEST3D Model of a Cooled Turbine Blade
A cube with a hole [54]:

A test problem of a cube of dimensions ‘a’ with a cylindrical hole of diameter ‘b’ was subjected to simple tension such that the deformed shape of the hole is elliptical (Figure 7.1.4a). Figure 7.1.4b shows the discretization used which includes a mixture of 8-noded and 9-noded surface elements. The hole, which is of constant radius, was modeled using the hole element with three nodes in the longitudinal direction. Young’s modulus = 100 psi, \( \nu = 0.3 \) and an applied traction of 100 psi were assumed.

Figure 7.1.4c shows the results of BEST3D analysis for various dimensions of the hole compared with BEST3D results in which the hole was modeled exactly. The diameter of the hole was varied from \( b/a = 0.01 \) to \( b/a = 0.6 \). It can be seen that the displacement at the middle node of the loaded face of the cube is in good agreement with the exact 2D modeling. The discrepancy at the larger void ratio is to be expected since the displacement field around the hole is too complex to be modeled by the coarse surface mesh.

Figure 7.1.4d shows the comparison of the displacement variation at the loaded face of the cube. Once again the results for the \( b/a = 0.1 \) case are found to be in good agreement with the results of BEST3D exact modeling. As expected, results at the middle section of the cube (i.e., nodes 1b, 2b and 3b) are in better agreement with the 2D results.

It was originally thought that the hole element, while perfectly adequate to quantify the loss of stiffness due to their presence, would not be suitable for the determination of the local stress concentration factors. Figure 7.1.4e shows that the actual stress concentration up to a point close to hole surface is quite well predicted and with some additional work, it should be nearly as good as the full 3D idealization.

\[ \text{Fig. 7.1.4a Three Dimensional Cube (of length a) with a hole (of diameter b) at the center} \]
Fig. 7.1.4b Discretization of the cube utilizing quadratic hole element to model the hole.

Fig. 7.1.4c Comparison between exact 2-D modelling of the cube and Hole element results for different hole diameters.
Elastic Analysis

Fig. 7.1.4d Comparison of edge displacements as given by 2-D BEM and the Hole Elements in BEST3D

Fig. 7.1.4e Comparison of the Stress Distribution around the Hole between 2-D models and 3-D Hole Element Model
Analysis of a thick cylinder with four holes [54]:

A more complex problem demonstrating the facilities of the current implementation is shown in Figure 7.1.5a, where a 22° segment of a thick cylinder (inner radius = 10 inches, outer radius = 20 inches, height of the cylinder = 2 inches) has four 1 inch diameter holes evenly spaced at radii of \( r = 11.25, 13.75, 16.25 \) and 18.75. The Young’s modulus of the material is \( E = 30 \times 10^6 \) psi and the Poisson’s ratio \( \nu = 0.3 \). The problem is first modeled with 18 quadratic surface elements and 4 quadratic hole elements. Sixteen (16) of the quadratic surface elements are 8-noded and 2 are 9-noded elements. The cylindrical segment is loaded with an internal pressure of 100,000 psi under the conditions of plane strain, which was simulated by applying rollers on all plane surfaces.

The problem was compared to an axisymmetric, multi-region BEM analysis (using a 2-D code) with a mesh shown in figure 7.1.5b. The radial displacements along the bottom surface is shown in Figure 7.1.5c. Both the axisymmetric and three-dimensional results with holes are in excellent agreement. For the sake of comparison, the results of the solid cylinder are also shown in the graph.

![Fig. 7.1.5a Multiple Holes in an Axisymmetric Three-dimensional Block](image-url)
Fig. 7.1.5b Axisymmetric Discretization of the Block with Four Holes

Fig. 7.1.5c Comparison between Axisymmetric results and 3-D modelling with Hole Elements for the 4-hole Block
Cube with a Single Insert [55]:

The first test of the formulation is on a unit cube with a single insert through its center of radius 0.1. 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. 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. 7.1.6a, 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. 7.1.6b 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. 7.1.6c, 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 \( E_i/E = 10 \). For the case \( E_i/E = 100 \), the insert formulation exhibits greater stiffness than the 3D results. This difference is contributed by the way the load is distributed from the insert to the composite matrix. In the full 3D 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 body 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 3D analysis.

In Fig. 7.1.6d, 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 \( E_i/E = 10 \), and deviates slightly from the full 3D results in the second case.

In Figs. 7.1.6e and 7.1.6f, the lateral displacements along the side of the cube are shown for a cube subjected to shear traction of 100. For the case of applied shear perpendicular to the insert (Fig. 7.1.6e), the results for both the insert and full 3D model show good agreement. Once again a slight deviation is observed for \( E_i/E = 100 \). In the case of the shear traction in the plane of the insert (Fig. 7.1.6f) the insert has little effect on the displacement (as anticipated) and all results fall in close proximity.
Fig. 7.1.6a Discretization of an Insert in a Unit Cube Utilizing Quadratic Insert Elements

Fig. 7.1.6b Full Three-Dimensional Multi-region Discretization of an Insert in a Unit Cube
Fig. 7.1.6c Comparison of Displacement Profiles between the full 3-D model and the Insert Element Model for the End of a Cube in Tension

Fig. 7.1.6d Axial Stress through the Cross Section of a Unit Cube in Tension with a Single Insert in Parallel with the Loading
Fig. 7.1.6e Lateral Displacement along a side of a Cube subjected to a Shear Force Perpendicular to an Insert

Fig. 7.1.6f Lateral Displacement along a side of a Cube subjected to a Shear force in the Cross-Plane of an Insert
Lateral Behavior of a Cube with Multiple Inserts [55]:

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. 7.1.7a) 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

\[ E^{\text{insert}} = 10000, \quad E^{\text{matrix}} = 100. \]
\[ \nu^{\text{insert}} = 0.3, \quad \nu^{\text{matrix}} = 0.3 \]

