GPBEST

General Purpose Boundary Element Software Technology

GPBEST USER'S MANUAL
Volume I

Elastic Analysis
Heat Transfer Analysis
Thermoelastic Analysis
Fracture Mechanics
Free Vibration Analysis
Acoustic Eigenfrequency Analysis
Periodic Acoustic Analysis
Forced Vibration Analysis
Consolidation Analysis
Plasticity Analysis

Version 6.0

October, 1999

Boundary Element Software Technology Corporation
P.O. Box 310
Getzville, NY 14068-0310, USA

PROPRIETARY DOCUMENT NOTIFICATION

Boundary Element Software Technology Corporation (called Best Corporation, hereafter) reserves the right to make changes in specifications and other information contained in this document without prior notice.

Although due care has been taken to present accurate information, Best Corporation DISCLAIMS ALL WARRANTIES WITH RESPECT TO THE CONTENTS OF THIS DOCUMENT (INCLUDING WITHOUT LIMITATION WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE) EITHER EXPRESSED OR IMPLIED. Best Corporation SHALL NOT BE LIABLE FOR DAMAGES RESULTING FROM ANY ERROR CONTAINED HEREIN, INCLUDING, BUT NOT LIMITED TO, FOR ANY SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF, OR IN CONNECTION WITH, THE USE OF THIS DOCUMENT.

GPBEST™ is a trademark of Best Corporation.
BESTVIEW™ is a trademark of Best Corporation.

This software product and its documentation set are copyrighted and all rights are reserved by Best Corporation. Usage of this product is only allowed under the terms set forth in the GPBEST License Agreement. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of Best Corporation is prohibited.

RESTRICTED RIGHTS NOTICE

- **If this software is furnished to a U.S. Department of Defense agency, the following applies**:

RESTRICTED RIGHTS LEGEND

''Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subdivision (b)(3)(ii) of the Rights in Technical Data and Computer Software clause at DOD FAR SUP 52.227-7013. Best Corporation, P.O. Box 310, Getzville, NY 14068-0310.''

- **If this software is furnished to the National Aeronautics and Space Administration, the following applies**:

RESTRICTED RIGHTS LEGEND

This computer software is submitted with ''restricted rights.'' Use, duplication or disclosure is subject to the restrictions as set forth in NASA FAR SUP 18-52.227-79 (April 1985) ''Commercial Computer Software-Restricted Rights (April 1985).'' Best Corporation, P.O. Box 310, Getzville, NY 14068-0310. If the contract contains the Clause at 18-52.227-74 ''Rights in Data General'' then the ''Alternate III'' clause applies.

Copyright ©1991, ©1993 and ©1995-1999 Boundary Element Software Technology Corporation
All rights reserved

If you would like to order more copies of this User Manual, please contact the Documentation Coordinator at Best Corporation. *U.S.A. orders:* All orders must be accompanied by a check or purchase order. Your order will be sent prepaid via UPS or fourth class mail and the shipping charges will be added to the invoice. F.O.B. will be the shipping point. Terms are net amount due within 30 days. *Outside U.S.A. orders:* Please contact Best Corporation for a quotation.

Part Number 970010

GPBEST User's Guide Comment Sheet (Release 6.0)

NAME : _____ DATE: _____

AFFILIATION : _____

ADDRESS : _____

TELEPHONE : _____

1. ERRORS/CORRECTIONS (Specify page, figure number, etc.)

_____

_____

_____

_____

_____

2. SUGGESTIONS/COMMENTS (e.g. overall manual organization and contents, future ideas, need for other manuals/primers, complaints, etc.)

_____

_____

_____

_____

_____

_____

_____

Please return to :

Dr. P. K. Banerjee
Boundary Element Software Technology Corporation
P.O. Box 310
Getzville, NY 14068-0310, USA

Telephone (716) 639-1920 Email support@gpbest.com

TABLE OF CONTENTS

SECTION **PAGE**

1.0 - Introduction and Installation 1.1
- 1.1 - Introduction 1.2
- 1.2 - GPBEST Installation Instructions 1.12
- 1.3 - Instructions for Use 1.16
- 1.4 - Contents of GPBEST System 1.17
- 1.5 - How to Request a GPBEST Password 1.21

2.0 - Theoretical Background 2.1
- 2.1 - Quasi-Static Elastic Analysis 2.2
- 2.2 - Analysis of Anisotropic Bodies 2.8
- 2.3 - Heat Conduction Analysis 2.11
- 2.4 - Thermoelastic Analysis 2.13
- 2.5 - Fracture Mechanics Analysis 2.16
- 2.6 - Poroelastic Analysis 2.19
- 2.7 - Periodic Acoustic Analysis 2.20
  - 2.7.1 - Pure Acoustics 2.20
  - 2.7.2 - Uncoupled Structural Acoustics - One-Way Coupling 2.21
  - 2.7.3 - Fully Coupled Structural Acoustics 2.23
- 2.8 - Periodic Dynamic Analysis 2.24
- 2.9 - Free-Vibration Analysis 2.26
- 2.10 - Acoustic Eigen-Frequency Analysis 2.28
- 2.11 - Point Source/Force Analysis 2.30
- 2.12 - Discontinuity Elements 2.32
- 2.13 - Inelastic Analysis 2.37

3.0 - Numerical Implementation 3.1
- 3.1 - Representation of Geometry and Functions 3.2
- 3.2 - Surface Integration 3.3
- 3.3 - Assembly of System Equations 3.5
- 3.4 - Solution of System Equations 3.8
  - 3.4.1 - Linear Analysis 3.8
  - 3.4.2 - Nonlinear Analysis 3.9

3.5 - Accuracy and Efficiency 3.17

4.0 - Guide to Using GPBEST 4.1
4.1 - Getting Started 4.2
4.2 - Graphics Interface 4.4
4.3 - Availability of GPBEST 4.7
4.4 - File System in GPBEST 4.8
4.5 - Special Features of GPBEST 4.12
4.5.1 - Definitions 4.12
4.5.2 - Mesh Size 4.14
4.5.3 - Specification of Boundary Restraints 4.15
4.5.4 - Problems of Edges and Corners 4.16
4.5.5 - Load Balance as a Measure of Accuracy 4.17
4.5.6 - Getting the Most from GPBEST 4.18
4.5.7 - Disk Space Requirements 4.25
4.6 - Useful UNIX-Based Operations 4.28

5.0 - GPBEST Input 5.0.1
5.0.1 - General Rules for Input Data 5.0.2
5.0.2 - General Recommendations for Input Data 5.0.6
5.0.3 - General Limits of GPBEST 5.0.7
5.0.4 - Individual Data Items 5.0.8
5.0.5 - GPBEST Keyword Input Card Quick Reference 5.0.9
5.0.6 - GPBEST Keyword Input Card Quick Reference Alphabetically Sorted 5.0.10
5.1 - Case Control 5.1.1
5.1.1 - Case Control Input Card 5.1.6
5.1.2 - Title 5.1.7
5.1.3 - Dimensionality of the Problem 5.1.8
5.1.4 - Type of Analysis 5.1.13
5.1.5 - Analysis Type Modifiers 5.1.31
5.1.6 - Times for Output 5.1.32
5.1.7 - Geometric and Loading Symmetry Control 5.1.39
5.1.8 - Restart Facility 5.1.46
5.1.9 - Efficiency and Precision Control 5.1.52
5.1.10 - Output Options 5.1.62
5.1.11 - Miscellaneous Control Options 5.1.78
5.1.12 - Nonlinear Algorithm Control Options 5.1.91

5.2 - Material Property Definition 5.2.1
5.2.1 - Material Property Input Card 5.2.7
5.2.2 - Material Identification 5.2.8
5.2.3 - Mass Parameters 5.2.9
5.2.4 - Isotropic Elastic Parameters 5.2.10
5.2.5 - Isotropic Thermal Parameters 5.2.14
5.2.6 - Temperature-Dependent Isotropic Thermoelastic Parameters . . . 5.2.16
5.2.7 - Anisotropic Elastic Parameters 5.2.20
5.2.8 - Additional Isotropic Poroelastic Parameters 5.2.34
5.2.9 - Additional Viscous Parameters 5.2.38
5.2.10 - Additional Elasto-Plastic Parameters 5.2.39
5.2.10.1 - Von Mises Model 5.2.40
5.2.10.2 - Concurrent Thermoplastic Model 5.2.45
5.2.10.3 - Modified Cam-Clay Model 5.2.48
5.2.10.4 - Isotropic Plasticity Soil Model 5.2.49
5.3 - Definition of Geometry 5.3.1
5.3.1 - Geometry Input Card 5.3.15
5.3.2 - Region Identification 5.3.17
5.3.3 - Nodal Point Definition 5.3.36
5.3.4 - Surface Element Definition 5.3.39
5.3.5 - Normal Definition 5.3.63
5.3.6 - Enclosing Element Definition 5.3.69
5.3.7 - Discontinuity Element Definition 5.3.73
5.3.8 - Line Element Definition 5.3.77
5.3.9 - Sampling Point and Sampling Surface Definition 5.3.95
5.3.10 - History Point Definition 5.3.101
5.3.11 - Volume Cell Definition 5.3.102
5.3.12 - Property Definition for FE GMR 5.3.110
5.4 - Definition of GMR Compatibility 5.4.1
5.4.1 - Interface Definition Input card 5.4.7
5.4.2 - Definition of Interface Surface 1 . . . . . 5.4.15
5.4.3 - Definition of Interface Surface 2 . . . . . 5.4.20
5.4.4 - Type of Interface Condition 5.4.29
5.4.5 - Additional Interface Input 5.4.47
5.4.6 - Cyclic Symmetry Interface Definition 5.4.57
5.5 - Definition of Boundary Conditions 5.5.1

5.5.1 - Boundary Condition Set Card 5.5.18
5.5.2 - Boundary Condition Identification 5.5.19
5.5.3 - Identification of Boundary Condition Type 5.5.21
5.5.4 - Definition of Surface for Application of Boundary Conditions 5.5.43
5.5.5 - Value Boundary Conditions for Surface Elements 5.5.56
5.5.6 - Definition of Hole/Cooling Line for Application of Boundary Conditions 5.5.78
5.5.7 - Value Boundary Conditions for Hole/Cooling Line Elements 5.5.83
5.5.8 - Definition Space/Time Variation 5.5.85
5.5.9 - Relation Boundary Condition 5.5.91
5.5.10 - Moving Boundary Condition 5.5.101
5.5.11 - Node Based Boundary Conditions 5.5.105
5.6 - Body Force Definition 5.6.1
5.6.1 - Body Force Input card 5.6.4
5.6.2 - Centrifugal Body Force 5.6.6
5.6.3 - Inertial Body Force 5.6.12
5.6.4 - Thermal Body Force 5.6.16
5.6.5 - Generalized Plane Stress/Plane Strain Input 5.6.25
5.6.6 - Point Source/Force Body Forces 5.6.31
5.6.7 - Discontinuity Surface Input 5.6.41
5.7 - Definition of Boundary Condition Changes 5.7.1
5.7.1 - Boundary Condition Change Card 5.7.2
5.8 - GPBEST-DCE Control 5.8.1
5.8.1 - GPBEST-DCE Input Card 5.8.3

6.0 - Tutorials and Examples 6.1
6.1 - GPBEST Features and Limits 6.2
6.1.1 - GPBEST.FEATURES File 6.2
6.1.2 - GPBEST Limits 6.7
6.2 - Data Flow In and Out of GPBEST 6.9
6.2.1 - Bestview 6.10
6.2.2 - PATRAN 6.13
6.2.3 - I-DEAS 6.16
6.2.4 - HYPERMESH 6.18
6.2.5 - FEMAP 6.20
6.2.6 - PRO/Engineer 6.22
6.2.7 - IGES 6.26

6.2.8 - Other Systems 6.27
6.3 - Example Problems 6.28
6.3.1 Acoustic Examples 6.31
6.3.2 Consolidation Examples 6.41
6.3.3 Concurrent-Thermoelasticity Examples 6.46
6.3.4 Elasticity Examples 6.63
6.3.5 Forced Vibration Examples 6.110
6.3.6 Fracture Mechanics Examples 6.120
6.3.7 Heat Transfer Examples 6.134
6.3.8 Elastoplasticity Examples 6.162
6.3.9 Concurrent-Thermoplasticity Examples 6.167

7.0 - Applications 7.1
7.1 - Elastic Analysis 7.2
7.2 - Heat Transfer Analysis 7.18
7.3 - Periodic Acoustic Analysis 7.42
7.4 - Periodic Dynamic Analysis 7.46
7.5 - Poroelastic Analysis 7.54
7.6 - Thermo-Elastic Analysis 7.59
7.7 - Electromagnetic Analysis 7.66
7.8 - Fracture Analysis 7.70
7.9 - Elasto-Plastic Analysis 7.76

8.0 - GPBEST Distributed Computing Environment (GPBEST-DCE) 8.1
8.1 - Installation Instructions 8.5
8.2 - Contents of GPBEST-DCE System 8.7
8.3 - GPBEST-DCE System Requirements 8.9
8.4 - Guide to Using GPBEST-DCE 8.12
8.5 - Instructions for Use 8.28
8.6 - General Limits of GPBEST-DCE 8.29
8.7 - GPBEST-DCE Startup Problems 8.30
8.8 - GPBEST-DCE Example Output 8.31

9.0 - References 9.1

1.0 INTRODUCTION AND INSTALLATION

GPBEST (General Purpose Boundary Element Solution Technology) 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 **GPBEST** 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, **GPBEST** 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 **GPBEST** is presented by analysis type in Table 1.2. Table 1.3 describes the new features that have been added in the current release of **GPBEST.**

The segments of the code dealing with quasi-static elastic, thermoelastic analyses, heat transfer and potential flow, periodic dynamic and acoustic analyses are relatively mature, since these benefit from a great deal of prior research effort. At the other extreme, the transient dynamic, eigenvalue and fluid dynamics and extensive nonlinear capabilities of **GPBEST** are still undergoing intensive evaluation and refinement. Hence, some of these advanced segments are not included in this manual.

1.1 INTRODUCTION

The primary objective of this User's Manual is to provide an overview of some **GPBEST** 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. (A Theoretical manual and Application-oriented Handbooks are planned to fill these voids.) However, sufficient material is included so that, after receiving this manual an engineering analyst should be able to prepare **GPBEST** input data sets.

In the next chapter, a brief review of the theoretical background is presented for each analysis category and references are cited to assist the interested reader in finding additional information. Then, Chapter 3 discusses the key aspects of the numerical implementation, while Chapter 4 provides a tutorial for the beginning **GPBEST** 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 some information that new users should be aware of, as well as 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 **GPBEST.** This chapter is primarily descriptive in nature and is intended merely to illustrate the level of engineering analysis that is possible within the present **GPBEST** system. Chapter 8 describes the **GPBEST-DCE** distributed computing environment implementation. format.

For the sake of overall computational efficiency the present **GPBEST 6.0** system has been divided into three executable segments:

'best-s24':

This includes all quasi-static analyses which involve the use of real arithmatic. Examples of analyses covered in this segment are elastic stress analysis, fracture analysis, gap and contact analysis, heat transfer analysis (both steady state and transient), thermal distortion analysis and concurrent thermoelastic analysis (both steady state and transient), free-vibration analysis and acoustic eigenfrequency analysis. Currently, this version is limited to 24,000 degrees of freedom (d.o.f.).

'best-s80':

This is similar to **best-s24**, except the maximum problem size is increased to 80,000 degrees of freedom. Use this version when the problem exceeds 24,000 d.o.f.

‘best-t24’:

This includes all analyses. In particular, problems of forced vibration analysis, periodic acoustic analysis, and plasticity analysis require this version. Currently, this version is limited to 24,000 degrees of freedom. Only use this version if **best-s24** will not run for the given problem.

‘best-t80’:

This is similar to **best-t24** , except the problem size is increased to 80,000 degrees of freedom. Use this version when the problem exceeds 24,000 d.o.f.

GPBEST User’s Manual Volume II discusses the BESTVIEW graphics system and the pre- and post-processing interfaces between **GPBEST**and graphics programs such as PATRAN, I-DEAS, FEMAP and HYPERMESH. Thus, a user may create a model using any finite element based graphics system by using linear (or parabolic) thin shell elements to model 3-D boundary elements and using linear (or parabolic) beam elements to model 2-D boundary elements. Next, by making minor modifications to the translator provided for pre-processing, these models may be converted to **GPBEST** data sets. Similarly, for post-processing of results obtained from **GPBEST** on a post-processing platform familiar to the user, the interface provided may be suitably modified.