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 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 for a cube with one insert and five inserts are shown in Figs. 7.1.7b and 7.1.7c. The results seem to be in good agreement with the two-dimensional results. The 2D results are approximations since plane stress is assumed. The 3D solutions for the one insert are within 2% error of the 2D solution and 6% for the case of nine inserts where the insert of volume to total volume ration is 28.2%. The result is also displayed in a plot of Effective Modulus vs. Insert Volume Ratio in Fig. 7.1.7d. The effective modulus is defined as the average stress/average strain. The three-dimensional results follow closely to the two-dimensional solution.
Fig. 7.1.7a Arrangement of Multiple Inserts in a Unit Cube Subjected to Lateral Tension
Elastic Analysis

Fig. 7.1.7b Displacement Profile of a Cube with a Single Insert under Lateral Tension

Fig. 7.1.7c Displacement Profile of a Cube with Five Inserts under Lateral Tension
Fig. 7.1.7d Effective Transverse Modulus of a Cube as a Function of Insert Volume to Total Volume
Thick Cylinder with Circumferential Insert Supports [55]:

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 with four inserts is analyzed. The inner and outer radii of the 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. 7.1.8a, 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 (using a 2-D code) were used for comparison with the 3D 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. 7.1.8b). The radial displacement through the thick cylinder along the top face is shown in fig. 7.1.8c 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. 7.1.8d, 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 3D insert model is in good agreement with the axisymmetric results for all cases. In Fig. 7.1.8e, the radial stress is displayed for the two models. The inserts have little effect on this stress and the curves for the five moduli fall close together for both approaches.
Fig. 7.1.8a Discretization of a Thick Cylinder with Four Inserts Utilizing Insert Elements

Fig. 7.1.8b Axisymmetric Multi-region, Discretization of a Thick Cylinder with Four Inserts
Fig. 7.1.8c Radial Displacement Through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0, 2.5, 5.0, 7.5, 10.0$

Fig. 7.1.8d Circumferential Stress Through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0, 2.5, 5.0, 7.5, 10.0$
Fig. 7.1.8e Radial Stress through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0, 2.5, 5.0, 7.5, 10.0$
Cube with Multiple Inserts with Random Orientation [55]:

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. 7.1.9a). 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. 7.1.9b-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 in 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. 7.1.9g). The displacement decreases with increasing number of inserts per cube and increasing $E_i/E$ values as expected.
Fig. 7.1.9 (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. 7.1.9g End Displacement of a Unit Cube Used in the Study of Random Oriented Inserts of $E_I/E = 5, 10, 15, 20,$ and 25
Beam with Insert Reinforcement in Bending [55]:

In the last example, the applicability of the present formulation to the study of the micro-mechanical behavior of the ceramic composite is apparent. The present formulation, however, is equally applicable to typical problems encountered by civil engineers. Using Composite-BEST 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. 7.1.10a, a 4x1x1 beam with four inserts is modeled using twenty-eight quadratic boundary elements. 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 analysis is shown in Fig. 7.1.10b as a function of Ei/E. The non-dimensional displacement is defined here as the end displacement of the reinforced beam divided by the displacement of a homogeneous beam under similar conditions.

The end displacement obtained from the mechanics of material solution is also displayed in Fig. 7.1.10b 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.

![Fig. 7.1.10a Discretization of a Reinforced Beam Utilizing Quadratic Insert Elements to Model the Four Inserts](image-url)
Fig. 7.1.10b Non-dimensionalized Vertical End Displacement of a Reinforced Beam in Bending vs. the Modulus of the Insert over the Modulus of the Beam
Laminated Fiber Composite [55]:

A laminated composite fabricated from a fiber composite material is shown in Fig. 7.1.11a. 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. 7.1.11b. 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 allows for sliding and spring connections also.

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 vary between 100 and 10,000. The Poisson ratio is 0.3 for both the composite matrix and inserts at all times.

Figure 7.1.11c 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.
Fig. 7.1.11b BEM Discretization of a Laminate-Fiber Composite

Fig. 7.1.11c Lateral Displacement of a Point at the corner of the interface of a Laminate-Fiber Composite under Bi-Axial Tension
Free-vibration of a rectangular parallelepiped [53]:

Mesh sensitivity studies for the BEST3D natural frequencies and mode shapes capability were done using three-dimensional solutions for cantilevered rectangular parallelepipeds obtained by Leissa and Zhang [56] using a Ritz technique. In their paper they report the first five modes of each type (torsion, easy bending, stiff bending and extension) for each of five geometries. Convergence studies indicate that their results are accurate to better than 1% for the first mode.

One of the cases studied is a parallelepiped (Fig. 7.2.1) of dimensions 1 x 1 x 0.5. One end of the structure is completely fixed in all three directions. Four single region, quadratic BEST3D models, summarized in Table 7.2.1a were used in the study. The results are tabulated (Table 7.2.1b) with mode type and variation from the Ritz solution listed. It can be seen that rapid convergence is achieved for the first mode of each type by the addition of elements in the axial direction. Convergence of higher modes requires the use of additional elements on the lateral surface of the parallelepiped.

\[ E = 16.126 \times 10^8 \text{ psi} \]
\[ v = .3 \]
\[ p = .0007 \text{ lbm/in}^3 \]
\[ \omega = \text{rad/sec} \]

Fig. 7.2.1 Geometry for Calibration of BEST3D Natural Frequency Analysis
Table 7.2.1a BEST3D Modes for 1 X 1 X .2 Parallelepiped

<table>
<thead>
<tr>
<th>Model</th>
<th>Elements Along Side</th>
<th>Total Elements</th>
<th>Source Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1 1 1</td>
<td>6</td>
<td>20</td>
</tr>
<tr>
<td>2</td>
<td>1 2 1</td>
<td>10</td>
<td>32</td>
</tr>
<tr>
<td>3</td>
<td>1 3 1</td>
<td>14</td>
<td>44</td>
</tr>
<tr>
<td>4</td>
<td>2 3 1</td>
<td>22</td>
<td>68</td>
</tr>
</tbody>
</table>

Table 7.2.1b Convergence of BEST3D Natural Frequency Results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Type</th>
<th>Ritz</th>
<th>Mesh 1 (%)</th>
<th>Mesh 2 (%)</th>
<th>Mesh 3 (%)</th>
<th>Mesh 4 (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>EB</td>
<td>0.447</td>
<td>0.472</td>
<td>5.5</td>
<td>0.429</td>
<td>-0.4</td>
</tr>
<tr>
<td>2</td>
<td>SB</td>
<td>0.667</td>
<td>0.664</td>
<td>-0.4</td>
<td>0.666</td>
<td>-0.3</td>
</tr>
<tr>
<td>3</td>
<td>T</td>
<td>0.788</td>
<td>0.887</td>
<td>12.5</td>
<td>0.829</td>
<td>5.1</td>
</tr>
<tr>
<td>4</td>
<td>L</td>
<td>1.596</td>
<td>1.625</td>
<td>1.8</td>
<td>1.620</td>
<td>1.5</td>
</tr>
<tr>
<td>5</td>
<td>EB</td>
<td>1.664</td>
<td>2.136</td>
<td>28.3</td>
<td>1.797</td>
<td>8.0</td>
</tr>
<tr>
<td>6</td>
<td>SB</td>
<td>1.774</td>
<td>1.836</td>
<td>3.5</td>
<td>1.789</td>
<td>0.9</td>
</tr>
<tr>
<td>7</td>
<td>T</td>
<td>2.220</td>
<td>2.552</td>
<td>14.9</td>
<td>2.448</td>
<td>10.3</td>
</tr>
<tr>
<td>8</td>
<td>EB</td>
<td>2.278</td>
<td>3.033</td>
<td>33.3</td>
<td>2.365</td>
<td>3.8</td>
</tr>
<tr>
<td>9</td>
<td>L</td>
<td>2.797</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>SB</td>
<td>3.068</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Mode Type Identification

EB - Easy Bending
SB - Stiff Bending
T - Torsion
L - Extension

Frequency Parameter = \( w \sqrt{\rho/E} \)
Calculation of Natural Frequencies for Twisted Plates [53]:

While the analyses discussed in the previous section demonstrated conclusively the ability of the boundary element method to calculate natural frequencies for three-dimensional bodies, the geometries involved were not of practical interest. In order to calibrate the method for more realistic geometries a number of twisted plate configurations were studied in considerable detail.

The geometries considered were used in the NASA sponsored Joint Research Effort on Vibrations of Twisted Plates. In this study four different combinations of plate length and thickness were studied, at each of five twist angles (equally spaced from $0^\circ$ through $60^\circ$). Specimens were machined and tested (at two laboratories) for each configuration. In addition, a large number of investigators made analytical predictions of the natural frequencies and mode shapes, mostly using finite element programs. The complete details of the experimental program are given in [57] and the various analytical results are collected in [58].

Because the three-dimensional boundary element method is directed primarily at the solution of truly three-dimensional problems, rather than the analysis of plates or shells, the most three-dimensional of the NASA plate geometries ($2\text{in.} \times 2\text{in.} \times .4\text{in.}$) was selected. A variety of BEST3D models was used, with that shown in Figure 7.2.2a finally chosen for detailed comparisons at all five twist angles.

![Two Region BEST3D Model of 60° Twisted Plate](image)
The twisted plate was modelled using two identical substructures. The calculated natural frequencies were compared with both experimental (when available) and finite element results. Four sets of finite element results were chosen for the comparison, since they gave the best results for this specimen configuration. The various analytical and experimental results considered and the designations given to them in the following figures are summarized in Table 7.2.2 In all of the figures the nondimensional frequency parameter, $\omega a^2 \sqrt{\rho h/D}$, is plotted against twist angle, where $\omega$ is the natural frequency (rad/sec), $a$ is the plate axial length, $\rho$ is mass density, $h$ is plate thickness and $D$ is the plate flexural rigidity.

<table>
<thead>
<tr>
<th>Designation</th>
<th>Type</th>
<th>Description</th>
<th>Degrees of Freedom</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEL</td>
<td>finite element</td>
<td>parabolic conoidal shell</td>
<td>252</td>
</tr>
<tr>
<td>FEM</td>
<td>finite element</td>
<td>quadrilateral thick plate</td>
<td>294</td>
</tr>
<tr>
<td>FEN</td>
<td>finite element</td>
<td>eight node solid isoparametric</td>
<td>660</td>
</tr>
<tr>
<td>FEO</td>
<td>finite element</td>
<td>sixteen node solid isoparametric</td>
<td>1872</td>
</tr>
<tr>
<td>BE</td>
<td>boundary element</td>
<td>BEST</td>
<td>288</td>
</tr>
<tr>
<td>EXA</td>
<td>experimental</td>
<td>NASA-LeRC</td>
<td></td>
</tr>
<tr>
<td>EXB</td>
<td>experimental</td>
<td>Air Force Aero Propulsion Lab.</td>
<td></td>
</tr>
</tbody>
</table>

Table 7.2.2 Analytical and Experimental Results used in Twisted Plate Comparison

As Figures 7.2.2b and 7.2.2c indicate, there is considerable scatter among the four sets of finite element results. The BEST3D results clearly fit within the range of the finite element results. BEST3D shows the closest agreement with the finite element results obtained using thick plate elements (method M) and sixteen node solid isoparametric elements (method O). The only analytical results available are the natural frequencies for the first bending and torsion modes for the untwisted plate. For these two modes BEST3D and the thick plate finite element method show closer and more consistent agreement than any of the other methods.

Experimental results are also shown in Figures 7.2.2b and 7.2.2c. They are consistently lower than the analytical values for all modes. The lower experimental values are due to the compliance of the specimen base and fixturing. For the first two modes, both BEST3D and the thick plate finite element method correctly predict the variation of natural frequency with twist angle, although the finite element method overpredicts the variation for the first torsion mode.