TABLE 1.1
GENERAL FEATURES OF GPBEST

- Axisymmetric, two and three-dimensional problems
- Conforming element approach to provide inter-element continuity of the field variables, along with efficient solutions
- Substructured Regions (Super-elements) to permit multiple materials and more efficient solutions
- Accurate analysis of thin structures
- Hole elements with minimum discretization
- Cyclic and planar symmetry
- Different thicknesses allowed in multi-region planar problems
- Local or global boundary condition specification
- Sliding, spring, gap, interference fit, frictional and resistance-type interfaces
- Exterior domains
- Restart capability for low cost re-analysis
- Free-format, keyword-driven input
- Automatic error checks of input data
- Automatic check of equilibrium and heat balance
- BESTVIEW, PATRAN^{TM}, IDEAS^{TM}, FEMAP^{TM} and HYPERMESH^{TM} interfaces for pre- and post-processing
- Visualization of results of axisymmetric bodies under non-axisymmetric loading is as a complete three-dimensional object
- Visualization of the interior results for two-dimensional bodies is accomplished with only a boundary mesh
- Documented **GPBEST** neutral file format
- Sampling surfaces to view the interior results of 3-D bodies
- Cartesian and Cylindrical coordinate input

TABLE 1.2
ENGINEERING ANALYSIS BY GPBEST

A. Elastic Analysis

- 2-D, 3-D, axisymmetric and generalised axisymmetric problems for isotropic media
- Anisotropic media for two and three dimensional problems
- Centrifugal and self-weight body forces
- Spring loaded or sliding interfaces
- Gaps, contacts and interference fits
- Problems involving $180°$ symmetric modes of loading for axisymmetric bodies under non-axisymmetric loading
- Mixed regions involving plane stress and axisymmetric bodies
- Parallel processing using DCE (Distributed Computing Environment)
- Hole elements
- Point forces

B. Thermal Stress Analysis

- Two, three-dimensional and axisymmetric problems
- Steady state linear thermal stress analysis
- Transient linear thermal stress analysis
- Concurrent heat transfer and stress analysis for isotropic and cross-anisotropic bodies
- Cross-anisotropic materials for three-dimensional thermal stress analysis
- Gaps, contacts and interference fits
- Cooling holes
- Point forces and heat sources

TABLE 1.2 (continued)
ENGINEERING ANALYSIS BY GPBEST

C. Heat Transfer Analysis

- Two, three-dimensional and axisymmetric problems involving isotropic as well as anisotropic media
- Steady state heat transfer
- Transient heat transfer with variable time steps
- Thermal resistance across subregions
- Higher order time stepping for two-dimensional, three-dimensional and axisymmetric regions
- Other potential flow or diffusive systems, such as corrosive electrolytic processes, electrostatics, and ground water flow
- Hole elements
- Runner and cooling line elements
- Point sources of heat

D. Geotechnical Analysis

- Two, three-dimensional and axisymmetric problems
- Inhomogenous media (piece-wise homogeneous regions)
- Body forces due to self-weight and seepage forces
- Ground water flow analysis
- Coupled stress analysis in consolidation
- Anisotropic, as well as isotropic medium
- Discontinuity elements

E. Fracture Mechanics Analysis

- Elastic analysis with body forces
- Thermal fracture analysis

TABLE 1.2 (continued)
ENGINEERING ANALYSIS BY GPBEST

F. Dynamic Analysis

- Two, three-dimensional and axisymmetric problems
- Forced response
- Free vibration
- Piecewise homogeneous regions
- Point sources within subregions

G. Acoustic Analysis

- Two, three-dimensional and axisymmetric problems
- Acoustic eigenfrequency
- Periodic (steady-state) analysis
- Finite element shell elements for one-way and fully-coupled structural acoustics

H. Elastoplasticity, Thermoplasticity, and Poroplasticity

- Two-dimensional, three-dimensional and axisymmetric bodies
- Centrifugal and self-weight body forces
- Spring loaded or sliding interfaces
- Gaps, contacts and interference fits

TABLE 1.3
WHAT IS NEW IN RELEASE 6.0

The following user requested bug fixes and enhancements were done in this revision.

A. General Enhancements

1. Input of data in a cylindrical coordinate system is now available. The coordinate system is the basic cylindrical system that is defined with the origin at the origin of the global system, the Z-axis corresponding to the global Z-axis, and θ=0 along the positive global X-axis and measured counterclockwise. (The default is the global Cartesian system.) This cylindrical system is assumed for all input, including **GMR, **INTERFACE, **BCSET, and **BODY force input. See the COORdinate-system card under **CASE input for details.
2. The ability to define axisymmetric data in a Z-R coordinate system. For example, the coordinate system that is used by **GPBEST** for axisymmetric systems (R corresponding to X and Z corresponding to Y) is convenient for pressure vessels but very inconvenient for rotating structures (where Z corresponds to X and R corresponds to Y). See the COORdinate-system card under **CASE input for details.
3. Input of boundary conditions in any arbitrary local coordinate system that is unique to the current **BCSET. There are two new options here: 1) the tangential directions in a LOCAl NORMAL system can now be defined, and 2) a completely arbitrary local coordinate system can be defined. See the LOCAl card under **BCSET for details.
4. A uniform mechanical and thermal LOAD-bc can now be applied to elements instead of traction and flux. See the LOAD-bc card under **BCSET for details.
5. A TORQue-bc can now be directly applied to elements. See the TORQue-bc card under **BCSET for details. Since boundary conditions can now be defined with an arbitrary local coordinate system (item 3), torque can also be applied with tangential tractions. See the LOCAl card under **BCSET for details.
6. Mechanical (traction, load and torque) and thermal loading (flux and heat load) can now be applied on interface elements. See the INTErface card under **BCSET for details.
7. Prestress springs are now available. For both springs on the boundary and springs on an interface an initial spring deflection can be specified. See the SPRIng-value cards under **INTERFACE and under **BCSET for details.
8. INCLude-file card now permits inclusion of external files in the **GPBEST** data file. See the INCLude-file card under **CASE input for details.

B. Acoustics

1. Successful removal of the fictitious eigenfrequencies from the exterior acoustics.
2. Implemented the ‘‘QUICk-solver’’ for acoustic analysis.

3. Implementation of proper definitions for pressure, velocity, intensity, energy density, applied power, output power, transmittal power, and radiation efficiency.
4. The POSTBEST translator has been updated to support the translation of all nodal quantities output by GPBEST and all quantities are output in result file.
5. A history file for XY-plots has been added. It records a history of selected nodal quantities at selected nodes during a frequency sweep. (This replaces the SWEEP file.)
6. The ‘‘RADIATION.OUT’’ file has been correctly implemented.
7. The user can now select which frequencies will be output to the ‘‘NEUTRAL.OUT’’ file in a frequency sweep analysis.
8. Linear BUBBLE elements (5-noded) are correctly implemented.
9. The BCFIle has been reimplemented with better input structure and better accuracy, this option was formerly known as the INPUt NDFILE card.
10. Frequency dependent boundary conditions are now available.
11. Exterior periodic acoustic analysis region degree of freedom increased from 6000 to 8000 total for best-t80.
12. Axisymmetric Acoustic Analysis capability was enhanced to the level of the three dimensional analysis.
13. Several enhancements, bug fixes and documentation were completed for acoustic analysis. Please refer to the following cards in **CASE input for details:
14. ACOUstic - documentation of auxiliary files for acoustic analysis
15. SACOustic - documentation of auxiliary files for structural-acoustic analysis
16. HISTory-quantities - documentation of Sweep file for acoustic analysis
17. The manual has been updated to reflect these changes.

C. GPBEST Translators

1. Chapter 9 of the BestView User’s Manual has been completely rewritten to bring the documentation up-to-date and to better inform the user about modeling requirements when using the **GPBEST** translators.
2. Property IDs can now be assigned to elements. Thus, GMRs can be partitioned into surfaces, and linear functional variation can be used in the same GMR along with quadratic functional variation. Now efficient modeling practices can be utilized to run large problems in **GPBEST**. Some problems that were previously too large to run in **GPBEST** now run quite easily.
3. Material IDs can now be assigned to elements. Thus, the user can check the materials in their modeling programs.
4. An ‘‘All’’ option has been added to PREBEST to skip repetitive questions.

5. If the user assigns property IDs and Material IDs to elements, the number of questions that must be answered during a PREBEST session is minimal.
6. Capabilities for acoustics analysis have been greatly improved. The translators are now integrated better with **GPBEST**. All acoustic nodal results are now available for post-processing.
7. The user now has much more control over the translation of results in POSTBEST. Instead of processing all time-steps (or frequencies), a range of time-steps may be chosen. Only result quantities that are relevant for the current analysis are output. Also, the user can now choose to process only selected results. These changes can save significant amounts of disk space, as well as free the user of having to manage unneeded results.
8. The use of ‘‘contruction groups’’ is now supported by PREBEST. This allows the use of temporary groups in the modeling programs without having to delete them before running PREBEST.
9. Sampling surfaces in 2-D are now supported by PREBEST and POSTBEST.
10. Direct support for HYPERMESH, FEMAP, along with PATRAN and I-DEAS is now available.
11. Efficiency improvements have been achieved in both prebest and postbest leading to faster execution times of the translators.
12. Now there is only one prebest executable (80,000 dof) for all versions of **GPBEST** .
13. Element based data can now be output from the postbest translator for an extremely accurate display of contact pressure, normal force and frictional force.
14. Sampling surfaces and sampling points are now fully supported in prebest and postbest.
15. The accuracy of the interface detection algorithm in prebest has been improved. This includes interfaces in cyclic symmetry problems.
16. GEOMetry LINEAR capability has been included in prebest.
17. Anisotropic material properties are now supported for all preprocessor formats (i.e., PATRAN, I-DEAS, FEMAP and HYPERMESH) in prebest.
18. The postbest translator now warns the user if warning messages or fatal messages were encountered during the **GPBEST** run.
19. Various bug fixes have been made.

D. Line Elements

1. Variable TIMEs STEP input for transient analysis is now available. This allows the user to use small time steps at the beginning of a transient run and then increase the time steps for efficiency as the diffusive process slows down. See the TIMEs STEP card under **CASE input.

2. Three types of line elements are now supported for analysis of injection molding. Refer to the following **GMR cards:

- LINE - general category of line elements was introduced which includes HOLE, RUNNer, and CLINe elements
- HOLE - hole elements, used to model tubular exclustions in a material
- RUNNer - runner elements are used to model tubular inclusions in a material
- CLINe - cooling line elements are used to model fluid flowing through a tubular exclusion in a material

3. Display of Line Element results is now available. See the CONTour-plot card in **CASE input.

E. Other Improvements

1. The nodal coordinates can now be output in the neutral file with eight decimal place precision. This is critical, since a number of models are created in which one coordinate is 2 orders of magnitude larger than the other two. As a result, accuracy may be lost when calculating the displacement gradients for an element. This problem is now resolved.
2. In the material property section of the neutral file, there was no value for density nor the material angle. In the case of heat transfer, there were no properties at all, especially conductivity and specific heat. All of these have now been corrected.
3. In the interface section of the neutral file there were some missing data. There was nothing in the neutral file to indicate that an interface is a FIT-value or CONTact type of interface. Also, some of the interface modifiers, such as fit values or friction coefficient were not output. Finally, for a cyclic symmetry interface, the angle and direction vectors were incorrect (they were zero). All of these bugs have been corrected.
4. In the current version of the code, boundary conditions that are applied to nodes that are not source points cause a fatal error. Thus, it was quite painful to change a model from quadratic to linear variation if any of the boundary conditions were nodal based. Boundary conditions at non-source points are now simply ignored (a warning message is given).
5. Several other minor changes to **GPBEST** Neutral file format have been made for acoustic analysis. See relevent pages in Chapter 10 of the GPBEST User's Manual, Volume II.
6. A third card for the main header (Section 10.1.1) has been added which outputs informational, warning and fatal error messages.

1.2 GPBEST INSTALLATION INSTRUCTIONS

The **GPBEST** Release Guide shipped with the current **GPBEST** release is the first source for a complete and detailed installation procedure for all machine types. Please read these instructions carefully and follow all steps of the installation. It ishighly recommended that the system administrator perform this installation.

GPBEST installation is a two-step process.

Step 1: Follow instructions in the **GPBEST** Release Guide.

Step 2: The password file contains encrypted license information which is required by **GPBEST** and BESTVIEW in order to execute. It contains a set of text lines which can be entered using a standard text editor (vi). The password file is independent of **GPBEST** software and is provided separately. If your system identification was available at the time **GPBEST** software was sent to you, the password information will be included with this distribution. Otherwise please fill out the password request form included with this distribution and either fax or mail it to us.

Fax: (716) 639-1919

Mail: Boundary Element Software Technology Corporation
P.O. Box 310
Getzville, NY 14068-0310
USA

Kernel Configuration for Hewlett Packard (HP-UX) Computers

For good performance in execution of **GPBEST** , it is recommended that certain configurable parameters in the HP-UX kernel be set at or above the values described below. The two parameters of concern are:

1. **maxdsiz** - the maximum data segment size (in bytes)
2. **maxssiz** - the maximum stack segment size (in bytes)

maxdsiz is typically set at 90% of the SWAP size, and **maxssiz** is typically set at 25% of the memory size. The minimum values recommended for execution of **GPBEST** are:

1. **maxdsiz** = $268,435,456$ (256 Mb)
2. **maxssiz** = $33,554,432$ (32 Mb)

If your kernel parameters are set higher than these values, we do NOT recommend that you lower your values since they were probably increased for some other application.

The parameters can be reset using the ‘‘System Administration Manager’’ (SAM). They are located under the heading ‘‘kernel configuration’’ and under the subheading ‘‘configurable parameters’’. Creation of a new kernel is required.

In order to reset these parameters you must be the superuser, and the system must be rebooted, so no other user should be accessing the system during this time.

It is recommended that a System Administrator knowledgeable in the HP-UX operating system perform this action and that the current parameter settings are noted for future reference. It is also recommended that before doing any system administration work, a complete backup of the system is performed.

BEWARE: If incorrect values are used to produce a new kernel, it could result in poor performance of the system or the new kernel could be incapable of booting the system altogether.

The *Boundary Element Software Technology Corporation* does **NOT** assume any responsibility in connection with the above recommendations.

Password Installation Instructions for All Computers

I. Install the GPBEST/BESTVIEW password file.

A password file is necessary in order to execute **GPBEST** and BESTVIEW. The password file contains a password for your specific installation and computer system identification. Instructions for obtaining the password file are given in Section 1.5 of this manual.

The password file is usually e-mailed to the user, so the e-mail headings and miscellaneous information must be stripped off. The system administrator can merely edit the file until it contains only the encrypted password information. Assuming the necessary password file has been obtained and edited, the system administrator can place it in one of two locations on the system. Either method can be used.

`Method 1.)` The password file can be placed in the directory /gpbest, which resides on the same disk as the **GPBEST** installation directory. The following steps show how to

create the /gpbest directory, insert the password file and adjust permissions so that all users can access the password file:

```
“ cd /  ”
“ mkdir gpbest ”
“ chmod ugo+rx gpbest ”
“ cd gpbest  ”
“ cp /etc/mail/password /gpbest/password  ”
“ chmod ugo+r password ”
```

Method 2.)As an alternative to the above procedure, the password can be placed in any directory, but that directory and file must be specified in the user’s startup file in an environment variable. The following steps place the password file in the directory /usr/local/gpbest:

```
“ cp /etc/mail/password /usr/local/gpbest/password  ”
“ chmod ugo+r /usr/local/gpbest/password ”
```

This directory and file name must be specified in each user’s startup file. For instance, if the user uses the c-shell, then insert the following line in the .cshrc file:

```
“ setenv GPBEST_PASSWORD_FILE /usr/local/gpbest/password ”
```

If the user uses the korn shell or borne shell, then insert the following lines in the .kshrc file or .profile file, respectively:

```
“ GPBEST_PASSWORD_FILE=/usr/local/gpbest/password ”
“ export GPBEST_PASSWORD_FILE  ”
```

II. Set the BVIEWDIR environment variable.

This step applies to BESTVIEW only. BESTVIEW users are required to set an environment variable in their startup file which permits BESTVIEW to find some additional files that it needs to access. For instance, if the user uses the c-shell on a HP workstation, then insert the following line in their .cshrc file:

```
“ setenv BVIEWDIR /usr/local/gpbest/bin/HPPA10 ”
```

If the user uses the korn shell or borne shell, then insert the following lines in the .kshrc file or .profile file, respectively:

```
“ BVIEWDIR=/usr/local/gpbest/bin/HPPA10 ”
“ export BVIEWDIR  ”
```

III. Specify fonts for BESTVIEW.

This step applies to BESTVIEW only. If the user receives a message complaining about improper fonts when running BESTVIEW, then the default BESTVIEW fonts must be changed. In order to determine the available fonts on the system, run the 'xlsfonts' utility while in X-Windows. The 'xlsfonts' utility will list each font available on your system (one font per line). Two of these fonts are needed. For example, on an HP workstation:

```
" xlsfonts "
" (choose two fonts from the resulting listing) "
" cd /usr/local/gpbest/bin/HPPA10 "
" vi BestviewFonts.inc (edit this file) "
```

Lines 1 and 3 in the BestviewFonts.inc file should be replaced with the two fonts chosen from the 'xlsfonts' listing. For example, after carrying out the above steps the BestviewFonts.inc file could look like the following:

```
-adobe-helvetica-medium-r-normal--14-140-75-75-p-77-hp-roman10
font1
-adobe-courier-medium-r-normal--24-240-75-75-m-150-hp-roman8
font2
```

The user may have to experiment with some fonts until two fonts are found that give acceptable display in BESTVIEW.

GPBEST and BESTVIEW installation is now complete. If you encounter any problem, please send us a fax at (716) 639-1919 with a brief description of your problem.

1.3 INSTRUCTIONS FOR USE

GPBEST

To execute **best-s24** as a background process, type in this c-shell

```
(time < data best-s24 ) > & result &
```

Please Note:

In order to run **GPBEST** you must have write permission to the directory where you are executing **GPBEST** .

Also you must have read permission to /gpbest/password file. This file is read in each time **GPBEST** is executed and contains **GPBEST** license information.

For more information on password file, refer to installation instructions.

GPBEST-DCE

To execute **best-dce** as a background process, type in this c-shell

```
(time < data best-dce ) > & result &
```

Please Note:

In order to run **GPBEST-DCE** you must have write permission to the directory where you are executing **GPBEST-DCE** .

Also you must have read permission to /gpbest/password file. This file is read in each time **GPBEST-DCE** is executed and contains **GPBEST-DCE** license information.

For more information on password file, refer to installation instructions.

PREBEST

prebest is the PATRAN neutral file, I-DEAS universal file, or HYPERMESH ASCII neutral file to **GPBEST** data translator. This program is interactive and will prompt you for required files.

To execute **prebest** type in

```
prebest
```

POSTBEST

postbest is the **GPBEST** to PATRAN, I-DEAS or HYPERMESH translator. This program is interactive and will prompt you for required files.

To execute **postbest** type in

```
postbest
```

This will convert **GPBEST** neutral file information into PATRAN files *.GEOM, *.DIS and *.NOD, I-DEAS universal file, or HYPERMESH neutral and results file for postprocessing.