It is clear that BEST3D predicts the twisted plate response at least as consistently as any of the finite element models. Final calibration of the method against experimental data required the construction of a BEST3D model containing at least a crude representation of the specimen base.
Fig. 7.2.2b First Bending Mode for 2 x 2 x .4 Plate

Fig. 7.2.2c First Torsion Mode for 2 x 2 x .4 Plate
The **BEST3D** model shown in Figure 7.2.2d was used to calculate the natural frequencies, at all five twist angles. The model consisted of three substructures, two for the plate and one for the specimen base. The lateral surface of the specimen base was completely fixed, to model the clamped condition intended in the experimental apparatus. As shown in Figures 7.2.2e and 7.2.2f, much better agreement with the experimental results was obtained, even with a relatively crude representation of the specimen base and clamping conditions.

Sub structuring was found essential in order to obtain satisfactory results with surface meshes of reasonable density. This was true for the plate models, and was even more pronounced when the entire structure, including the base, was modelled. In part this simply reflects the well known advantages of sub structuring in the static case, especially for the more complicated displacement and stress fields characteristic of bending. A further effect of sub structuring in the natural frequency problem is the fact that it effectively introduces interior nodes into the problem. Since the ρω²u term cannot be exactly converted to boundary integrals, the introduction of interior sampling points (even indirectly, as boundary nodes on an interface) can be expected to produce significant improvement in the results.

---

**Fig. 7.2.2d BEST3D Model of 2 x 2 x 4 Plate Including Base**
Fig. 7.2.2e Effect of Specimen Base on First Bending Frequency

Fig. 7.2.2f Effect of Specimen Base on First Torsion Frequency
Comparison with Finite Element Results for an Automotive Component [53]:

The final example in this paper concerns the determination of the natural frequencies of an automobile crankshaft. A three-dimensional boundary element eigenvalue analysis was done using BEST3D. The boundary element mesh for this analysis is shown in Figure 7.2.3. A three-dimensional finite element analysis was also done, using a mesh in which the surface breakup was identical to the BEST3D model.

The results obtained from the two models are compared in Table 7.2.3a, where excellent agreement can be seen for the first, second and fourth modes (as determined by the BEST3D analysis). Computing times, on a CRAY XM-P, are summarized in Table 7.2.3b.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Finite Element</th>
<th>BEST</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>17400</td>
<td>18200</td>
</tr>
<tr>
<td>2</td>
<td>35800</td>
<td>34300</td>
</tr>
<tr>
<td>3</td>
<td>—</td>
<td>47500</td>
</tr>
<tr>
<td>4</td>
<td>66300</td>
<td>66300</td>
</tr>
</tbody>
</table>

Table 7.2.3a Natural Frequencies (Hz) for Automobile Crankshaft

<table>
<thead>
<tr>
<th></th>
<th>Finite Element</th>
<th>BEST</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU (sec)</td>
<td>115</td>
<td>230</td>
</tr>
<tr>
<td>I/O (sec)</td>
<td>402</td>
<td>44</td>
</tr>
</tbody>
</table>

Table 7.2.3b Computing Times for Crankshaft Natural Frequency Analysis

Initial concern over a mode identified by BEST3D, but not by the finite element analysis, was resolved when a somewhat more refined finite element model gave a natural frequency at 47600 Hz, while leaving the first, second and fourth frequencies essentially unchanged. CPU time for the refined finite element analysis was very close to that for the BEST3D analysis.
In addition to providing further verification of the three-dimensional boundary element method free vibration capability, this example provides ample illustration of the desirability of having more than one analytical technique available for the solution of real engineering problems. In dealing with actual components the correct solution is not known a priori, experimental results are often difficult to obtain and intuition may well fail. In such cases only the use of multiple analyses can give confidence in the calculated results.
Tube and Disk Fin Heat Exchanger [42]:

This problem is the heat transfer analysis of a portion of a heat exchanger. Figure 7.3.1a shows a section of a thin fin attached to a thick tube carrying hot fluid. The tube (ID = 0.275 inch, OD = 0.375 inch) is modelled using 22 quadratic boundary elements. The thin (0.02 inch) fin is represented using 18 quadratic elements. Both regions have the same thermal conductivity of 25 in-lb/sec-in-°F. The tube carries a fluid with a temperature of 300°F and a convection coefficient of 60 lb/sec-in-°F. The fin is exposed to air at 100°F, with a convection coefficient of 15 lb/sec-in-°F. Since manufacturing processes cannot ensure perfect contact between the tube and the fin, various values of the thermal resistance at the disk-fin interface (0, 0.001, 0.005 and 0.02 in-sec °F/lb) were employed in a parametric study. Figure 7.3.1b shows the temperature distribution along a radial line through the tube and disk for the various values of thermal resistance. As expected, the temperature distribution becomes continous across the interface when the thermal resistance vanishes.
Fig. 7.3.1b Tube and disk fin heat exchanger
Results

BEST3D User Manual
April, 1989
Page 7.41
Heat Transfer with Hole Elements [42]:

A new element class has been developed for the heat transfer version of BEST3D. These 'hole elements' are designed for the representation, without an explicit surface mesh, of cavities of cylindrical cross-section. The centerline of a single tube element is defined as a two- or three-noded isoparametric curve in three-space. Temperature and flux are allowed to vary along the centerline, but circumferential variation is ignored. Hole elements may be joined end-to-end, to improve definition of variation along the centerline or to allow representation of complex hole geometries.

In order to evaluate the accuracy and efficiency of the tube elements compared to explicit modelling, a model problem consisting of three cylindrical holes in a cube was solved. The tube length and radius are, respectively, 80% and 1% of the cube dimension. The location of the cavities in the cube is shown in Figure 7.3.2a. The front and back faces of the cube are maintained at 0 ° and 100°F. The cavity surface is subject to convection with a film coefficient of 5 lb/sec in °F and an ambient temperature of 200°F. The surface mesh for the cube consists of six quadratic boundary elements. In one analysis, each cavity was explicitly modeled using 48 linear boundary elements yielding a mesh with 150 elements, 422 geometric nodes and 152 source points. In the second analysis, each cavity was represented using 2 tube elements, giving a mesh of 6 surface elements, 6 tube elements, 35 geometric nodes and 35 source points. The analysis using tube elements required only 5% of the computer time needed for the explicit analysis.

The temperature variation along the tube for the two analyses is shown in Figure 7.3.2b. The tube element analysis almost exactly reproduces the much more expensive results of explicit modelling. Comparison of results on the surface of the cube shows agreement of unknowns (either temperature or flux) to better than 1%.

Fig. 7.3.2a Cube with embedded cylindrical holes Boundary Element Model
Fig. 7.3.2b Cube with embedded cylindrical holes
Temperature along central exclusion
Cooling of a Steel Sphere [59]:

A solid steel sphere of radius 1.5 in., initially at a uniform temperature of 100°F, is slowly cooled in 0°F air by convection. The exact solution to this problem is given by Carslaw and Jaeger (1947) as

\[ \theta(r, t) = \frac{2H \theta_0}{r} \sum_{n=1}^{\infty} e^{-\alpha_n^2 r} \frac{R^2 \alpha_n^2 + (RH - 1)^2}{\alpha_n^2 [r^2 \alpha_n^2 + RH(RH - 1)]} \sin \alpha_n \sin \alpha_n \]  

(7.3.1)

where \( \theta_0 \) is the initial temperature, \( c \) is the diffusivity, \( R \) is the outer radius, and \( \alpha_n \) are the roots of

\[ RacotRa + RH - 1 = 0. \]  

(7.3.2)

Furthermore,

\[ H = \frac{h}{k} \]  

(7.3.3)

in which \( h \) is the convection film coefficient and \( k \) is the thermal conductivity. For positive \( H \), all the roots of (7.3.2) are real. To complete the problem specification, let

\[ c = 0.0205 \text{in.}^2/\text{sec.}, \]

\[ k = 25 \text{in.} - \text{lb/sec.in.}^\circ\text{F}, \]

\[ h = 1.25 \text{in.} - \text{lb/sec.in.}^2\text{F}. \]

Three levels of mesh refinement are considered to study convergence of the boundary element solution. In each case, only the positive octant of the sphere is modeled, while symmetry constraints are imposed. Figures 7.3.3a - 7.3.3c illustrate the boundary element idealizations employed in this analysis. Note that the plotting package, used to produce these plots, connects all nodes by straight lines, however, within BEST3D, a quadratic variation of geometry is utilized for each surface element. The results from BEST3D, for a time step of 1.25 sec., are compared with the exact solution in Figure 7.3.3d for points on the surface and at the center of the sphere. Correlation is good, even for the three element model which never deviates from the exact solution by more than 2°F. Dramatic improvement occurs for the four element mesh, but then diminishing returns are evident upon further refinement. However, the trend toward convergence is apparent from the graph.
Heat Transfer Analysis

COOLING OF A STEEL SPHERE
Boundary Element Model

Figure 7.3.3a

COOLING OF A STEEL SPHERE
Boundary Element Model

Figure 7.3.3b

BEST3D User Manual
April, 1989
Page 7.45
COOLING OF A STEEL SPHERE
Boundary Element Model

Figure 7.3.3c

COOLING OF A STEEL SPHERE
3d Results

Figure 7.3.3d
Injection Mold [42]:

In this problem, the steady-state temperature distribution in an injection mold is investigated. The specific mold under consideration is used to manufacture semicircular thermoplastic tanks. The tanks have an inner radius of 1.0 in., a thickness of 0.25 in., and an overall length of 5.5 in. Meanwhile, the mold itself is quite simple. It consists of two steel components, a core and a cavity, with combined outer dimensions of 15 in. x 20 in. x 20 in. Several 0.25 in. diameter cooling lines, running parallel to the length of the tank, are provided in both components.

The boundary element model of the positive quadrant of the mold is depicted in Figure 7.3.4a. In that diagram, the core and cavity are shown individually as separate GMR’s, which are viewed from different vantage points. Thirty-two quadratic surface elements model the core, while the cavity is discretized into twenty-nine elements. Additionally, four quadratic hole elements are employed to represent each cooling line. The location of these cooling lines are defined in the cross-sectional slice displayed in Figure 7.3.4b.

Convective heat transfer is assumed on all exposed surfaces. Specifically, on the surface of the tank \( h = 25 \text{ in.}-\text{lb./in.}^2\text{sec} F \) and \( \theta_{\text{amb}} = 450\degree \), while for the cooling lines \( h = 50 \text{ in.}-\text{lb./in.}^2\text{sec} F \) and \( \theta_{\text{amb}} = 120\degree \). A much lower convection coefficient (\( h = 0.1 \text{ in.}-\text{lb./in.}^2\text{sec} F \)) is assumed on the outer mold surfaces which are in contact with 100°F ambient air. Also, the thermal resistance of 0.01 in.\(^2\)F/in.-lb. is imposed across the imperfect core to cavity interface. The conductivity of steel is taken as 5.8 in.-lb./sec.in.*F.

The resulting steady-state temperature contours are displayed in Figure 7.3.4c. Notice that the highest temperatures occur near the crown of the tank, while the largest thermal gradients appear near the foot. Also, as is evident from Figure 7.3.4d, there are significant temperature differences between the inner (core side) and outer (cavity side) surfaces, which may lead to the development of residual stresses, and subsequently, warpage in the molded tank.