1.4 CONTENTS OF GPBEST SYSTEM

This distribution contains a number of subdirectories. The basic directory structure is:

```
(USER_DEFINED_DIRECTORY)/bin/HPPA10
(USER_DEFINED_DIRECTORY)/lib/HPPA10
(USER_DEFINED_DIRECTORY)/bin/RS6K
(USER_DEFINED_DIRECTORY)/lib/RS6K
(USER_DEFINED_DIRECTORY)/bin/SGI6
(USER_DEFINED_DIRECTORY)/lib/SGI6
(USER_DEFINED_DIRECTORY)/bin/SUN4SOL2
(USER_DEFINED_DIRECTORY)/lib/SUN4SOL2
(USER_DEFINED_DIRECTORY)/bin/WINNT
(USER_DEFINED_DIRECTORY)/lib/WINNT
(USER_DEFINED_DIRECTORY)/ADOBE
(USER_DEFINED_DIRECTORY)/manuals
(USER_DEFINED_DIRECTORY)/examples
(USER_DEFINED_DIRECTORY)/templates
```

These directories are located under the (USER_DEFINED_DIRECTORY) directory that was created during installation. If the (USER_DEFINED_DIRECTORY) directory was called /usr/local/gpbest. Hence, the above directories would be called /usr/local/gpbest/bin, etc. The installation of the **GPBEST** system has machine dependent and independent files, thus not all of the above listed directories will appear in a normal **GPBEST** installation.

It should also be noted that the `bin` directory contains a subdirectory for each machine that is supported and installed. A single **GPBEST** installation directory can be used for multiple machine dependent files and directories. The following is a list of the bin subdirectories and their corresponding machines:

```
HPPA10      - Hewlett-Packard (HPUX 10.XX or above)
RS6K        - IBM RS600 (AIX 4.XX or above)
SGI6        - Silicon Graphics (IRIX 6.XX)
SUN4SOL2    - SUN Microsystems (SOLARIS 2.6X or above)
WINNT       - Microsoft Corporation (WINNT 4.0 Service Pack 3)
```

To obtain a list of all directories and files in the **GPBEST** distribution:

1. cd to the **GPBEST** `bin` directory
2. `find . -print`

Basic Directory	Subdirectories & Files	Explanation
1. bin/		As explained above, this is actually `bin/HPPA10`, etc. depending on the user's specific machine.
	BestviewFonts.inc	(non executable) Fonts used in BESTVIEW.

This file lists the fonts used by the bestview software for displaying the menu and other messages. This file should be placed in the same directory as bestview.

COLORBAR (non-executable) Colorbar for BESTVIEW. BESTVIEW has a unique coloring scheme. If the user would like to use the traditional colors (red, orange, yellow, green, blue), then include the appropriate **COLORBAR** in your working directory and select it from within BESTVIEW (see the BESTVIEW manual for details).

bestview BESTVIEW executable. BESTVIEW (Boundary Element Software Technology Viewer) is a graphic pre- and post-processor for generation and viewing of structural models as well as for preparation of input data for subsequent analysis through GPBEST.

best-s24 gpbest (standard) executable for elastic, thermoelastic distortion, heat conduction, diffusion, consolidation, fracture, contact and gap, free vibration and acoustic eigenfrequency analysis. (24,000 d.o.f.)

best-s80 gpbest (standard) executable (80,000 d.o.f.). For smaller problems use **best-s24** for best efficiency.

best-t24 gpbest executable for ALL analysis types. Standard analyses contained in best-s24 are also contained in best-t24. However, it is recommended that best-s24 be used for standard analysis since best-s24 requires less computer resources to execute. best-t24 should be used for all other analyses not contained in best-s24. These are periodic acoustics, structural acoustics, forced vibration and plasticity analysis. (24,000 d.o.f.)

best-t80 gpbest (total) executable (80,000 d.o.f.). For smaller problems use **best-t24** for best efficiency.

iges iges executable. iges executable module is used by the bestview software to import IGES files.

prebest GPBEST translator. Translates PATRAN neutral files, I-DEAS universal files, HYPERMESH ASCII neutral files, and FEMAP ASCII neutral files to GPBEST input data files.

postbest GPBEST translator. Translates GPBEST neutral file `NEUTRAL.OUT` (created during GPBEST execution to a PATRAN neutral file or an I-DEAS universal file or a HYPERMESH ASCII neutral file or a FEMAP ASCII neutral file for

postprocessing of GPBEST results.

README Miscellaneous information about this directory. View this file for the most recent information.

2. examples/ Example problem directory. The example problem files are stored in `gzip` format. To uncompress an entire directory, `cd` to the directory and:

```
gunzip *.gz
```

EX.summary File contains a tabular description of the features covered in the **GPBEST** serial example problems.

DCE.summary File contains a tabular description of the features covered in the **GPBEST-DCE** parallel example problems.

The examples directory contains subdirectories of examples of GPBEST data (*.dat) and result (*.res) files. Several problems also include a GPBEST neutral file (*.neu).

ex.small/ Contains small examples that require very little computing time. Prefix q is used to designate the use of quartic elements and prefix s has been used to designate problems involving body forces or sources.

dce/ Contains small examples that require very little computing time. These example problems should be used as an example of **GPBEST-DCE** parallel configuration and will not run without modifications on the users local installation.

The names of the files in subdirectory `ex.small` and `dce` signify the type of problem as indicated below:

acou Acoustic analysis.

aeig Acoustic eigenfrequency analysis.

anis Elastic anisotropy (includes thermal and mechanical body forces).

cons Consolidation.

cpls Concurrent thermoplasticity (one-way coupling of heat conduction and thermoplasticity).

cthe Concurrent thermoelasticity (one-way coupling of heat conduction and thermoelasticity).

dist	Thermal distortion with holes (heat from cooling holes causes distortion).
elas	Elasticity (includes thermal and mechanical body forces).
forc	Periodic elastodynamic analyses.
frac	Fracture mechanics.
frvb	Free vibration analysis.
gaxis	Generalized axisymmetric elastic analysis under nonaxisymmetric loading.
heat	Steady-state and transient heat conduction (includes problems with holes).
plas	Elastoplasticity.
saco	Structural acoustic analysis.
ex.large/	Contains a series of data for very large problems of elastic stress analysis involving complex geometries.

3. lib/ — Contains files for machine dependent **GPBEST-DCE** system.

4. templates/ — Contains files for the PATRAN translator system.

5. manuals/ — Contains online manual Acrobat files for GPBEST User's Manual Volume I, GPBEST User's Manual Volume II, and Introduction to GPBEST: Elastic Analysis Manuals.

6. ADOBE/ — Contains online manual Acrobat executable files for the GPBEST online manuals.

7. README — Miscellaneous information about this directory. View this file for the most recent information on the **GPBEST** system.

1.5 HOW TO REQUEST A GPBEST PASSWORD

If the password information was not included with your **GPBEST** package, please make a copy of password request form, fill in the required information on the copy and either fax or mail it to us.

Fax: (716) 639-1919

Mail: Boundary Element Software Technology Corporation
P.O. Box 310
Getzville, NY 14068-0310
USA

It is important that you use large and clear letters, because it is difficult to read small letters on a fax.

Currently, distribution tapes are available on all supported platforms in the following tape formats:

CDROM
quarter inch tape QIC 150
4mm DAT

In order for us to create a password for you, we will need your system ID. You will need this information to fill out the password request form in the next page (Section 1.6). Please enter the appropriate unix command for your system

SUN Solaris 2.6X systems: /usr/bin/showrev
IBM RS6000 AIX 4.XX systems: uname -m
Silicon Graphics IRIX 6.XX systems: lmhostid
Hewlett Packard HPUX 10.XX systems: uname -i

NOTE: If any of these commands are not found, use the 'whereis' command (e.g., whereis hostname) to find the directory path of the command and add it to your own path. Then re-execute the command.

If you need more than one license for a specific computer type, please fill out another request form with only model, hostname and system identification information. Multiple computers can be incorporated into one password file.

This Page Intentionally Blank

GPBEST AND BESTVIEW PASSWORD REQUEST FORM

Fax your password request forms to (716) 639-1919

Date: _____ Fax: _____

Name: _____ Phone: _____

E-mail Address*: _____

Company/Organization: _____

Address: _____

_____

_____

New Order: _____ Renewal: _____ Trial: _____ Upgrade: _____ Info Change: _____

(Check one)

Computer Type: _____

(ex: SUN Sparc / IBM RS6000 / Silicon Graphics / HP9000-700 series)

Operating System Version: _____

Model: _____

Hostname: _____

(use hostname command)

System Identification: _____

(see instructions in Section 1.5)

Starting Date: _____ Ending Date: _____

System Administrator: _____ Phone: _____

* Passwords will be sent via E-mail if an E-mail address is provided.

2.0 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, elastodynamic, wave and potential flow equations have existed in the literature for, at least, 50 years. With the emergence of digital computers the method had begun to gain popularity as the 'boundary integral equation method', 'panel method', or 'integral equation method' during the sixties. The name was changed to the 'Boundary Element Method' by Banerjee and Butterfield [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 **GPBEST** are briefly outlined.

2.1 QUASI-STATIC ELASTIC ANALYSIS

It is well known that, for a homogeneous, elastic material, there exists point force solutions to the governing differential equations. 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.

Boundary Integral Equation

The governing differential equation can be written as,

$$L(u_i) + f_i = 0 \tag{2.1.1}$$

where

L stands for the differential operator
u_i the displacement vector
f_i the body force vector
$i = 1, 2$ for 2D and, $= 1, 2, 3$ for 3D from displacements one can obtain the tractions (t_i) via the strain-displacement relation and the constitutive law.

If the body force appearing in equation (2.1.1) is a point load given by

$$f_i^*(x) = \delta_{ij}\ \delta(x - \xi) e_j(\xi) \tag{2.1.2}$$

then the solution to equation (2.1.1) is known as the fundamental solution. In equation (2.1.2) e_j stands for the unit vector in the direction j.
The fundamental solution is written as,

$$u_i^*(x) = G_{ij}(x, \xi) e_j(\xi) \tag{2.1.3a}$$

also,

$$t_i^*(x) = F_{ij}(x, \xi) e_j(\xi) \tag{2.1.3b}$$

in the above expressions the tensors $G_{ij}(x, \xi)$ and $F_{ij}(x, \xi)$ represent respectively the displacement and traction at the field point x in the direction i due to a unit load applied at the source point ξ in the direction j. These terms are functions of the elastic properties of the continuum i.e. the elastic modulus E and Poisson's ratio ν. The asterisk ($*$) refers to the fundamental solution state.

The Reciprocal Work Theorem between the actual state and the fundamental solution state is,

$$\int_S t_i(x) u_i^*(x) ds + \int_V f_i(x) u_i^*(x) dv = \int_S t_i^*(x) u_i(x) ds + \int_V f_i^*(x) u_i(x) dv \tag{2.1.4}$$

where V and S are the volume and the surface of the body respectively.

the volume integral appearing on the right side of equation (2.1.4) can be written, in the light of equation (2.1.2), as:

$$\int_V u_i(x)\,\delta_{ij}\,\delta(x-\xi)\,dV = u_j(\xi) \tag{2.1.5}$$

Using equations (2.1.3a,b) and (2.1.5) in equation (2.1.4), one obtains, finally,

$$u_j(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_V G_{ij}(x,\xi)f_i(x)dv \tag{2.1.6}$$

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. It is the well-known Somigliana's Identity. 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 source point to the boundary of the body, obtaining:

$$c_{ij}(\xi)\,u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_V G_{ij}(x,\xi)f_i(x)dv \tag{2.1.7}$$

a constraint equation relating the known and unknown displacements and tractions on the surface of the body. where $c_{ij}(\xi)$ is known as the 'jump term' and has the following values:

i) δ_{ij} for ξ within volume, V
ii) $0.5\delta_{ij}$ for ξ on smooth boundary, S and
iii) 0 for ξ outside V.

Equation (2.1.7) is the Boundary Integral Equation and its numerical implementation is the essence of the Boundary Element Method.

In the simplest case, a homogeneous elastic problem without thermal or mechanical body forces, all the volume integrals vanish and (2.1.7) 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.

Interior Quantities

The displacement at designated interior points can be obtained from equation (2.1.6). Differentiation of equation (2.1.6) and use of the elastic stress-strain relations allow the evaluation of strains and stresses at arbitrary points within the body. Thus,

$$\epsilon_{ij}(\xi) = \int_S [G^{\epsilon}_{kij}(x,\xi)t_k - F^{\epsilon}_{kij}(x,\xi)u_k(x)]ds + \int_V G^{\epsilon}_{kij}(x,\xi)f_k(x)dv \tag{2.1.8a}$$

$$\sigma_{ij}(\xi) = \int_S [G^{\sigma}_{kij}(x,\xi)t_k - F^{\sigma}_{kij}(x,\xi)u_k(x)]ds + \int_V G^{\sigma}_{kij}(x,\xi)f_k(x)dv \tag{2.1.8b}$$

can be written for interior strains and stresses. The kernel functions G^{ϵ}_{ij}, F^{ϵ}_{ij}, G^{σ}_{ij} and F^{σ}_{ij} can be found in reference [2]. The above expressions can not be used to obtain strains and stresses at boundary points because of hypersingularity of the kernels. Thus an alternate procedure is used where these quantities are obtained from the boundary displacements, tractions and their local derivatives. The procedure for three-dimensional problems is explained below.

A local Cartesian coordinate system is constructed with the axes 1 and 2 directed along the tangential directions and the axis 3 in the direction of the outward normal. The stresses $\bar{\sigma}_{ij}$ referred to these local axes (indicated by overbars) are then given by:

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

where $\bar{\epsilon}_{ij}$ defines the components of the elastic strains in the local axes obtained from the derivative of the shape functions and nodal displacements and $\bar{t}_i$ are the traction on the boundary.

The above formulations are applicable to both two-dimensional (axisymmetric, plane strain, plane stress and generalized plane stress and strain) and three-dimensional problems.

Thermal Loading and Body forces by Particular Integrals

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 **GPBEST** . Rather, all of these effects on linear analyses have been incorporated using particular integrals where no surface integrations are required. This approach is explained herein.

The governing differential equation for a homogenous, isotropic body subjected to body force loading, as stated earlier, can be expressed in operator notation as [35,38,50,51,62]:

$$L(u_i) + f_i = 0 \tag{2.1.9}$$

where

$L(u_i) = (\lambda+\mu)u_{i,ji} + \mu u_{i,jj}$

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 $\beta = (3\lambda + 2\mu)\alpha$, and,

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

A solution satisfying a linear, inhomogenous, 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 [35,38,50,51,52]

$$L(u_i^c) = 0 \tag{2.1.10}$$

and a particular integral u_i^p satisfying the inhomogenous equation

$$L(u_i^p) = f_i \tag{2.1.11}$$

The total solution u_i is expressed as

$$u_i = u_i^c + u_i^p \tag{2.1.12}$$

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 ξ, is expressed as

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

$$i, j = 1, 2, 3 \qquad (2.1.13)$$

where the u_i^c and t_i^c are the complementary functions for displacement and traction, respectively. The total solution for displacement u_i and traction t_i are

$$u_i = u_i^c + u_i^p$$

$$t_i = t_i^c + t_i^p \tag{2.1.14}$$

where u_i^p and t_i^p 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 (2.1.9) is a particular integral, regardless of boundary conditions or how it was obtained.

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

$$Gt^c - Fu^c = 0 \tag{2.1.15}$$

By introducing equation (2.1.14) into equation (2.1.15), the complementary function can be eliminated;

$$Gt - Fu = Gt^p - Fu^p \tag{2.1.16}$$

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 (2.1.15), 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 (2.1.16).

Axisymmetric body under non-axisymmetric loading

For a general axisymmetric problem, i.e. an axisymmetric elastic body under non-axisymmetric loadings, the displacement and traction fields are expressed in terms of Fourier series as

$$u_r = \sum_{n=0}^{\infty}(u_r^{nc}\cos n\theta + u_r^{ns}\sin n\theta) \tag{2.1.17a}$$

$$u_z = \sum_{n=0}^{\infty}(u_z^{nc}\cos n\theta + u_z^{ns}\sin n\theta) \tag{2.1.17b}$$

$$u_\theta = \sum_{n=0}^{\infty}(u_\theta^{ns}\sin n\theta + u_\theta^{nc}\cos n\theta) \tag{2.1.17c}$$

$$t_r = \sum_{n=0}^{\infty}(t_r^{nc}\cos n\theta + t_r^{ns}\sin n\theta) \tag{2.1.17d}$$

$$t_z = \sum_{n=0}^{\infty}(t_z^{nc}\cos n\theta + t_z^{ns}\sin n\theta) \tag{2.1.17e}$$

$$t_\theta = \sum_{n=0}^{\infty}(t_\theta^{ns}\sin n\theta + t_\theta^{nc}\cos n\theta) \tag{2.1.17f}$$

In each circumferential order n, the first terms denote the symmetric displacement and traction fields and the second terms antisymmetric fields. In **GPBEST** the maximum circumferential order is 3 (n=0,1,2,3). Then the equation (2.1.7), in the absence of body forces leads to the decoupled boundary integral equations in cylindrical coordinates as

$$c_{ij}(\xi)\, u_i^n(\xi) = \int_L \left[G_{ij}^n(x,\xi)\; t_i^n(x) - F_{ij}^n(x,\xi)\; u_i^n(x) \right]\; r_x dL(x) \qquad (2.1.18a)$$

$$c_{ij}(\xi)\, u_i'^{\,n}(\xi) = \int_L \left[G_{ij}'^{\,n}(x,\xi)\; t_i'^{\,n}(x) - F_{ij}'^{\,n}(x,\xi)\; u_i'^{\,n}(x) \right]\; r_x dL(x) \qquad (2.1.18b)$$

Equation (2.1.18a) is for symmetric displacement and traction fields, while (2.1.18b) is for the antisymmetric field.

Note that only the discretization of the generator L of axisymmetric body is needed. For further information on general axisymmetric analysis by BEM, please refer to the paper by Wang and Banerjee [60] and Henry, Pape and Banerjee [38].

Multi-region Problems

Although the above sections describe the Boundary Element Formulation for a single region, bodies with multiply-connected regions can be analysed by formulating the Boundary Integral Equation for each region separately and then assembling together by satisfying the compatibility of displacements and equilibrium of tractions across the interface between two adjacent regions. Within each region material properties are homogeneous but they may vary in different regions.

As these regions are being assembled many complex interface relations involving sliding, spring loaded or bonded interfaces are introduced with relative ease.

2.2 ANALYSIS OF ANISOTROPIC BODIES

In many aerospace applications, the structures often have materials with directional dependence. The boundary element method posseses the unique ability to include this anisotropic behavior within the formulation through the use of anisotropic Green's functions. Within the BEM community, **GPBEST** has the unique distinction of being the only BEM code with the kernels to deal with directionally dependent material behavior.