In order to provide a more uniform surface temperature profile, typically, the cooling lines must be repositioned and/or resized. Each design iteration can be examined quite efficiently with the present boundary element approach. In fact, once the outer surface of the mold is defined, at each iteration, the mold designer is only required to position the cooling hole centerline nodes, and provide the appropriate boundary conditions. On the other hand, it is difficult to visualize the utility of a finite element or finite element difference approach, which would require the generation of an extremely complicated mesh for a three-dimensional solid with an irregular outer boundary and a number of embedded holes.
Heat Transfer Analysis

Figure 7.3.4a Injection Mold - Boundary Element Model
Figure 7.3.4b Cross-section - Injection Mold
Figure 7.3.4c Temperature Contours - Injection Mold
Figure 7.3.4d Temperatures on Tank Surface - Injection Mold
Thermal Response of a Turbine Blade [33]:

In this example, the transient thermal response of a turbine blade is examined during a warm startup. The boundary element surface model of the component is shown in Figure 7.3.5a. The model is composed of three GMR's with a total of 128 elements and utilizes symmetry about the x=0 plane.

The initial state is first determined with a steady-state thermal analysis, conducted under the following conditions. The exposed portion of the blade, as indicated in Figure 7.3.5a, is heated via convection by a gas at 1170°C. Convection coefficients are a function of position along the surface of the blade and are plotted versus the chord length in Figure 7.3.5b. Meanwhile, the lower blade and cylinder edges are heated by a 500°C gas and the cylinder base is exposed to a 300°C fluid. In both of these latter instances, a relatively low convection coefficient of 0.000050 W/mm°C is specified. The tip of the blade is completely insulated. Additionally, the thermal material properties are specified as follows:

\[
\begin{align*}
K &= 0.0216 \text{ W/mm°C} \\
C &= 6.14 \text{ mm}^2/\text{sec}.
\end{align*}
\]

The resulting steady-state temperature distribution is presented in Figure 7.3.5c. Peak temperatures occur near the tip, however the highest thermal gradients appear near the blade-to-cylinder junction. Next, the turbine blade is analyzed under transient conditions. For this case, the gas temperatures vary according to the profiles depicted in Figure 7.3.5d. Convection coefficients remain unchanged from the steady-state values detailed previously. The thermal response of the component, as determined with a time step of 0.25 sec., is provided at various times in Figures 7.3.5e-h. Notice that the blade cools dramatically during the initial two seconds, and produces relatively sharp thermal gradients, particularly near the leading edge. Then, as the gas temperature surrounding the blade continues to rise, the peak gradients are again shifted toward the blade-to-cylinder junction. Of course, all of these results could next be used as input for a thermal stress analysis.
Fig. 7.3.5a Thermal Response of a Turbine Blade
Boundary Element Model
Fig. 7.3.5b Thermal Response of a Turbine Blade
Convective Coefficients
Fig. 7.3.5c Thermal Response of a Turbine Blade
Steady-state Temperatures
THERMAL RESPONSE OF A TURBINE BLADE

Ambient Gas Temperatures

- Exposed Blade
- Cylinder Edges
- Cylinder Base

Fig. 7.3.5d Thermal Response of a Turbine Blade
Ambient Gas Temperatures
Fig. 7.3.5e Thermal Response of a Turbine Blade
Transient Temperatures at \( t = 1 \) sec.
Fig. 7.3.5f Thermal Response of a Turbine Blade
Transient Temperatures at $t = 2$ sec.
Fig. 7.3.5g Thermal Response of a Turbine Blade
Transient Temperatures at $t = 5$ sec.
Fig. 7.3.5h Thermal Response of a Turbine Blade
Transient Temperatures at $t = 10$ sec.
Dynamic Expansion of a Spherical Cavity [26,28]:

The problem of the dynamic radial expansion of a spherical cavity within an infinite solid was used as a test problem to validate both the Laplace transform transient dynamic analysis and the time domain transient analysis. The radius of the cavity was assumed to 212 inches, and the material parameters were Young's modulus = 8.993 x 10^6, = 0.25, mass density = 0.00025. A radial pressure of 1000 psi was suddenly applied and maintained.

Three different discretizations used to analyze this problem are shown in Figure 7.4.1a. Figure 7.4.1b shows the Laplace transform and the transient time domain results for mesh 2 compared with the exact analytical solution. In general, the numerical results are in good agreement with the analytical solution. These is some oscillations in the results of the Laplace transform solution which becomes progressively worse beyond t = 0.0056 secs. The transient time domain solution, on the other hand, remains stable and reaches the expected static solution at larger times. The numerical errors in the displacement-time response characteristics for the meshes 1, 2 and 3 were about 15%, 9% and 6%, respectively.

Fig. 7.4.1a Discretization of the surface of a spherical cavity
Fig. 7.4.1b Comparison of the Laplace transform and the time domain transient results with the analytical solution.
A Column Subjected to a Transient End Load [26,27]:

A column (2 x 2 x 8) held along its sides by lubricated rollers and supported on a frictionless base is subjected to a suddenly applied pressure of 1000 at the free end. The material properties are Young’s modulus = 1x10^5 and Poisson’s ratio = 0. Figure 7.4.2 shows the discretization and numerical results for two time step sizes compared with the exact analytical solution for one-dimensional stress wave propagation. Although the numerical results are in good agreement with the analytical solution it is clearly very difficult to reproduce the sharp jump in the stress as the disturbance reaches the point initially and when the reflected stress wave returns to the same location.

---

Fig. 7.4.2 Longitudinal stress at the mid-length of a column subjected to a suddenly applied end load
Dynamic analysis of a buried foundation [61]:

A number of meshes used are shown in Figure 7.4.3a,b and the results are compared with the exact analytical solution and previous BEM results in Figure 7.4.3c. It can be seen from Figure 7.4.3c that the results from the present analysis agree very well with the analytical and finite element results. At this point, it is important to mention that in the FEM analysis 1000 axisymmetric elements were used to obtain the correct results, whereas in the present BEM analysis the problem is modeled in three-dimensions with only 16 surface elements.