In the most commonly applied uncoupled quasistatic thermoelastic theory, the spatial derivatives of temperature enter the Navier equations of elastostatics with the same effect of an applied body force field. In the context of the conventional direct boundary element formulation, this thermal body force amounts to an additional volume integral [14]. In an alternative approach, the boundary element formulation is stated in terms of complementary solutions of the Navier equations by choosing arbitrary particular integrals, and the above-stated volume integrals warranting volume (domain) discretization can be avoided [40]. In what follows, BEM is applied to the thermal stress analysis of anisotropic media of the most general form. Both the volume integration and the particular integral-based approaches are considered. In the latter formulation, the derivation of the particular integrals for the exact modeling of a quadratic temperature distribution is presented. It is of relevance to point out in this context that particular integrals need to be chosen judiciously for competent results.

Governing Equation

The relevant equations [102], in the absence of mechanical body forces, for three-dimensional, uncoupled, quasistatic thermoelasticity for an anisotropic medium in terms of net temperature $T(\mathbf{x})$ and displacement field $u_k(\mathbf{x})$ are:

$$\lambda_{ij} T_{,ij} + W = 0 \tag{2.2.1}$$

$$c_{ijkl} u_{k,lj} - \beta_{ij} T_{,j} = 0 \tag{2.2.2}$$

$$(i, j, k, l = 1, 2, 3)$$

where λ_{ij} are the thermal conductivities, W is a distributed heat source, c_{ijkl} are the stiffness coefficients and $\beta_{ij} = \alpha_{kl} c_{ijkl}$, α_{kl} being the coefficients of linear thermal expansion. For general anisotropy, there are twenty-one independent stiffness coefficients c_{ijkl} and six independent thermal coefficients α_{ij}. For a problem with well-defined boundary conditions, equations (2.2.1) and (2.2.2) bear a cause-and-effect relationship since the solution of equation (2.2.1) provides the necessary body force type input $-\beta_{ij} T_{,j}$ in the Navier equation (2.2.2). Our interest here lies in solving the thermal stress analysis problem represented by (2.2.2) by the direct boundary element method assuming that the task of solving the causal problem of heat conduction (2.2.1) has already been carried out. It is noted that for the case of two-dimensional heat conduction with plane strain elasticity, the highest degree of anisotropy permissible corresponds to a material with a plane of symmetry [100]. For such a material, there are thirteen independent elastic constants c_{ijkl} and four independent thermal coefficients α_{ij} [100], [103]. Thus for the case of plane strain,

the last of the equations (2.2.2) is identically satisfied and the thermal stress analysis problem will be reduced to solving two second order simultaneous differential equations corresponding to $i, j, k, l = 1$ and 2. The case of plane stress is obtained from the plane strain analysis by replacing the material stiffnesses c_{ijkl} and coefficients of thermal expansion α_{ij} by the corresponding reduced quantities obtained by incorporating the assumptions of generalized plane stress in the Duhamel-Neumann constitutive relations.

Boundary Element Formulation

By treating the quantity $-\beta_{ij}T_{,j}$ in equation (2.2.2) as a body force, the following direct boundary integral equation can be written using the notation of Banerjee and Butterfield [2]:

$$c_{ij}(\xi)u_i(\xi) = \int_S [G_{ij}(\mathbf{x},\xi)t_i(\mathbf{x}) - F_{ij}(\mathbf{x},\xi)u_i(\mathbf{x})]\, ds(\mathbf{x}) + \int_V B_{ikj}(\mathbf{x},\xi)\beta_{ik}T(\mathbf{x})dv(\mathbf{x}) \tag{2.2.3}$$

$$(i, j, k = 1, 2 \text{ in 2D}$$
$$= 1, 2, 3, \text{ in 3D})$$

where

$G_{ij}(\mathbf{x},\xi)$ are Green functions or displacement kernels;

$F_{ij}(\mathbf{x},\xi)$ are traction kernels derived using $G_{ij}(\mathbf{x},\xi)$;

$c_{ij}(\xi)$ = δ_{ij} (Kronecker delta) for interior points and is dependent on surface geometry at ξ for boundary points

$$B_{ikj} = \frac{\partial G_{ij}}{\partial x_k} \tag{2.2.3a}$$

When ξ is a point on the boundary, the integrals containing the kernels G_{ij} and B_{ikj} are weakly singular and exist in their normal sense while the integral containing the kernel F_{ij} is strongly singular and exists in the sense of its Cauchy principal value together with a free term that is absorbed in the left-hand side. The Green functions G_{ij} in equation (2.2.3) are fundamental solutions of the corresponding Navier equations of anisotropic elastostatics for two- and three-dimensions and are described below.

For the two-dimensional case (Snyder and Cruse [104]), the fundamental solutions are given by the following closed forms:

$$G_{kj} = 2Re[A_{k1}B_{j1}lnZ_1 + A_{k2}B_{j2}lnZ_2] \qquad (k, j = 1, 2) \tag{2.2.4}$$

where the quantities on the right of relation (2.2.4) are defined below (equations (2.2.4a) through (2.2.4g)).

The complex constants A_{kl} are given as follows:

$$\left\{ \begin{array}{c} A_{1l} \\ A_{2l} \end{array} \right\} = \left\{ \begin{array}{c} b_{11}\mu_l^2 + b_{12} - b_{16}\mu_l \\ b_{12}\mu_l + \frac{b_{22}}{\mu_l} - b_{26} \end{array} \right\} \qquad (l = 1, 2) \tag{2.2.4a}$$

The complex quantities $\mu_k (k = 1, 2)$ together with the conjugates $\bar{\mu}_k (k = 1, 2)$ are the roots of the following fourth order characteristic equation in terms of reduced stiffness coefficients b_{ij}:

$$b_{11}\mu^4 - 2b_{16}\mu^3 + (2b_{12} + b_{66})\mu^2 - 2b_{26}\mu + b_{22} = 0 \tag{2.2.4b}$$

It is assumed above that μ_k are the roots of equation (2.2.4b) with positive imaginary parts.

The complex constants B_{kl} are solutions of the following equations (2.2.4c)-(2.2.4f):

$$\mu_1 B_{j1} - \bar{\mu}_1 \bar{B}_{j1} + \mu_2 B_{j2} - \bar{\mu}_2 \bar{B}_{j2} = \frac{\delta_{j1}}{2\pi i}, \quad i = \sqrt{-1} \tag{2.2.4c}$$

$$B_{j1} - \bar{B}_{j1} + B_{j2} - \bar{B}_{j2} = -\frac{\delta_{j2}}{2\pi i}, \quad i = \sqrt{-1} \tag{2.2.4d}$$

$$A_{11} B_{j1} - \bar{A}_{11} \bar{B}_{j1} + A_{12} B_{j2} - \bar{A}_{12} \bar{B}_{j2} = 0 \tag{2.2.4e}$$

$$A_{21} B_{j1} - \bar{A}_{21} \bar{B}_{j1} + A_{22} B_{j2} - \bar{A}_{22} \bar{B}_{j2} = 0 \tag{2.2.4f}$$

($j = 1, 2$ in equations (2.2.4c)-(2.2.4f)).

The complex variables $Z_k (k = 1, 2)$ are defined as

$$Z_k = (x_1 - \xi_1) + \mu_k (x_2 - \xi_2) \tag{2.2.4g}$$

The traction kernel $F_{ij}(\mathbf{x}, \xi)$ in equation (2.2.3) is obtained by suitable differentiations of $G_{ij}(\mathbf{x}, \xi)$ and appropriate applications of the constitutive relations and the Cauchy equilibrium equations on the boundary.

In the previous applications of the boundary element method for general anisotropy in three-dimensions, Wilson and Cruse [107] used Green functions in the following form (Synge [105]):

$$G_{ij}(\mathbf{x}, \xi) = \frac{1}{8\pi^2 |\mathbf{x} - \xi|} \oint_{|t|=1} K_{ij}^{-1}(\mathbf{t}) ds \qquad (i, j = 1, 2, 3) \tag{2.2.5}$$

where the line integral is taken on the unit circle in the plane normal to the vector $(\mathbf{x} - \xi)$ and passing through $\mathbf{x}$, and the function

$$K_{ij}^{-1}(\mathbf{t}) = [c_{ijkm} t_k t_m]^{-1} \tag{2.2.5a}$$

Since it is not possible to evaluate the line integral of equation (2.2.5) in closed form for the general case, numerical integration was employed. The boundary element procedure based on the above numerically computed Green functions proved to be extremely time-consuming despite improvements in which interpolation schemes are used to avoid actually using relationship (2.2.5) at every Gauss point (Wilson and Cruse [107]). A more efficient approach is adopted in the present work in which the Green functions are calculated at the grid points and bivariate cubic spline functions are then fitted through these points for the purpose of interpolation. The Green functions adopted in the present work can be expressed in terms of spherical coordinates as (Deb et al [106]):

$$G_{ij}(r, \theta, \phi) = -\frac{1}{4\pi r} \sum_{\nu=1}^{6} \mathrm{Sign}(Re(\nu)) \eta_i(\nu) \eta_j(\nu) \qquad (i, j = 1, 2, 3) \tag{2.2.6}$$

where, in the real variable formulation, the ν and $\eta(\nu)$ are the eigenvalues and eigenvectors of a 6x6 real matrix (Malen [101]). The eigenvalues and eigenvectors occur as complex conjugates. The ϕ derivative of G_{ij} can be expressed as

$$G_{ij,\phi} = -\frac{1}{4\pi r} \sum_{\nu=1}^{6} Sign(Re(\nu)) \left[\frac{\partial \eta_i(\nu)}{\partial \phi} \eta_j(\nu) + \eta_i(\nu) \frac{\partial \eta_j(\nu)}{\partial \phi} \right] \quad (i, j = 1, 2, 3) \tag{2.2.7}$$

with a similar expression for $G_{ij,\theta}$. These derivatives are required for the computation of the traction kernels F_{ij}. For the purpose of calculating interior stresses and strains, the necessary second derivatives have also been developed.

2.3 HEAT CONDUCTION ANALYSIS

The governing differential equation for transient heat conduction can be written as

$$k\frac{\partial^2 T}{\partial x_i \partial x_i} - \rho c\frac{\partial T}{\partial t} + \psi = 0, \tag{2.3.1}$$

with temperature T, conductivity k, density ρ, specific heat c, and body sources ψ. The corresponding reciprocal theorem

$$\int_S q(x) * T^*(x)ds + \int_V \psi(x) * T^*(x)dv = \int_S q^*(x) * T(x)ds + \int_V \psi^*(x) * T(x)dv \tag{2.3.2}$$

relates any two states of temperature T, heat flux q, and ψ body sources occupying in the volume V with surface S. Note that since convolutions appear in (2.3.2), the entire time history is involved. Using the infinite space fundamental solution of (2.3.1) 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:

$$C(\xi)T(\xi,t) = \int_S [G(x,\xi,t) * q(x,t) - F(x,\xi,t) * T(x,t)]ds(x). \tag{2.3.3}$$

In (2.3.3), 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

$$k\frac{\partial^2 T}{\partial x_i \partial x_i} + \psi = 0, \tag{2.3.4}$$

and the boundary integral equation becomes

$$C(\xi)T(\xi) = \int_S [G(x,\xi)q(x) - F(x,\xi)T(x)]ds(x). \tag{2.3.5}$$

G and F are now the well-known potential flow kernel functions. Equation (2.3.5) is, of course, time independent. For more information, please refer to Banerjee and Butterfield [2] and Dargush and Banerjee [42].

GPBEST contains multiregion capability for both steady-state and transient heat conduction as boundary-only problems. The spatial and temporal discretization of (2.3.5) and (2.3.3) 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. As a 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 (2.3.3) is valid at any point ξ, 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 (2.3.3) 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,

$$C(\xi)T(\xi,t) = \int_C [G^c(x,\xi,t) * q(x,t) - F^c(x,\xi,t) * T(x,t)]dc(x), \tag{2.3.6}$$

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

$$\begin{aligned} C(\xi)T(\xi,t) = &\int_S [G(x,\xi,t) * q(x,t) - F(x,\xi,t) * T(x,t)]ds(x) \\ &+ \sum_{n=1}^{n} \Big[G^c(x,\xi,t) * q(x,t), -F^c(x,\xi,t) * T(x,t) \Big] dc_n(x). \end{aligned} \tag{2.3.7}$$

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 (2.3.1) or (2.3.4). For example, the steady flow of inviscid incompressible fluids reduces to (2.3.4), as does problems of electrostatics, while the diffusion of chemical substance is represented by (2.3.1). In all of these cases, with an appropriate translation of the field variables, **GPBEST** is directly applicable.

2.4 THERMOELASTIC ANALYSIS

In the boundary element method the thermal contribution can be incorporated in the system in one of two ways.

(1) Thermal Boundary Approach - the temperature and fluxes are boundary only quantities that are solved concurrently with the diplacements and tractions on the boundary.

(2) Body Force Approach via Particular Integral - the effect of the temperature distribution is incorporated in the system through a particular integral without the need of volume integration. Temperature distributions can be defined either as a piecewise quadratic variation or may be defined at discrete points through the body.

Thermal Boundary Approach

The formulation is based upon the theory of uncoupled thermoelasticity governed by the following set of differential equations:

$$(\lambda + \mu)u_{j,ij} + \mu u_{i,jj} - \beta T_{,i} + f_i = 0 \qquad (2.4.1a)$$

$$kT_{,jj} - \rho c\dot{T} + \psi = 0 \qquad (2.4.1b)$$

which involves displacement (u_i), temperature (T), Lame elastic moduli (λ, μ), thermal conductivity (k), density (ρ), specific heat (c), body forces (f_i) and body sources (ψ). Additionally, the parameter β can be written

$$\beta = (3\lambda + 2\mu)\alpha$$

where α is the coefficient of thermal expansion.

Based upon (2.4.1), a generalized boundary integral statement can be obtained in the time domain. Specifically, in the absence of the body forces and sources,

$$c_{\beta\alpha}(\xi)u_\alpha(\xi,t) = \int_S [G_{\beta\alpha}(x,\xi,t) * t_\alpha(x,t) - F_{\beta\alpha}(x,\xi,t) * u_\alpha(x,t)]ds(x) \qquad (2.4.2)$$

where u_α and t_α are generalized displacements and tractions, which include temperature and heat flux, respectively, as an additional component. The kernels $G_{\beta\alpha}$ and $F_{\beta\alpha}$ are derived from the point force and point source fundamental solutions, and $c_{\beta\alpha}$ is the discontinuity tensor. The nature of $G_{\beta\alpha}$ and $F_{\beta\alpha}$ permit certain simplifications of (2.4.2) which are accounted for in **GPBEST** . For example, convolution is actually only needed for the temperature and flux field variables. Further simplifications result in steady-state analyses. For additional information please refer to Dargush and Banerjee [45,46].

The **GPBEST** implementation provides complete displacement, stress, temperature and heat flux information at each time step. Consequently, this option is called Concurrent Thermoelasticity, and is available for both transient and steady-state problems.

Thermal Body Force Approach via Particular Integrals

The governing differential equation for a homogenous, isotropic body subjected to body force loading, as stated earlier, can be expressed in operator notation as [35,38,50,51,62]:

$$L(u_i) + f_i = 0 \tag{2.4.3}$$

where

$L(u_i) = (\lambda + \mu)u_{i,ji} + \mu u_{i,jj}$

u_i represents the displacement vector,

$f_i = -\beta T_{,i}$ for thermoelastic loading, with $\beta = (3\lambda + 2\mu)\alpha$, and,

$f_i = -\sigma^{\circ}_{ij,j}$ for initial stress (nonlinear) problems

When other body forces are present, the net body force is the summation of the effects of individual body forces.

A solution satisfying a linear, inhomogenous, 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 [35,38,50,51,52]

$$L(u_i^c) = 0 \tag{2.4.4}$$

and a particular integral u_i^p satisfying the inhomogenous equation

$$L(u_i^p) = f_i \tag{2.4.5}$$

The total solution u_i is expressed as

$$u_i = u_i^c + u_i^p \tag{2.4.6}$$

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 ξ, is expressed as

$$c_{ij}(\xi)u_i^c(\xi) = \int_S [G_{ij}(x,\xi)t_i^c(x) - F_{ij}(x,\xi)u_i^c(x)]dS(x) \tag{2.4.7}$$

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

where the u_i^c and t_i^c are the complementary functions for displacement and traction, respectively. The total solution for displacement u_i and traction t_i are

$$u_i = u_i^c + u_i^p$$

$$t_i = t_i^c + t_i^p \tag{2.4.8}$$

where u_i^p and t_i^p 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 (2.4.3) is a particular integral, regardless of boundary conditions or how it was obtained.

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) [50,51].

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

$$Gt^c - Fu^c = 0 \tag{2.4.9}$$

By introducing equation (2.4.8) into equation (2.4.9), the complementary function can be eliminated;

$$Gt - Fu = Gt^p - Fu^p \tag{2.4.10}$$

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 (2.4.9), 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 (2.4.10).

2.5 FRACTURE MECHANICS ANALYSIS

Initiation and propagation of cracks in elastic solids under loads are of considerable importance since they are crucial in predicting the failure of structures. For crack propagation, the stress-intensity at the crack-tip must exceed some critical value and the stress intensity factor (SIF) plays a key role in characterizing this behavior.

The boundary element method because of its ability to produce accurate stresses in region with high stress gradient is well suited for this type of analysis. However the crack-tip displacement and stress fields cannot be represented accurately by regular quadratic boundary elements. Both traction-singular and ordinary quarter-point elements (explained later) are employed for this purpose.

Special Elements

It is well known that the behavior of displacement and stress near the crack-tip are $O(\sqrt{r})$ and $O(\frac{1}{\sqrt{r}})$ respectively. Thus special type of elements are needed near the crack tip for proper modeling of the correct variation of the stresses and displacements. These elements are developed by simple modification of the isoparametric quadratic elements. These are:

(i) Quarter point (QP) element

By shifting the mid-node to the quarter-point from the crack-tip, the correct variation of displacement can be obtained. However, because of the use of same shape function, the traction and thus stress variation is also similar to that of displacement.

(ii) Traction Singular (TS) Element

In traction singular elements, the tractions are modified by multiplying the actual traction by a non-dimensional parameter $\sqrt{\frac{l}{r}}$, where l is the element length in the direction, perpendicular to crack front.

Stress Intensity Factor (SIF)

The stress intensity factors are of three types depending upon the type of crack,

(i) Mode I - the opening mode
(ii) Mode II - in-plane shear mode
(iii) Mode III - out-of-plane shear mode

These factors can be determined either from nodal tractions or from the crack opening displacements.

From Nodal Traction:

$$K_I = t_A^2 \sqrt{2\pi\ell}$$

$$K_{II} = t_A^1 \sqrt{2\pi\ell}$$

where, K_I and K_{II} are the mode-I (opening mode) and mode-II (shearing mode) stress intensity factors respectively; t_A refers to the modified traction at node A (the crack-tip), while 1 and 2 indicate, respectively, the direction of crack-propagation and the tranverse direction. The length of the element is represented by ℓ. It is found that in 3D analysis, the SIFs based on tractions are not accurate due to the nature of the integration used.

From Crack Opening Displacement:

Referring to Figure 2.5.1, the stress intensity factors are given by Blandford et al [109]:

$$K_I = \frac{\mu}{\kappa+1}\sqrt{\frac{2\pi}{\ell}}[4(v_B - v_D) + v_E - v_C]$$

$$K_{II} = \frac{\mu}{\kappa+1}\sqrt{\frac{2\pi}{\ell}}[4(u_B - u_D) + u_E - u_C]$$

where, μ is the shear modulus, $\kappa = 3 - 4\nu$ for plane strain and $(3-\nu)/(1+\nu)$ for plane stress; u and v indicate the displacement along the crack-propagation and the transverse displacement respectively. The points B, C, D and E are defined in Figure 2.5.1.
For a symmetric crack

$$v_B = -v_D\,; \qquad v_C = -v_E,$$

then expression for K_I simplifies to:

$$K_I = \frac{2\mu}{\kappa+1}\sqrt{\frac{2\pi}{\ell}}(4v_B - v_C).$$

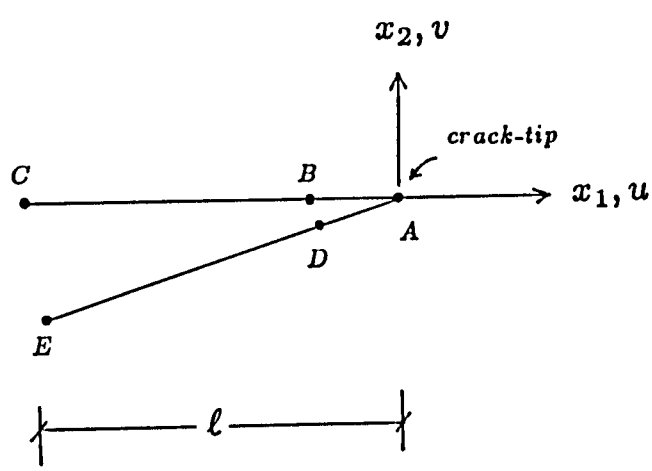


Figure 2.5.1 Element geometry near crack-tip

Alternatively, the SIFs may be obtained from the displacements at the quarter points using

$$K_I = \frac{H}{8}\sqrt{\frac{2\pi}{r}}\Delta U_2$$

$$K_{II} = \frac{H}{8}\sqrt{\frac{2\pi}{r}}\Delta U_1$$

$$K_{III} = \frac{H}{8(1-\nu)}\sqrt{\frac{2\pi}{r}}\Delta U_3$$

where $\Delta U_1, \Delta U_2$ and ΔU_3 are crack opening displacements in mode I, II and III directions, $H = E$ for plane stress and $H = \frac{E}{(1-\nu^2)}$ for plane strain.

It should be noted that the stress at the crack tip is infinite. Numerical values determined at these points, based upon a boundary stress calculation, are meaningless. Only the stress intensity factor has significance.

2.6 POROELASTIC ANALYSIS

GPBEST includes a unique boundary-only formulation for Biot's consolidation theory. The governing differential equations for poroelasticity are written

$$(\lambda + \mu)u_{j,ij} + \mu u_{i,jj} - \beta p_{,i} + f_i = 0 \tag{2.6.1a}$$

$$kp_{,jj} - \frac{\beta^2}{\lambda_u - \lambda}\dot{p} - \beta \dot{u}_{j,j} + \psi = 0 \tag{2.6.1b}$$

which involves displacement (u_i), excess pore pressure (p), drained elastic moduli (λ, μ), permeability (k), undrained moduli (λ_u), body forces (f_i) and body sources (ψ). Additionally, the parameter β can be written

$$\beta = \frac{3}{B}[\frac{\lambda_u - \lambda}{3\lambda_u + 2\mu}]$$

where B is Skempton's pore pressure coefficient. $K,\ K_u$, the drained and undrained bulk modulus, $\nu,\ \nu_u$, the drained and undrained Poison's ratio.

Based upon (2.6.1), a generalized boundary integral statement can be obtained in the time domain. Specifically, in the absence of body forces and sources,

$$c_{\beta\alpha}(\xi)u_\alpha(\xi, t) = \int_S [G_{\beta\alpha}(x, \xi, t) * t_\alpha(x, t) - F_{\beta\alpha}(x, \xi, t) * u_\alpha(x, t)]ds(x) \tag{2.6.2}$$

where u_α and t_α are generalized displacements and tractions, which include pore pressure and flux, respectively, as an additional component. The kernel functions $G_{\beta\alpha}$ and $F_{\beta\alpha}$ are derived from the point force and point source fundamental solutions, and $c_{\beta\alpha}$ is the discontinuity tensor. For additional information please refer to Dargush and Banerjee [44].

The **GPBEST** implementation includes three-dimensional, two- dimensional, and axisymmetric options under both transient and steady-state (drained) conditions. In general, displacements, effective stresses, pore pressures and mass flux are calculated at each time step.

2.7 PERIODIC ACOUSTIC ANALYSIS

2.7.1 PURE ACOUSTICS

The governing differential equation for propagation of acoustic waves can be written as:

$$\phi_{,ii} - \frac{1}{c^2}\frac{\partial^2 \phi}{\partial t^2} = 0 \qquad (2.7.1.1)$$

where

c is the speed of sound

ϕ is the potential

The solution of the above equation for a unit harmonic point force of the form $e^{-i\omega t}$ is known as the fundamental solution. This solution is denoted in the form as $G(x,\xi,\omega)$ and the corresponding solution for flux is $F(x,\xi,\omega)$. Both can be found in reference [2].

The boundary integral equation, in the frequency domain, can be derived by writing the dynamic reciprocal work theorem between the actual state and the above mentioned fundamental solution (point-force solution) state as (similar to section 2.1 and reference [2]):

$$c(\xi)\,\phi(\xi,\omega) = \int_S \left[G(x,\xi,\omega)\ \psi(x,\omega) - F(x,\xi,\omega)\ \phi(x,\omega)\right]\ dS(x) \qquad (2.7.1.2)$$

where S is the surface enveloping the body and ψ is the normal flux. $\left(\frac{\partial \phi}{\partial n}\right)$

In order to take into account the variations in density across the interface in a piecewise homogeneous medium, equation (2.7.1.2) is modified by multiplying it by $i\omega\rho$ as

$$c(\xi)p(\xi,\omega) = \int_S \left[\hat{G}(x,\xi,\omega)\vartheta_n(x,\omega) - F(x,\xi,\omega)p(x,\omega)\right] dS \qquad (2.7.1.3)$$

where $p = i\omega\rho\phi$ and $\hat{G} = i\omega\rho G$ and ϑ_n is the normal velocity.

The above velocity-pressure form is implemented in the **GPBEST** system.

Although the solution to exterior problems is unique, the boundary element representation given above fails at frequencies associated with the corresponding interior eigenfrequency problem. One way to overcome this is by following the method proposed by Burton and Miller [124], in which a linear combination of the integral equation (2.7.1.2) and the normal derivative of that equation are combined through a complex parameter (e.g. [124])

NOTE: Periodic acoustic analysis involves complex quantities.

2.7.2 UNCOUPLED STRUCTURAL ACOUSTICS - ONE-WAY COUPLING

For a specific frequency, ω, any discretized structural system can be solved for the specified mechanical loading and prescribed boundary conditions, using a Finite Element technique as:

$$[(K - \omega^2 M) + i\omega D]\{u\} = \{f\} \tag{2.7.2.1}$$

where K is the assembled structural stiffness matrix
M is the assembled mass matrix
D is the assembled damping matrix
u is the unknown nodal displacement vector
f is the assembled nodal load vector
i is the $\sqrt{-1}$

Once the structural system is solved, the nodal displacements are now known. Hence under identical hypothesis (harmonic loadings cause harmonic deflections) it is possible to obtain normal velocities from the normal component of displacements:

$$\{\vartheta_n\} = i\omega\{u_n\}$$

where u_n is the normal displacement.

Once the above velocity values are input into the pure acoustic problem, whether it is being solved by the *Direct* BEM formulation (equation 2.7.1.3) or the *Indirect* BEM technique (equation 2.7.1.5), the basic process of initiating a one-way coupling between the structure and the fluid is complete.

In terms of the system equation, the whole process looks like the following:

a) Indirect BEM formulation:

$$\begin{bmatrix} \hat{K} & O \\ C_I & Q_I \end{bmatrix} \begin{Bmatrix} \hat{u} \\ \mu \end{Bmatrix} = \begin{Bmatrix} \hat{f} \\ I_I \end{Bmatrix} \tag{2.7.2.2}$$

where $\hat{K}$ = Reduced structural matrix
C_I = Reduced coupling matrix (indirect)
Q_I = Fluid matrix (indirect)
$\hat{u}_n$ = Normal displacement vector
μ = Pressure-drop vector
$\hat{f}$ = Reduced mechanical force vector
I_I = Right hand side (indirect) acoustic vector including the effects of specified acoustic sources inside the acoustic fluid region

The first set is solved: $\{\hat{u}_n\} = [\hat{K}]^{-1}\{\hat{f}\}$

The second set is then solved: $\{\mu\} = [Q]^{-1}(\{I_I\} - [C]\{\hat{u}_n\})$

Equation (2.7.1.4) may now be used with a known distribution $\{\mu\}$ to find the pressure anywhere within the fluid.

b) Direct BEM formulation:

The acoustic part may be written as (see (2.7.1.3))

$$[A]\{p\} = [B]\{\vartheta_n\}$$

where $[A]$ and $[B]$ can be formed from (2.7.1.2). Thus,

$$\begin{bmatrix} \hat{K} & O \\ C_D & A \end{bmatrix} \begin{Bmatrix} \hat{u}_n \\ p \end{Bmatrix} = \begin{Bmatrix} \hat{f} \\ I_D \end{Bmatrix} \qquad (2.7.2.3)$$

where C_D = Reduced acoustic coupling matrix (direct)
I_D = Right hand side (direct) acoustic vector which includes the effects of specified body sources

The first set is solved and then the second set gives $\{P\}$. The pressure anywhere else may be found using (2.7.1.3).

2.7.3 FULLY-COUPLED STRUCTURAL ACOUSTICS

a) Indirect BEM formulation:

$$\begin{bmatrix} \hat{K} & C_I \\ C_I & Q_I \end{bmatrix} \begin{Bmatrix} \hat{u} \\ \mu \end{Bmatrix} = \begin{Bmatrix} \hat{f} \\ I_I \end{Bmatrix} \tag{2.7.3.1}$$

which is really the full system matrix, instead of the approximate system given in (2.7.2.2). The solution gives the unknown nodal displacements and pressure drops. (2.7.1.4) may be used with the known distribution μ to find the pressure anywhere within the fluid.

b) Direct BEM formulation:

$$\begin{bmatrix} \hat{K} & C_D \\ C_D & A \end{bmatrix} \begin{Bmatrix} \hat{u}_n \\ p \end{Bmatrix} = \begin{Bmatrix} \hat{f} \\ I_D \end{Bmatrix} \tag{2.7.3.2}$$

which again represents the full system instead of the approximate one given in (2.7.2.3). The solution gives the unknown nodal displacements and pressures. The pressure elsewhere may be found using (2.7.1.3).

It should be noted that a fully coupled system takes more time to solve than the corresponding one-way coupled system. However, the latter case is just an approximation valid only for stiff structures with very light fluid loading.

2.8 PERIODIC DYNAMIC ANALYSIS

Boundary Integral Equation

The governing differential equation for a linear, elastic isotropic body undergoing time-harmonic excitation can be written as:

$$(c_1^2 - c_2^2)\, u_{j,ij} + c_2^2\, u_{i,jj} + b_i - \rho\omega^2\, u_i = 0 \tag{2.8.1}$$

where

c_1 and c_2 are the pressure and shear wave speeds

u_i and b_i are the vectors of displacement and body force respectively (in the transformed frequency domain)

The solution of the above equation for a unit harmonic point force of the form $e^{i\omega t}$ is known as the fundamental solution. This solution is denoted in the tensor form as $G_{ij}(x,\xi,\omega)$. The corresponding tensor for traction is $F_{ij}(x,\xi,\omega)$. They can be found in references [41] and [59] for the three-dimensional and two-dimensional problems. These functions depend on elastic modulus E, Poisson's ratio ν and density ρ of the material.

The boundary integral equation, in the frequency domain, can be derived by writing the dynamic reciprocal work theorem between the actual state and the above mentioned fundamental solution (point-force solution) state as (similar to section 2.1 and reference [2]):

$$c_{ij}(\xi)\, u_i(\xi,\omega) = \int_S \left[G_{ij}(x,\xi,\omega)\, t_i(x,\omega) - F_{ij}(x,\xi,\omega)\, u_i(x,\omega)\right] dS(x) \tag{2.8.2}$$

where S is the surface enveloping the body

In the above equation the body force is neglected. It should be noted here that although the functions G_{ij} and F_{ij} becomes identical to their static counterpart as ω tends to zero, it is important to evaluate them at this limit carefully because of the presence of ω in the denominator.

Material Damping

The periodic dynamic formulation can take account of internal viscous dissipation of energy (damping). This is acomplished by a simple transformation of the Lame's constants into complex form,

$$\lambda^* = \lambda(1 + 2i\beta)$$

$$\mu^* = \mu(1 + 2i\beta)$$

where β is the internal material damping parameter.

Interior Quantities

Once the boundary solution is obtained, the interior version of equation (2.8.2) can be used to find the interior displacements; and the interior stresses can then be obtained using the strain-displacement and stress-strain relations as:

$$\sigma_{jk}(\xi,\omega) = \int_S \left[G^{\sigma}_{ijk}(x,\xi,\omega)\ t_i(x,\omega) - F^{\sigma}_{ijk}(x,\xi,\omega)\ u_i(x,\omega) \right]\ dS(x) \tag{2.8.3}$$

The stress kernels $G^{\sigma}_{ijk}(x,\xi,\omega)$ and $F^{\sigma}_{ijk}(x,\xi,\omega)$ can be found in reference [59]. This explicit BEM equation, however, can not be used to obtain stresses at points on the boundary because of high singularity of the kernel functions. The stresses at those points 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 [2].

$\mathbf{Note}$: The perodic dynamic formulation involves complex quantities.

2.9 FREE-VIBRATION ANALYSIS

A new method for free-vibration analysis by boundary element method using particular integrals has been implemented in the current version of **GPBEST** . 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, 48, 63].

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

$$L(u_i) + \rho w^2 u_i = 0 \tag{2.9.1}$$

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

$$L(u_i^c) = 0 \tag{2.9.2}$$

and a particular integral, u_i^p, satisfying

$$L(u_i^p) + \rho w^2 u_i = 0 \tag{2.9.3}$$

In equation (2.9.3), the unknown displacement u_i within the domain is approximated by an infinite summation of the products of an unknown fictitious density function, ϕ_k, and a known function D_{ik}. More specifically

$$u_i(x) \simeq \sum_{m=1}^{\infty} D_{ik}(x, \xi^m)\phi_k(\xi^m) \tag{2.9.4}$$

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

$$c_{ij}\, u_i^c(\xi) = \int_S [G_{ij}(x, \xi) t_i^c(x) - F_{ij}(x, \xi) u_i^c(x)] ds \tag{2.9.5}$$

where G_{ij} and F_{ij} are the elasto-static kernels (i.e. fundamental solutions of eq. 2.9.2).

By usual discretization of the boundary S, equation (2.9.6) can be expressed in a matrix form as

$$[G]\{t^c\} - [F]\{u^c\} = \{0\} \tag{2.9.6}$$

The complementary solutions are related to the real displacement u_i and tractions t_i via

$$u_i^c = u_i - u_i^p \tag{2.9.7}$$

$$t_i^c = t_i - t_i^p \tag{2.9.8}$$

Substituting equations (2.9.4), (2.9.7) and (2.9.8) into equation (2.9.6), the following equations [29] are obtained

$$[G]\{t\} - [F]\{u\} = \rho w^2 [M]\{u\} \tag{2.9.9}$$

By incorporating the known boundary conditions (i.e. at a node either $t_i = 0$ or $u_i = 0$) in eq. 2.9.9, an algebraic expression results for eigenvalue extraction, i.e.

$$[A]\{x\} = \rho w^2 [\overline{M}]\{x\} \tag{2.9.10}$$

Finally, equation (2.9.10) is solved by using an eigenvalue extraction routine developed by Boundary Element Software Technology Corporation based on some earlier efforts of Wilson, Miller and Banerjee and those of Garbow of Argonne National Laboratory to obtain the eigen frequencies and mode shapes.

This analysis is available for two and three dimensional problems.