---

Fig. 7.4.3a Discretizations of hemispherical foundation-soil interfaces
Fig. 7.4.3b Discretizations of the free field for a hemispherical foundation
Fig. 7.4.3c Comparison of torsional stiffness coefficients for a hemispherical foundation
Dynamic Analysis

Application to the vibration isolation problem [41]:

In this example, results using BEST3D for a number of vibration isolation cases are compared with the actual field observation of Woods (62). He carried out a series of field tests in his attempt to define the screened zone and degree of amplitude reduction within the screened zone for trenches of a few specific shapes and sizes. He classified the foundation isolation problem into two categories, namely active isolation and passive isolation. Active isolation is the employment of barriers close to or surrounding the source of vibrations to reduce the amount of wave energy radiated away from the source. Passive isolation is the employment of barriers at points remote from the source of vibrations but near a site where the amplitude of vibration must be reduced. These tests were performed at a selected site of a two-layer system as shown on Figure 7.4.4a. A layer of uniform silty fine sand (SM) with dry density $\rho = 104 \text{ lb/ft}^3$, water content $w = 7\%$, void ratio $e = 0.61$, pressure wave velocity $v_p = 940 \text{ ft/sec}$, and the Rayleigh wave velocity $v_R = 413 \text{ ft/sec}$ rests on a layer of sandy silt (ML) with dry density $\rho = 91 \text{ lb/ft}^3$, water content $w = 23\%$, void ratio $e = 0.68$, and pressure wave velocity $v_p = 1750 \text{ ft/sec}$. The field tests for the active isolation were carried out for four lengths of the trench forming an arc of 90, 180, 270, and 360 degrees. The depth of the trench was varied between 0.5, 1.0, and 2.0 feet, while the width was 0.25 feet for all trenches. The distance from the center of footing to the centerline of trench is 1.0 foot. The field tests for the passive isolation were carried out for six lengths of trench, namely 1.0, 2.0, 3.0, 4.0, 6.0, and 8.0 feet. Distances from the center of footing to the centerline of trench were 5 and 10 feet and depths of trench were 1.0, 2.0, 3.0, and 4.0 feet. A constant input excitation force vector of 18 lb was used in all active and passive tests. The applied operational frequencies of the machine were 200, 250, 300, and 350 Hertz.

In the present investigation, the soil profile is modeled as a two-layer system defined by the shear modulus $G$, Poisson's ratio $\nu$, and mass density $\rho$. The material properties of the top layer are basically determined by the relations among the pressure wave velocity, the shear wave velocity, and the Rayleigh wave velocity. Accordingly, the values are taken as shear modulus $G = 647200 \text{ lb/ft}^2$, Poisson's ratio $\nu = 0.35$, and the mass density $\rho = 3.25 \text{ lb-sec/ft}^3$. The material properties of the bottom layer are determined by assuming the Poisson's ratio to be the same as that of the top layer. Accordingly, the values are taken as shear modulus $G = 1991150 \text{ lb/ft}^2$, Poisson's ratio $\nu = 0.35$, and the mass density $\rho = 2.84 \text{ lb-sec/ft}^3$. The diameter of the vibration excitor which was not specified, is assumed to be 0.50 feet. It was thought that this dimension has relatively less influence on the results. A distributed pressure of 81.5 lb/ft which is equivalent to 18 lb force is applied to the footing.

For the active isolation problem, a typical comparison of the BEM solutions and the experimental results are shown in Figures 7.4.4b and 7.4.4c. Figure 7.4.4b shows the overview of a contour diagram for a half circle trench with depth of 1.0 foot under an operating frequency of 250 Hertz. Both approaches predict similar screened zones which are area symmetrical about a radius from the source of excitation through the center of the trench and bounded laterally by two radial lines extending from the center of the source.
of excitation through points 45 degrees from each end of the trench. The expanding of screened zones may be noted when the screened zones in Figure 7.4.4c, which is obtained by extending the length of trench from half circle to 270 degrees, are compared with those of Figure 7.4.4b.

For the passive isolation problem, comparisons of the BEM solutions and the experimental results are shown in Figure 7.4.4d which shows the contour diagram of amplitude reduction factor for a rectangular trench of six feet in length and 2 feet in depth under an operating frequency of 250 Hertz. This trench is located 5 feet from the center of source. The comparison shows that again the experimental results are predicted qualitatively by the BEM solution. It should be noted that similar isolation analyses could also be carried out by BEST3D in the areas of transient dynamics as well as those occurring in the areas of periodic and transient acoustics.

Fig. 7.4.4a Field site soil properties and schematic of instrumentation
Fig. 7.4.4b Amplitude reduction factor contour diagram
Dynamic Analysis

Fig. 7.4.4c Amplitude reduction factor contour diagram

Fig. 7.4.4d Amplitude reduction factor contour diagram
Non-linear Stress Analysis

Benchmark Notch Specimen [34,36]:

The benchmark notch specimen is a double edge notch specimen developed by General Electric/Louisiana State University (GE/LSU) under NASA-Lewis Contract NAS3-22522. These data have been used to verify the elastic and inelastic capabilities of the present analysis.

Stress analysis was carried out for the gage section only, a procedure already known to be satisfactory. For the elastic analysis three different models were used, all idealizing one-quarter of the specimen gage section. Detailed comparison of results was carried out among the present analyses and finite element results. While these comparisons are not discussed here, it should be noted that, with sufficient mesh refinement, equivalent results were obtained with all analysis tools.

The discussion in this paper is directed at the comparison of present results with the GE/LSU strain gage data. The major characteristics of all of the analyses are summarized in the table below. The maximum peripheral strain in the notch (at the free surface) is given for each analysis. All analysis methods and the test data show that this value should be between 1700 and 1800 (microstrain).