2.10 ACOUSTIC EIGEN-FREQUENCY ANALYSIS

The boundary element formulation for acoustic eigen-frequency analysis implemented in **GPBEST** is based on the classical concept of complementary functions and particular integrals to solve non-homogenous differential equations. The governing differential equation for the acoustic eigen-value problem can be written as

$$\frac{\partial^2 p}{\partial x_i \partial x_i} + \left(\frac{w}{c}\right)^2 p = 0 \tag{2.10.1}$$

where p is the sound pressure, c is the speed of sound and w is the eigen frequency. Equation (2.10.1) can also be written in operator notation as

$$L(p) + \left(\frac{w}{c}\right)^2 p = 0 \tag{2.10.2}$$

The solution of the above equation can be represented as the sum of a complementary function, p^c, satisfying

$$L(p^c) = 0 \tag{2.10.3}$$

and a particular integral, p^p, satisfying

$$L(p^p) + \left(\frac{w}{c}\right)^2 p^p = 0 \tag{2.10.4}$$

In equation (2.10.4), the sound pressure p within the domain is approximated by an infinite summation of the products of an unknown fictitious density function, ϕ, and a known function D, i.e. [47]

$$p(x) = \sum_{m=1}^{\infty} D(x, \xi^m)\phi(\xi^m) \tag{2.10.5}$$

The direct boundary integral equation related to the complementary pressure p^c can be written as

$$C(\xi)p^c(\xi) = \int_S [G(x,\xi)\frac{\partial p^c}{\partial n}(x) - F(x,\xi)p^c(\xi)]ds \tag{2.10.6}$$

where G and F are the fundamental solutions of the Laplace equation (2.10.3).

By the usual discretization of the boundary S, equation (2.10.6) can be expressed in matrix form as

$$[G]\{\frac{\partial p^c}{\partial n}\} - [F]\{p^c\} = \{0\} \tag{2.10.7}$$

The complementary solutions are related to the real pressure p and normal gradient $\frac{\partial p}{\partial n}$ via

$$p^c = p - p^p \tag{2.10.8}$$

$$\frac{\partial p^c}{\partial n} = \frac{\partial p}{\partial n} - \frac{\partial p^p}{\partial n} \tag{2.10.9}$$

Substituting equations (2.10.5), (2.10.8) and (2.10.9) into equation (2.10.7), the following equations are obtained

$$[G]\{\frac{\partial p}{\partial n}\} - [F]\{p\} = \rho\omega^2[M]\{p\} \tag{2.10.10}$$

By incorporating the known boundary conditions (i.e., either $p = 0$ or $\frac{\partial p}{\partial n} = 0$ in equation (2.10.10), an algebraic expression results for eigenvalue extraction, i.e.,

$$[A]\{X\} = \rho\omega^2[\bar{M}]\{X\} \tag{2.10.11}$$

Finally, equation (2.10.11) is solved by using an eigenvalue extraction routine developed by Boundary Element Software Technology Corporation based on some earlier efforts of Wilson, Miller and Banerjee and those of Garbow of Argonne National Laboratory to obtain the eigen frequencies and mode shapes.

This analysis is available for two and three dimensional problems.

2.11 POINT SOURCE / FORCE ANALYSIS

An analysis for point sources/forces is obtained by adding the effect of source/force through the fundamental solution.For example, for an elastic problem, the boundary integral equation becomes,

$$C_{ij}(\xi)\, u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds(x) \tag{2.11.1}$$

In the presence of point force of magnitude f_i at $x = x_o$ the equation is modified as

$$C_{ij}(\xi)\, u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds(x) + G_{ij}(x_o,\xi)\, f_i(x_o) \tag{2.11.2}$$

Similar expression can be written for other analyses.

It should be noted that point forces introduce singularities at the point of application of the point force, therefore these points must not coincide with any node where the results are necessary.

In the present version of **GPBEST** point sources or forces are available in all interior regions (closed regions) involving all analyses covered in this manual. In the near future point sources or forces in the exterior regions will also be available. However, the user can easily simulate these effects by defining a small cavity and applying the necessary surface tractions or surface fluxes.

Figure 2.11.1 illustrates a two-dimensional elastic problem with a concentrated force acting at a point inside the region. Figure 2.11.2 depicts a concentrated heat source acting inside a three-dimensional body.

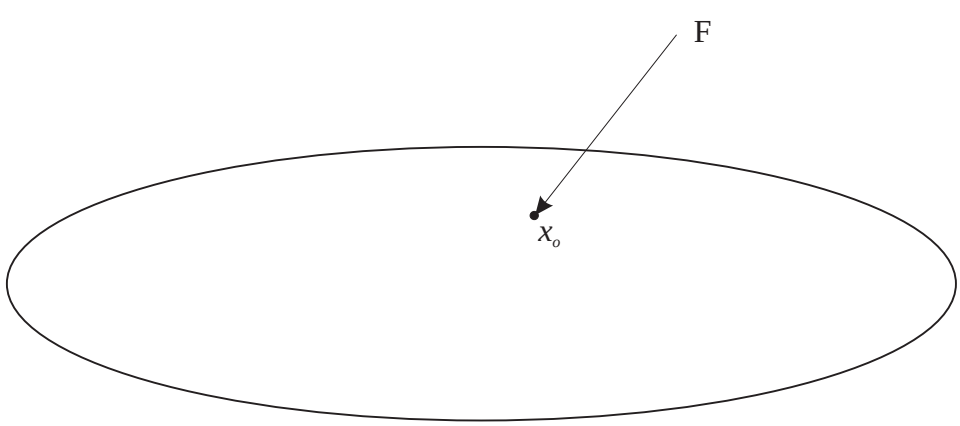


Figure 2.11.1 Concentrated Force at Interior Point x_o

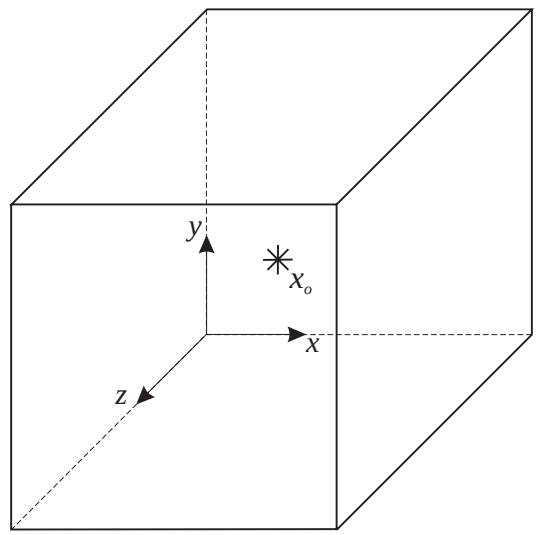


Figure 2.11.2 Concentrated Heat Source at Interior Point x_o

2.12 DISCONTINUITY ELEMENTS

Discontinuity elements are useful for a variety of specialty analyses. Basically, they are used to model a small disturbance inside a region. They are commonly used to model ground faults, faulting and other seismic phenomenon. They can also be used to model mining problems with ore extraction. Two-dimensional and three-dimensional discontinuity elements are available in **GPBEST** .

A two-dimensional square region is illustrated in Figure 2.12.1. In the interior is what appears to be a single boundary element, but in actuality is a special kind of element known as a discontinuity element. Figure 2.12.2 shows a close-up view of the discontinuity element. In particular, the element actually has two surfaces, $\mathcal{A}$ and $\mathcal{B}$, which are very close together. The key attribute of discontinuity elements is that the relative displacement between surfaces $\mathcal{A}$ and $\mathcal{B}$ is specified $apriori$. However, the specified displacements should be small with respect to the overall problem dimensions, as well as the discontinuity element length.

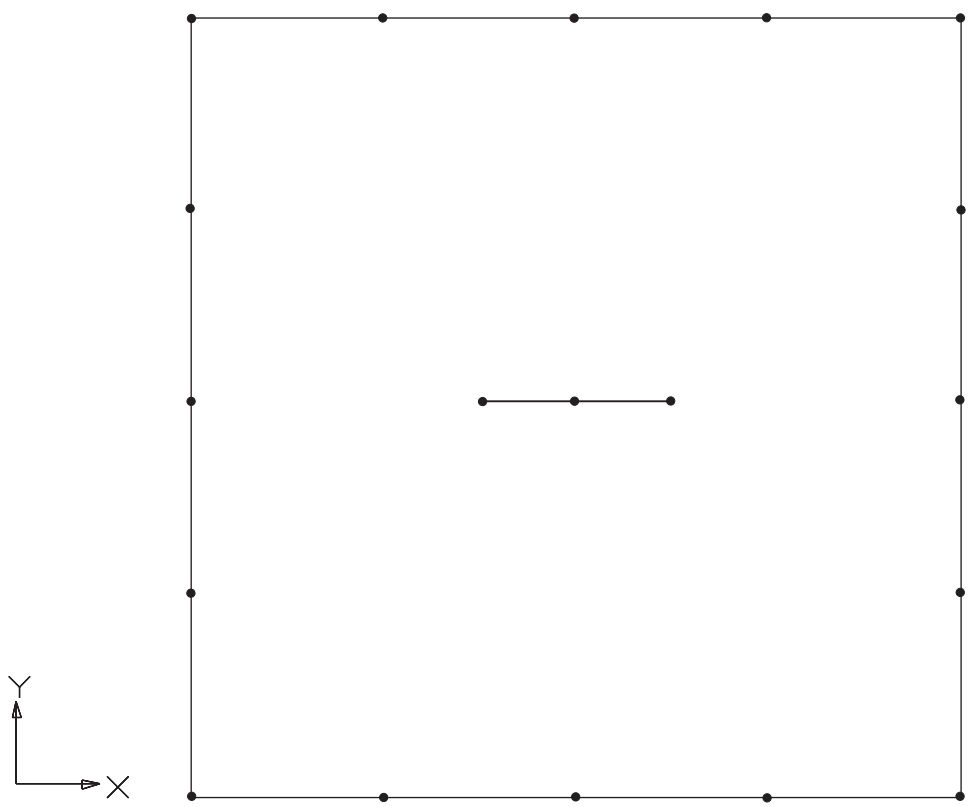


Figure 2.12.1 Two-dimensional Discontinuity Element Inside a Region

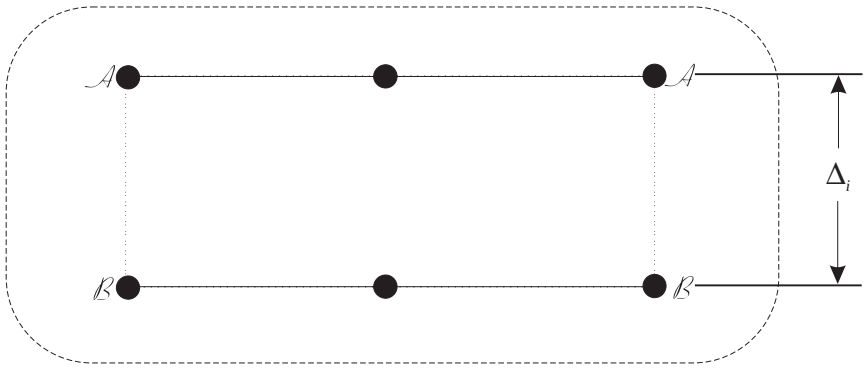


Figure 2.12.2 Close-up of Discontinuity Element

The use of discontinuity elements is intended to examine the effect of internal discontinuities on the boundary, or the far-field. They are not suitable for examining the region close to the discontinuity. Most often, they are used to examine the qualitative effects of various discontinuities on the boundary. As an example, one might use discontinuity elements to model a ground fault, but study the behavior at the ground surface (modeled with boundary elements). Figure 2.12.3 illustrates this situation.

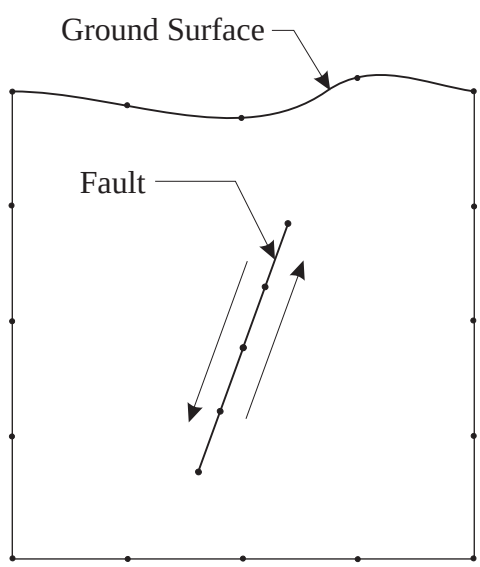


Figure 2.12.3 Modeling a Sub-Surface Fault

Some possible modes of deformation for a two-dimensional discontinuity element are illustrated in Figure 2.12.4. In general, the relative displacement between the two surfaces of the discontinuity element is given by (refer to Figure 2.12.2):

$$u_i^{\mathcal{B}} - u_i^{\mathcal{A}} = \Delta_i \tag{2.12.1}$$

where u_i is the displacement of surfaces $\mathcal{A}$ and $\mathcal{B}$, and Δ_i is the specified relative displacement. Δ_i can be specified on a node-by-node basis, but often, the discontinuity is uniform on each element. In **GPBEST** , Δ_i is specified as a special type of body force.

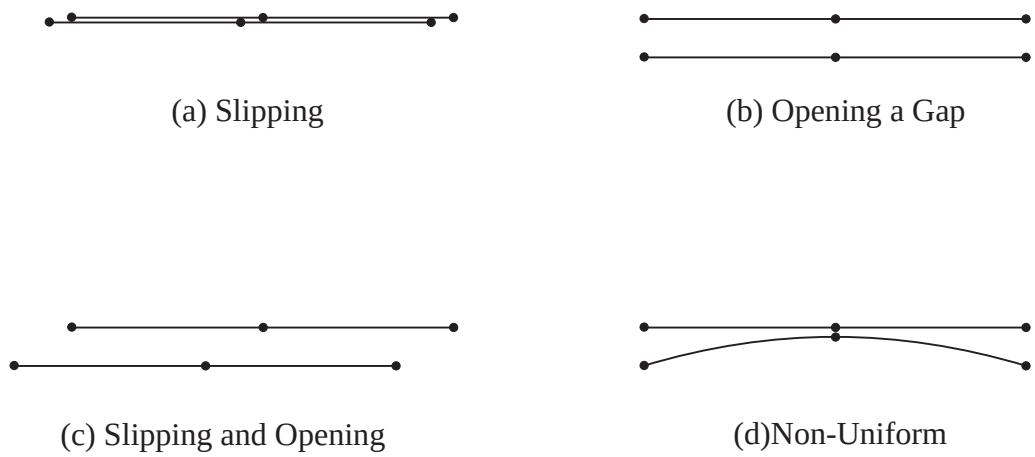


Figure 2.12.4 Deformation Modes for 2-D Discontinuity Elements

In order to develop the boundary element equations for discontinuity elements, consider the usual boundary integral, equation 2.1.7, while ignoring the volumetric body force term:

$$c_{ij}(\xi)\, u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_V G_{ij}(x,\xi)f_i(x)dv \qquad (2.12.2)$$

Consider the two-dimensional region of Figure 2.12.5. The integration path is shown outside the boundary for illustrative purposes only. In fact, the integration of surface $S(x)$ will take place along the boundary elements, over to the discontinuity element, and then the two surfaces of the discontinuity element. Hence, the total surface that is traversed can be broken up into three portions: $S = S_1 + S_2 + S_3$.

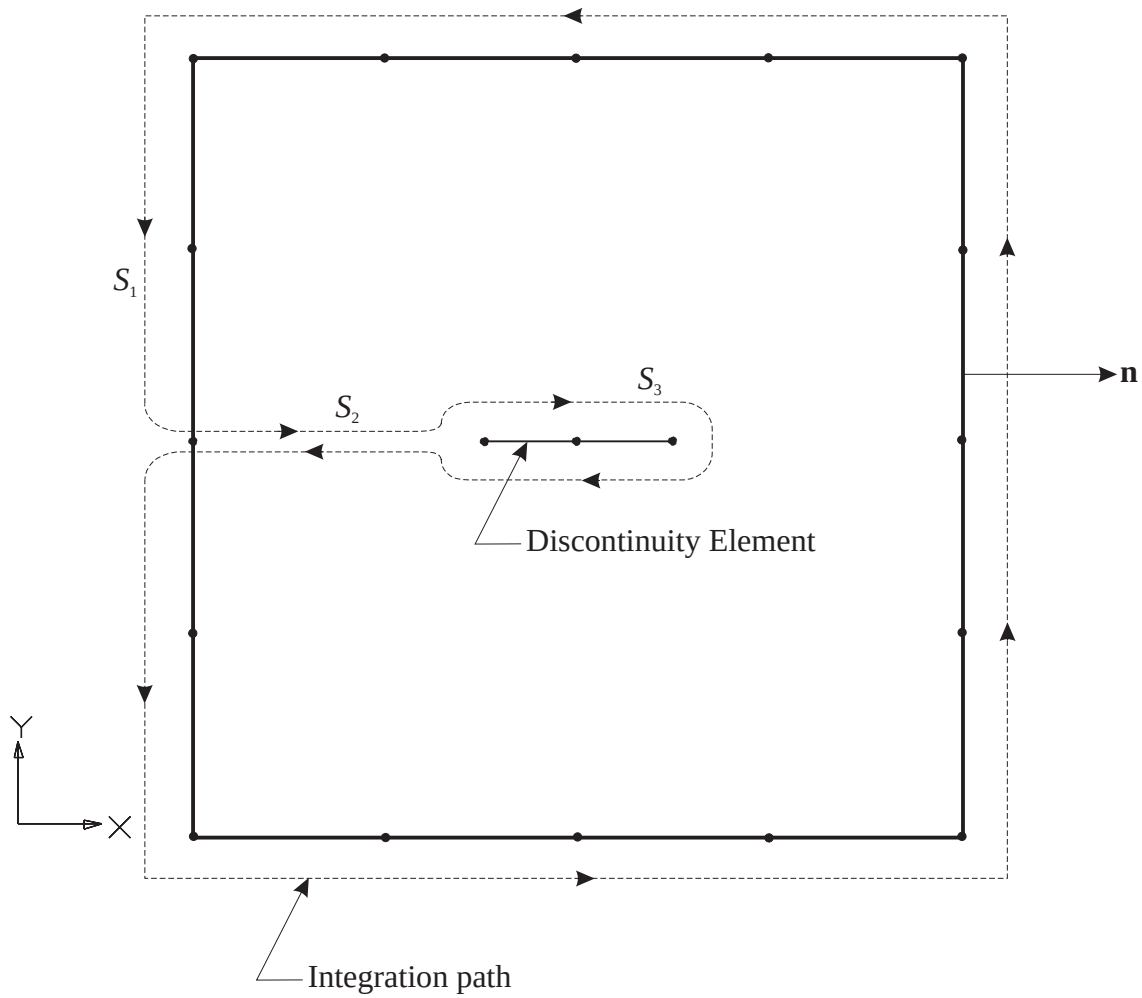


Figure 2.12.5 Integration Path

S_1 merely represents the outer boundary contribution to equation 2.12.2. Since surface S_2 is traversed in two opposing directions, the contribution of S_2 is zero. However, the contribution of S_3 will not be zero. Thus, this leaves the following integral equation:

$$c_{ij}(\xi)\, u_i(\xi) = \int_{S_1} [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_{S_3} [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds \qquad (2.12.2)$$

Figure 2.12.6 shows the integration path, S_3, over the two surfaces of the discontinuity element. Assuming that the two surfaces of the discontinuity element are very close together, the first term in the S_3 integral will result in zero contribution due to equal and opposite tractions. Traversing the second term in the S_3 integral over top and bottom surfaces, while realizing that they are traversed in an equal and opposite sense results in the following equation:

$$\int_{S_3} [-F_{ij}(x,\xi)u_i(x)]ds = -[u_i^{\mathcal{A}} - u_i^{\mathcal{B}}] \int_{S_3} F_{ij}(x,\xi)ds = +\Delta_i \int_{S_3} F_{ij}(x,\xi)ds \qquad (2.12.3)$$

where Δ_i is the imposed relative displacement given in equation 2.12.1.

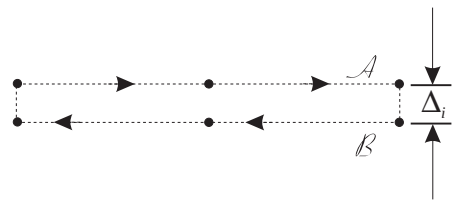


Figure 2.12.6 Integration Over a Discontinuity Element

Thus, the final integral equation, which includes the contribution for the discontinuity is:

$$c_{ij}(\xi)\, u_i(\xi) = \int_{S_1} [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \Delta_i \int_{S_3} F_{ij}(x,\xi)ds \qquad (2.12.4)$$

For more information regarding the application of discontinuity elements in solving problems with **GPBEST** , refer to Sections 5.3.7 and 5.6.7 of this manual.

2.13 INELASTIC ANALYSIS

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 (2.1.7) and (2.1.8b) are rewritten in incremental form in terms of the initial stresses as [18-25]:

$$c_{ij}(\xi)\dot{u}_i(\xi) = \int_S [G_{ij}(x,\xi)\dot{t}_i(x) - F_{ij}(x,\xi)\dot{u}_i(x)]ds + \int_V B_{ijk}(x,\xi)\dot{\sigma}^o_{ik}(x)dv \tag{2.13.1}$$

and

$$\dot{\sigma}_{ij}(\xi) = \int_S [G^\sigma_{kij}(x,\xi)\dot{t}_k(x) - F^\sigma_{kij}(x,\xi)\dot{u}_k(x)]ds + \int_{V-V_\varepsilon} M_{ijkl}(x,\xi)\dot{\sigma}^0_{kl}(x)dv + J_{ijkl}\dot{\sigma}^o_{kl}(\xi) \tag{2.13.2}$$

where $J_{ijkl}\dot{\sigma}^o_{kl}$ is a jump term arising out of analytical treatment of the volume integral over a small spherical region around ξ [25].

The volume integrals in (2.13.1) and (2.13.2) both involve unknown initial stresses whose evaluation involve the use of the nonlinear constitutive model. Thus equations (2.13.1), (2.13.2) 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 **GPBEST**, the material models provided are:

(a) Nonlinear, strain-hardening Von Mises model and temperature-dependent Von Mises model [2,7,16]

(b) Modified Cam-clay soil model and isotropic plasticity soil model [55]

The pertinent details for these models are provided below:

a. Von Mises Model and Temperature- Dependent Von Mises Model

Von Mises model can be considered as a special case of the temperature-dependent Von Mises model. All assumptions in deriving the incremental stress-strain relations of plasticity are based on the incremental theory of plasticity. The starting point is the standard decomposition of total incremental strain into elastic, plastic and thermal components,

$$\dot{\varepsilon}_{ij} = \dot{\varepsilon}^e_{ij} + \dot{\varepsilon}^p_{ij} + \dot{\varepsilon}^T_{ij}\ , \qquad \text{with } \dot{\varepsilon}^T_{ij} = \delta_{ij}\alpha\dot{T} \tag{2.13.3}$$

where α is the expansion coefficient of the material. It is convenient to define a mechanical strain rate as:

$$\dot{\varepsilon}^m_{ij} = \dot{\varepsilon}^e_{ij} + \dot{\varepsilon}^p_{ij} = \dot{\varepsilon}_{ij} - \dot{\varepsilon}^\theta_{ij} \tag{2.13.4}$$

Meanwhile, the elastic strain rate in (2.13.3) and (2.13.4) holds the elastic relationship to the stress rate. That is:

$$\dot{\sigma}_{ij} = D^e_{ijkl}\dot{\varepsilon}^e_{kl} = D^e_{ijkl}\dot{\varepsilon}^m_{kl} - D^e_{ijkl}\dot{\lambda}\frac{\partial F}{\partial \sigma_{kl}} \tag{2.13.5}$$

where D^e_{ijkl} can be described by Lame constants λ, μ as:

$$D^e_{ijkl} = \lambda\delta_{ij}\delta_{kl} + 2\mu\delta_{ik}\delta_{jl} \tag{2.13.6}$$

Also, we have the following relations between λ, μ, Young's modulus E, Poisson's ratio ν, shear modulus G and bulk modulus K:

$$\mu = G = \frac{E}{2(1+\nu)}, \quad \lambda = \frac{\nu E}{(1+\nu)(1-2\nu)}, \quad K = \lambda + \frac{2}{3}\mu \tag{2.13.7}$$

<u>Yielding Criterion</u>

The yielding surface of the temperature-dependent Von Mises model is defined as

$$F\left(\sigma_{ij}, T, \varepsilon^p\right) = \frac{1}{2}S_{ij}S_{ij} - \frac{1}{3}\sigma_s^2(\varepsilon^p)\Gamma(T) = 0 \tag{2.13.8}$$

where the deviatory stress is given by:

$$S_{ij} = \sigma_{ij} - \frac{1}{3}\delta_{ij}\sigma_{kk} \tag{2.13.9}$$

and the total plastic strain is defined by:

$$\varepsilon^p = \int \dot{\varepsilon}^p dt, \qquad \dot{\varepsilon}^p = \left(\frac{2}{3}\dot{\varepsilon}^p_{ij}\dot{\varepsilon}^p_{ij}\right)^{\frac{1}{2}} . \tag{2.13.10}$$

$\sigma_y(\varepsilon^p)$ is the simple tensile strength of the material. As shown in Figure 2.13.1, $\Gamma(T)$ is the temperature-dependent function assumed as:

$$\Gamma(T) = 1 - \left(\frac{T'}{T_{\text{melt}} - T_{\text{peak}}}\right)^2 \tag{2.13.11}$$

with

$$T' = \begin{cases} 0 & \text{for } T \le T_{\text{peak}} \\ T - T_{\text{peak}} & \text{for } T > T_{\text{peak}} \end{cases} \tag{2.13.12}$$

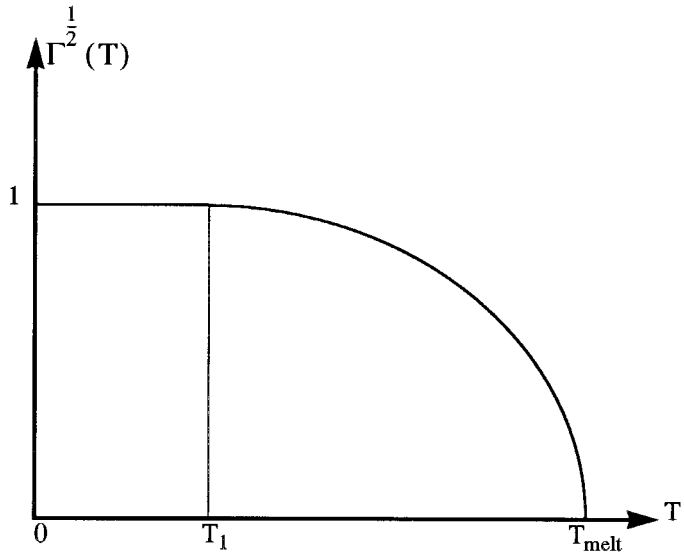


Figure 2.13.1 Shape of the Temperature- Dependent Function

Where T_{melt} is the melting temperature and T_{peak} is the temperature up to which the yield strength of the material remains constant. Also, the following derivative relations hold:

$$\frac{\partial F}{\partial \sigma_{ij}} = S_{ij} \tag{2.13.13}$$

Plastic Flow Rule

According to the plastic flow rule, the plastic strain rate is assumed as:

$$\dot{\varepsilon}^p_{ij} = \dot{\lambda}\frac{\partial F}{\partial \sigma_{ij}} = \dot{\lambda} S_{ij} \tag{2.13.15}$$

Substituting (2.13.5) and (2.13.8) to (2.13.10), we get:

$$\dot{\varepsilon}^p = \frac{2}{3}\dot{\lambda}\sigma_y(\varepsilon^p)\Gamma^{\frac{1}{2}}(T) \tag{2.13.16}$$

Hardening Rule

The hardening effect of the material is presented through function $\sigma_y(\epsilon^p)$. A typical function $\sigma_y(\varepsilon^p)$ of a hardening material is shown in Figure 2.13.2 and figure 2.13.3. Figure 2.13.2 is the experimental result normally obtained. It gives the relationship of yielding strength via total strain ε. Figure 2.13.3 is derived from Figure 2.13.2 by subtracting the elastic component from the total ε. σ_0 is the initial yielding strength, and H_p is the hardening slope known by:

$$H_p = \frac{d}{d\varepsilon^p}\sigma_y(\varepsilon^p) \tag{2.13.17}$$

The derivative of the yielding function to ε^p is then

$$\frac{\partial F}{\partial \varepsilon^p} = -\frac{2}{3}\sigma_y(\varepsilon^p)H_p\Gamma(T) \tag{2.13.18}$$

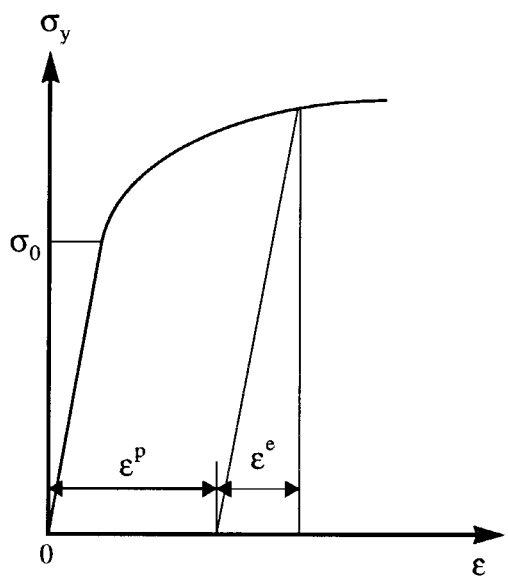


Figure 2.13.2 Simple Tension Test

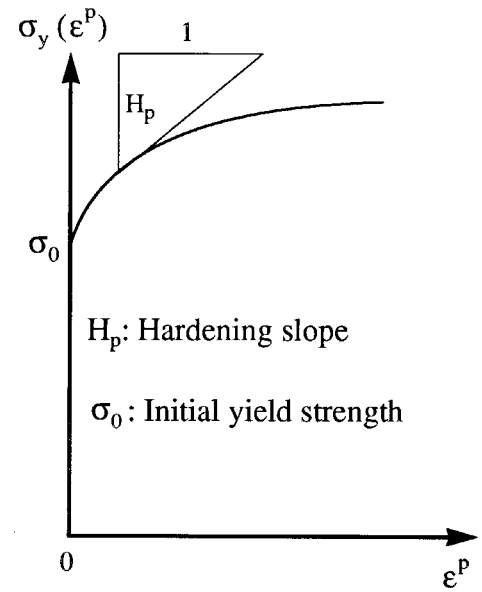


Figure 2.13.3 Definition of Hardening Slope

Incremental Stress-Strain Relationship

Based on the above rules, the incremental stress-strain relationship can now be derived. Since consistency relation requires the stress state must remain on newly developed surface during hardening, we have:

$$\dot{F}\left(\sigma_{ij}, T, \varepsilon^p\right) = \frac{\partial F}{\partial \sigma_{ij}}\dot{\sigma}_{ij} + \frac{\partial F}{\partial T}\dot{T} + \frac{\partial F}{\partial \varepsilon^p}\dot{\varepsilon}^p = 0 \tag{2.13.19}$$

Substituting (2.13.18), (2.13.16), (2.13.14) and (2.13.13) to (2.13.19), the $\dot{\lambda}$ can be expressed in terms of stress rate $\dot{\sigma}_{ij}$ and temperature rate $\dot{T}$ as:

$$\dot{\lambda} = L^{\sigma}_{ij}\dot{\sigma}_{ij} + L^T\dot{T} \tag{2.13.20}$$

where:

$$L^{\sigma}_{ij} = \frac{S_{ij}}{\frac{4}{9}H_p\sigma_y^2(\varepsilon^p)\Gamma^{\frac{3}{2}}(T)} \tag{2.13.21}$$

$$L^T = \frac{T'}{\frac{2}{3}H_p(T_{\text{melt}} - T_1)^2\Gamma^{\frac{3}{2}}(T)} \tag{2.13.22}$$

L^{σ}_{ij} and L^T are important in developing the nonlinear algorithm for BEM plasticity.

If substituting (2.13.18), (2.13.16), (2.13.14), (2.13.13) and (2.13.5) to (2.13.19), we can get the expression of $\dot{\lambda}$ in terms of mechanical strain rate $\dot{\varepsilon}^m_{kl}$ and temperature rate $\dot{T}$

$$\dot{\lambda} = \frac{2GS_{kl}\dot{\varepsilon}^m_{kl} + \frac{2}{3}\sigma_y^2(\varepsilon^p)\frac{T'}{(T_{\text{melt}}-T_{\text{peak}})^2}\dot{T}}{\frac{4}{9}G\sigma_y^2(\varepsilon^p)\Gamma(T)\left(3G + H_p\Gamma^{\frac{1}{2}}(T)\right)} \tag{2.13.23}$$

Taking (2.13.23) back to (2.13.5), we have the incremental stress-strain relationship of temperature-dependent Von Mises model:

$$\sigma_{ij} = D^{ep}_{ij\,kl}\dot{\varepsilon}^m_{kl} + D^T_{ij}\dot{T} \tag{2.13.24}$$

where

$$D^{ep}_{ij\,kl} = D^e_{ij\,kl} - \frac{9G^2 S_{ij}\,S_{kl}}{\sigma_y^2(\varepsilon^p)\Gamma(T)\left(3G + H_p\Gamma^{\frac{1}{2}}(T)\right)} \tag{2.13.25}$$

$$D^T_{ij} = -\frac{3GT'S_{ij}}{(T_{\text{melt}} - T_{\text{peak}})^2\Gamma(T)\left(3G + H_p\Gamma^{\frac{1}{2}}(T)\right)} \tag{2.13.26}$$

Equation (2.13.24) is the derivative form and state related. Therefore, some special integral schemes must be employed for finite values of $\dot{\varepsilon}^m_{kl}$ and $\dot{T}$. The Runge-Kutta integral method is selected in this work.

<u>Ordinary Von Mises Model</u>

For temperature-independent material, $T' = 0, \Gamma(T) = 1$, the above equations will reduce to the relationships of ordinary Von Mises model.

b. Modified Cam-clay Soil Model and Isotropic Plasticity Soil Model

The total stresses in soil can be decomposed into the effective stresses subjected by the soil skeleton and the pore pressure due to the fluid within the soil.

$$\sigma^{\text{total}}_{ij} = \sigma_{ij} - \delta_{ij}\,\beta p \tag{2.13.27}$$

where p is the pore pressure, conventionally defined positive for compression. β is a dimensionless material parameter relating to the compressibility of the soil skeleton and the fluid. For fully saturated soil, β may be equal to 1.

For the studying of plastic behavior of soil, Roscoe and Burland (1968) developed the well-known modified Cam-clay model. By using the incremental theory of plasticity and relating the hardening behavior to the consolidation test, the modified Cam-clay model successfully gives the incremental stress-strain relationship. The modified Cam-clay model employs the Drucker-Prager failure criterion. Since the Mohr-Coulomb failure criterion is more accurate than Drucker-Prager failure criterion for soil behavior, an isotropic plasticity model of soil based on Mohr-Coulomb criterion is developed in **GPBEST** . The isotropic plasticity model also includes the effect of cohesion of the soil.

2.13.b.1 Definition of Invariants

The principal stresses σ_1, σ_2 σ_3 are the solutions of the equation:

$$\sigma^3 - I\sigma^2 + I_2\sigma - I_3 = 0 \tag{2.13.28}$$

where:

$$I_1 = \sigma_{11} + \sigma_{22} + \sigma_{33}, I_2 = \begin{vmatrix} \sigma_{11} & \sigma_{12} \\ \sigma_{21} & \sigma_{22} \end{vmatrix} + \begin{vmatrix} \sigma_{22} & \sigma_{23} \\ \sigma_{32} & \sigma_{33} \end{vmatrix} + \begin{vmatrix} \sigma_{33} & \sigma_{31} \\ \sigma_{13} & \sigma_{11} \end{vmatrix} \ , \ I_3 = \begin{vmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{vmatrix} \tag{2.13.29}$$

By defining:

$$J_1 = \frac{1}{3} I_1 \ , \qquad S_{ij} = \sigma_{ij} - J_1 \tag{2.13.30}$$

the principle values of S_{ij} are the solutions of equation:

$$S^3 - J_2^2 S - J_3 = 0 \tag{2.13.31}$$

where:

$$J_2 = \sqrt{\frac{1}{2} S_{ij} S_{ij}} = \sqrt{\frac{1}{2}\left(\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - 3J_1^2\right)} \tag{2.13.32}$$

$$J_3 = \frac{1}{3} S_{ij} S_{jk} S_{ki} = (\sigma_1 - J_1)(\sigma_2 - J_1)(\sigma_3 - J_1) \tag{2.13.33}$$

In soil mechanics, the following invariants of stresses and invariants of strains are commonly used and are defined as:

$$p = -J_1, \ q = \sqrt{3} J_2, \ \theta = \frac{1}{3} \mathrm{a}\sin\left(\frac{3\sqrt{3}}{2} \frac{J_3}{J_2^3}\right) \tag{2.13.34}$$

$$\varepsilon_v = \varepsilon_1 + \varepsilon_2 + \varepsilon_3, \ \varepsilon_d = \frac{\sqrt{2}}{3}\left((\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2\right)^{\frac{1}{2}} \tag{2.13.35}$$

where θ is known as the Lode angle, and ε_1, ε_2, ε_3 are the principal strains. Although the definitions in this section are presented in total values, they are also valid in incremental form.