<table>
<thead>
<tr>
<th>Model</th>
<th>Elements</th>
<th>Variation</th>
<th>Equations</th>
<th>Subregions</th>
<th>CPU Time (Seconds)</th>
<th>Max Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>50</td>
<td>linear</td>
<td>156</td>
<td>1</td>
<td>64</td>
<td>1688</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>quadratic</td>
<td>456</td>
<td>1</td>
<td>234</td>
<td>1780</td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>linear</td>
<td>78</td>
<td>2</td>
<td>20</td>
<td>1594</td>
</tr>
<tr>
<td>4</td>
<td>22</td>
<td>quadratic</td>
<td>210</td>
<td>2</td>
<td>60</td>
<td>1742</td>
</tr>
<tr>
<td>5</td>
<td>22</td>
<td>mixed</td>
<td>117</td>
<td>2</td>
<td>31</td>
<td>1729</td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>linear</td>
<td>36</td>
<td>1</td>
<td>10</td>
<td>1186</td>
</tr>
<tr>
<td>7</td>
<td>10</td>
<td>quadratic</td>
<td>96</td>
<td>1</td>
<td>28</td>
<td>1605</td>
</tr>
</tbody>
</table>

It is clear from the table that models 2, 4, and 5 all yield results of accuracy entirely comparable with the strain gage data. The variation in peak strain among these three is within ±1.5%. It is also clear that the most cost-effective analysis is that which combined substructuring with mixed linear and quadratic variation.
The first nonlinear analysis carried out for the specimen was a monotonic loading from zero load to the maximum load used in the test program. For this analysis quadratic variation was used for all surface elements. It was observed that the results gave generally good agreement with the experimental data, but that the overall strain level was somewhat too low. The analysis was repeated using a weighted surface model (Figure 7.5.1a) to capture better the gradients near the notch.

EQUALLY SPACED MESH
WEIGHTED MESH

Fig. 7.5.1a Regular and Weighted Boundary Element Surface Model for Benchmark Notch
The weighted model was then used for the cyclic analysis of the specimen. The results of this analysis (Figure 7.5.1b) show excellent agreement with the measured notch root strains.
Three-dimensional Analysis of a Notch Plate [43,44,45]:

The plastic deformation of a notch plate subjected to tension is analyzed under plane stress conditions using both the iterative and variable stiffness algorithms. The material properties for the plate are:

\[ E = 7000 \text{ kg/mm}^2 \]
\[ \nu = 0.2 \]
\[ \sigma_0 = 24.3 \text{ kg/mm}^2 \text{ (Von Mises yield criterion)} \]
\[ h = 0.0 \]

A 90 degree notch is cut out of the sides of the plate. The maximum to minimum width ratio is 2 and the thickness is 3/30 of the maximum width. A quarter of the plate is discretized in two subregions as shown in figure 7.5.2a. The region, containing the notch, has 30 quadratic boundary elements. The inset in this figure shows 6 (twenty-node) isoparametric cells (with 68 distinct cell nodes) which are used in volume integration. The second region has 16 quadratic boundary elements. No volume discretization is required in this region since it remains elastic throughout the analysis. Boundary conditions on the front and back faces are assumed traction free.

In figure 7.5.2b, the stress-strain response on the mid-plane of the root is given for the two (3-D) BEM analyses and compared with the two-dimensional plane stress and plane strain BEM solutions obtained by Banerjee and Raveendra [25]. The three-dimensional results are in good agreement with one another, and fall between the two-dimensional solutions, closer to the plane stress result, as one would expect. Also shown here are the results of particular integral analysis.

![Fig. 7.5.2a Boundary and volume discretization of a three-dimensional notch plate](image-url)
Fig. 7.5.2b Stress-strain response (on the mid-plane) at the root of a three-dimensional notch plate
Three-dimensional Analysis of a Perforated plate[43,44,45]:

The plastic deformation of a perforated plate in tension is analyzed under plane stress conditions using both the iterative and the variable stiffness solution algorithms. The material properties for the plate are:

\[
E = 7000 \text{ kg/mm}^2 \\
\nu = 0.2 \\
\sigma_0 = 24.3 \text{ kg/mm}^2 \text{ (Von Mises yield criterion)} \\
h = 224.0
\]

The diameter of the circular hole, at the center of the plate, is one-half the width, and the thickness is one-fifth the width. A quarter of the plate is discretized in 2 subregions, as shown in figure 7.5.3a. The first region, containing the root of the plate has 30 quadratic boundary elements and the second region has 23 boundary elements. No volume discretization is required in this region since it remains elastic throughout the analysis. Boundary conditions on both the front and back faces are assumed traction free.

This problem was previously analyzed experimentally by Theocaris and Marketos and by Zienkiewicz [16] using the finite element method. The stress-strain response at the root of the plate is shown in figure 7.5.3b. The results obtained using the various BEM algorithms show good agreement with one another and with the variable stiffness FEM analysis. Differences between the iterative and variable stiffness BEM formulations are much less significant than the differences in the 2 FEM algorithms. In order to evaluate the degree of convergence of the results, a mesh with 16 volume cells was studied. The results were unchanged from those shown. Those obtained from the particular integrals are also shown here for comparison.

The present analysis was carried out on the Cray-1 computer. The CPU times for the two algorithms were: 272 seconds for iterative procedure and 361 seconds for the variable stiffness method.

![Three-dimensional mesh of a perforated plate](image)
Fig. 7.5.3b Stress-strain response at the root of a three-dimensional perforated plate
8.0 REFERENCES


**Abstract**

This report documents the theoretical basis and programming strategy utilized in the construction of the computer program BEST3D (Boundary Element Solution Technology - Three Dimensional) and provides detailed input instructions for the use of the program. An extensive set of test cases and sample problems is included in the manual and is also available for distribution with the program. The BEST3D program was developed under the 3-D Inelastic Analysis Methods for Hot Section Components contract (NAS3-23697). The overall objective of this program was the development of new computer programs allowing more accurate and efficient three-dimensional thermal and stress analysis of hot section components, i.e., combustor liners, turbine blades, and turbine vanes. The BEST3D program allows both linear and nonlinear analysis of static and quasi-static elastic problems and transient dynamic analysis for elastic problems. Calculation of elastic natural frequencies and mode shapes is also provided.