2.13.b.2 Critical State Line

Here, critical state means failure state. Mohr-Coulomb criterion was developed early in 1880s, but it is still the widely used failure criteria for soil. It says, under the conditon of:

$$\tau_{\max} = C + \sigma_n \tan\phi \tag{2.13.36}$$

the material fails. C and ϕ are the cohesion and friction angle of the material respectively. In the principal stress space $\sigma_1 - \sigma_2 - \sigma_3$, Mohr-Coulomb criterion is a cone composed of six flat surfaces. Figure 2.13.4 shows the intersection of the cone with the J_1 constant surface. In soil mechanics, $\theta = \pi/6$ and $\theta = -\pi/6$ are the two extreme states known as compression status and extension status. In comparison, Drucker-Prager criterion is also plotted in Figure 2.13.4. Since p and q are the commonly used invariants in soil mechanics, it is necessary to present the criteria in $p - q$ space also, as shown in Figure 2.13.5.

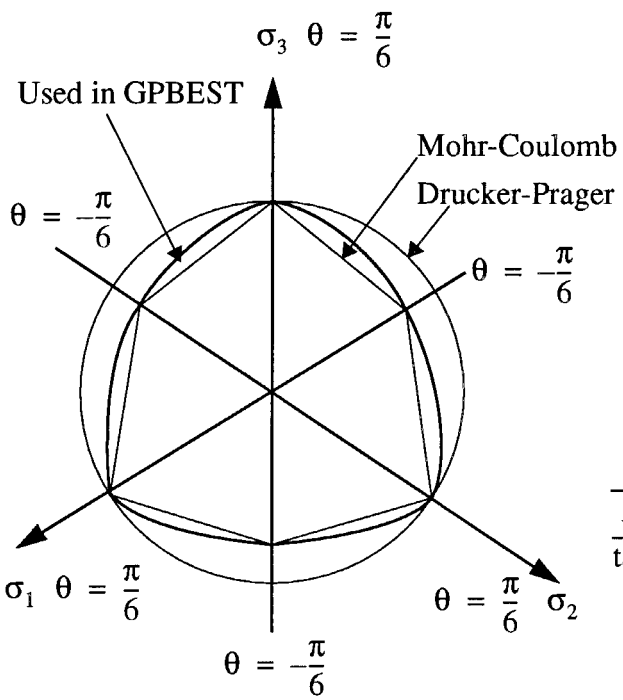


Figure 2.13.4 Failure Criteria in Principal Stress Space

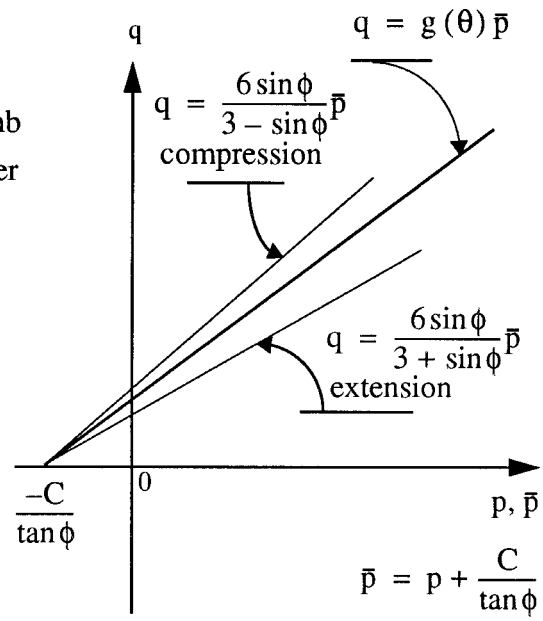


Figure 2.13.5 Critical State Lines in $p-q$ Space

Generally, the critical state line should fall between the compression and extension line:

$$q = g(\theta)\left(p + \frac{C}{\tan\phi}\right) = g(\theta)\overline{p} \tag{2.13.37}$$

where $g(\theta)$ is taken as:

$$g(\theta) = \frac{6\sin\phi}{3 - \sin 3\theta \sin\phi} \tag{2.13.38}$$

2.13.b.3 Formation of Isotropic Plasticity Soil Model

Yielding Criterion

The surface for soil is assumed as:

$$F(p, q, p_0) = \frac{q^2}{g^2(\theta)} + \overline{p}^2 - \overline{p}_0\overline{p} = 0 \tag{2.13.39}$$

where:

$$\overline{p} = p + \frac{C}{\tan\phi}, \quad \overline{p}_0 = p_0 + \frac{C}{\tan\phi} \tag{2.13.40}$$

and p_0 is considered the previous maximum pressure of the soil. For a given Lode angle, the yield surface is shown in Figure 2.13.6.

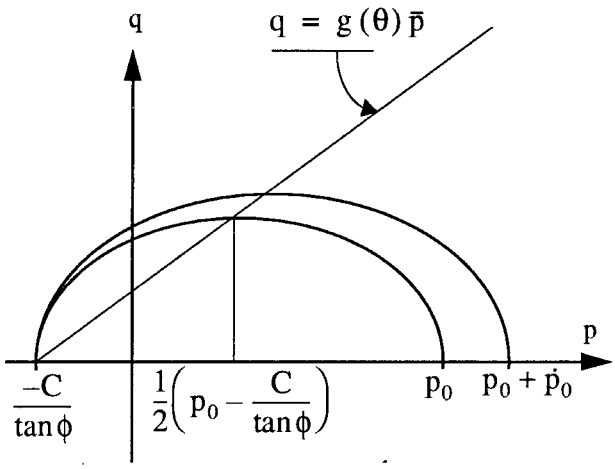


Figure 2.13.6 Yield Surfaces in $p-q$ Space

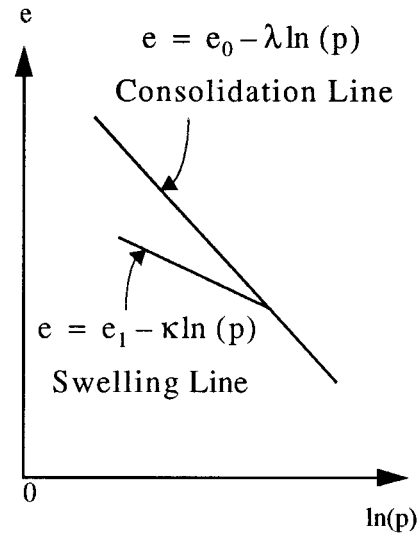


Figure 2.13.7 Consolidation Behavior of Soil

Considering the definition of invariants, we have:

$$\frac{\partial F}{\partial \sigma_{ij}} = \frac{3S_{ij}}{g^2(\theta)} + (2\overline{p} - \overline{p}_0)\frac{1}{3}\delta_{ij} \tag{2.13.41}$$

Plastic Flow Rate

Considering (2.13.41), the plastic flow rule for soil becomes:

$$\dot{\varepsilon}^p_{ij} = \dot{\lambda}\frac{\partial F}{\partial \sigma_{ij}} = \dot{\lambda}\left(\frac{3S_{ij}}{g^2(\theta)} + (2\overline{p} - \overline{p}_0)\frac{1}{3}\delta_{ij}\right) \tag{2.13.42}$$

Hardening Rule

The hardening behavior of soil is expressed through the increase of p_0. Under the plastic deformation, the yielding surface changes to a new position as shown in Figure 2.13.6. The value $\dot{p}_0$ is related to the plastic volume strain rate by:

$$\dot{p}_0 = \frac{1+e}{\lambda - \kappa} p_0 \dot{\varepsilon}^p_v \tag{2.13.43}$$

where λ and κ are the slopes of the $e - \ln(p)$ curves of the consolidation line and the swelling line of the soil as shown in Figure 2.13.7. e_0 and e are the initial and current void ratio, respectively. The plastic volume strain rate in (2.13.43) can be written as:

$$\dot{\varepsilon}^p_v = \dot{\varepsilon}^p_{ii} = \dot{\lambda}\frac{\partial F}{\partial \sigma_{ii}} = \dot{\lambda}(2\overline{p} - \overline{p}_0) \tag{2.13.44}$$

Incremental Stress-Strain Relationship of Soil

The procedures of derivation are similar to what is used in the temperature-dependent Von Mises model. Only the results are presented here. The tensors K_{ij} and L_{ij} which will be used in developing the non-linear algorithm are:

$$K_{ij} = D^{e}_{ij\,kl}\frac{\partial F}{\partial \sigma_{kl}} = \frac{6GS_{ij}}{g^2(\theta)} + K(2\overline{p} - \overline{p}_0)\delta_{ij}\,, \tag{2.13.45}$$

and:

$$L_{ij} = \frac{\frac{3S_{ij}}{g^2(\theta)} + (2\overline{p} - \overline{p}_0)\frac{1}{3}\delta_{ij}}{\overline{p}\overline{p}_0 H_p} \tag{2.13.46}$$

where K is the bulk modulus and:

$$H_p = \frac{1+e}{\lambda - \kappa}\left(\frac{\partial F}{\partial \sigma_{ii}}\right) = \frac{1+e}{\lambda - \kappa}(2\overline{p} - \overline{p}_0) \tag{2.13.47}$$

The $\dot{\lambda}$ can be expressed by the strain rate as:

$$\dot{\lambda} = \frac{K_{ij}}{K_{mn}\frac{\partial F}{\partial \sigma_{mn}} + \overline{p}\overline{p}_0 H_p}\dot{\varepsilon}_{ij} \tag{2.13.48}$$

The incremental stress-strain relationship is:

$$\dot{\sigma}_{ij} = D^{ep}_{ij\,kl}\dot{\varepsilon}_{kl} = \left(D^{e}_{ij\,kl} - \frac{K_{ij}K_{kl}}{K_{mn}\frac{\partial F}{\partial \sigma_{mn}} + \overline{p}\overline{p}_0 H_p}\right)\dot{\varepsilon}_{kl} \tag{2.13.49}$$

2.13.b.4 Modified Cam-clay Model

Modified Cam-clay model is a simplified case of the isotropic plasticity model, by letting $C = 0, g(\theta) = \frac{6\sin\phi}{3-\sin\phi}$. Lode angle is not used in modified Cam-clay model.

3.0 NUMERICAL IMPLEMENTATION

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 geometry and the variation of field quantities (e.g., displacement, traction, temperature, pressure) using a suitable set of computationally convenient simple functions.

2 - Evaluate the surface 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. Further details are available in the sources previously cited [2-13].

3.1 REPRESENTATION OF GEOMETRY AND FUNCTIONS

The method employed in **GPBEST** 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 modeled as the union of isoparametric patches. Over each patch the variation of each of the cartesian coordinates is approximated as

$$x_i = N^k(\eta) x_i^k \tag{3.1.1}$$

where the x_i^k 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 **GPBEST** the variation of (known and unknown) field variables is carried out over each surface patch using linear, quadratic, and quartic 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 each. Geometry is modeled using linear or quadratic interpolation functions. Four special modeling capabilities should be noted:

1 - Substructuring capability is provided. This allows a part to be modeled as an assembly of several geometric modeling 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, quadratic and quartic variation of field quantities may be mixed within a single GMR.
3 - Enclosing elements are provided for use in problems in which boundaries extend to infinity.
4 - The surface patches can be specialized for fracture mechanics analysis by an appropriate relocation of certain midside geometry nodes.

3.2 SURFACE INTEGRATION

With the exceptions of some integrals for elasticity and heat transfer, all surface integrations in **GPBEST** 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. 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 integrations in **GPBEST** 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 transient formulation has somewhat different requirements than in the quasi- static case. Reduced integration tolerances are required for transient problems. The surface integration procedures are outlined below.

Surface Discretization

Using isoparametric quadratic elements, the coordinates and the functions (displacements and tractions, for example) at any point over an element (2D or Axisymmetry) can be expressed in terms of their nodal values as:

$$x_i = N_\alpha(\eta)\, X_{i\alpha}$$
$$u_i = N_\alpha(\eta)\, U_{i\alpha}$$
$$t_i = N_\alpha(\eta)\, T_{i\alpha}$$

where

$$i = 1, 2$$
$$\alpha = 1, 2, 3$$

and $N_\alpha(\eta)$ are the spatial shape functions in the intrinsic coordinates (η) of the element.
For three-dimensional elements the corresponding expressions are

$$x_i = N_\alpha(\eta_1, \eta_2)\, X_{i\alpha}$$
$$u_i = N_\alpha(\eta_1, \eta_2)\, U_{i\alpha}$$
$$t_i = N_\alpha(\eta_1, \eta_2)\, T_{i\alpha}$$

where

$$i = 1, 2, 3$$
$$\alpha = 1, 2, 3, ..A,$$ A=no. of nodes in the element

and $N_\alpha(\eta_1, \eta_2)$ are the spatial shape functions in the intrinsic coordinates (η_1, η_2) of the element.

Numerical Integration

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

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

2. Depending on the proximity between the field point (ξ) and the element subdivision and additional mapping for improved accuracy.

3. Gaussian quadrature formulas are employed for the evaluation of the discretized integral over each element (or sub-element). These formulas approximate the integral as a sum of weighted function values at designated points. The error in the approximation is dependent on the order of the (Gauss) points employed in the formula. To minimize error while at the same time maintaining computational efficiency, optimization schemes are used to choose the best number of points for a particular field point and element [32].

4. When the field point coincides with a node of the element being integrated, the integration becomes singular. In this case, the value of the coefficients of the F_{ij} kernel corresponding to the singular node cannot be calculated accurately by numerical integration. Instead, after the integration of all elements is complete, this value is determined so as to satisfy a rigid body displacement of the body [2].

3.3 ASSEMBLY OF SYSTEM EQUATIONS

The intermediate result of the surface 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.

The equations for the entire system, after transferring the unknowns to the left and the knowns to the right, can be written in matrix form as

$$\mathbf{A}\,\mathbf{x} = \mathbf{B}\,\mathbf{y} + \mathbf{R} \tag{3.3.1}$$

and

$$\sigma = \mathbf{A}^{\sigma}\,\mathbf{x} + \mathbf{B}^{\sigma}\,\mathbf{y} + \mathbf{R}^{\sigma} \tag{3.3.2}$$

where:

x	= vector of unknown displacements and tractions
y	= vector of known displacements and tractions
R, R^{σ}	= 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 transient time-domain analysis R and R^{σ} represent the past 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, an advanced substructured multiregion assembly process is introduced.

In this improved assembly process it is observed that in a multizone model, the source points can be collected into two distinct sets. The first set includes all source points residing on a region-to-region interface. These nodes will be labeled with the subscript a, denoting analysis-set. Considering a scalar potential flow BEM analysis and let N_a^m represent the number of such nodes. The remaining source points are each associated with a single region, and consequently may be eliminated at the zonal level. The latter nodes are considered part of the omitted-set and identified by the subscript o. Typically, the o-set, which contains N_o^m nodes, is much larger than the a-set. Introducing this notation into the system equations produces

$$\begin{bmatrix} G_{aa}^m & G_{ao}^m \\ G_{oa}^m & G_{oo}^m \end{bmatrix} \begin{Bmatrix} q_a^m \\ q_o^m \end{Bmatrix} = \begin{bmatrix} F_{aa}^m & F_{ao}^m \\ F_{oa}^m & F_{oo}^m \end{bmatrix} \begin{Bmatrix} p_a^m \\ p_o^m \end{Bmatrix} \tag{3.3.3}$$

Next, boundary conditions can be applied at the zonal level. All of the quantities $\{q_a^m\}$ and $\{p_a^m\}$ are unknown and can be combined to form the vector $\{x_a^m\}$. However, the boundary conditions

must specify all but N_o^m of the quantities $\{q_o^m\}$ and $\{p_o^m\}$. The unknown components form $\{x_o^m\}$. Then, equation (3.3.3) becomes

$$\begin{bmatrix} A_{aa}^m & A_{ao}^m \\ A_{oa}^m & A_{oo}^m \end{bmatrix} \begin{Bmatrix} x_a^m \\ x_o^m \end{Bmatrix} = \begin{Bmatrix} b_a^m \\ b_o^m \end{Bmatrix} \tag{3.3.4}$$

for each region. The matrix A^m is not square. It is of size $(N_a^m + N_o^m) \times (2N_a^m + N_o^m)$. On the other hand, A_{oo}^m is a square matrix of order N_o^m. By applying gaussian elimination, equation (3.3.4) can be reduced to

$$[\bar{A}_{aa}^m]\{x_a^m\} = \{\bar{b}_a^m\}, \tag{3.3.5}$$

where symbolically

$$[\bar{A}_{aa}^m] = [A_{aa}^m] - [A_{ao}^m][A_{oo}^m]^{-1}[A_{oa}^m] \tag{3.3.6a}$$

$$[\bar{b}_a^m] = \{b_a^m\} - [A_{ao}^m][A_{oo}^m]^{-1}\{b_o^m\}. \tag{3.3.6b}$$

Once this is accomplished for each zone, then $[\bar{A}_{aa}^m]$ and $[\bar{b}_a^m]$ for the regions are assembled together. After the interface conditions are imposed the Residual equations can be written as

$$[\bar{A}_{aa}]\{x_a\} = \{\bar{b}_a\}. \tag{3.3.7}$$

However, by this time, since many equations have been eliminated, the resulting system matrices are much smaller. In fact the order of the square Residual $[\bar{A}_{aa}]$ matrix is only N_a, where for an M region problem

$$N_a = \sum_{m=1}^{M} N_a^m.$$

After (3.3.7) is solved for $\{x_a\}$, the remaining unknowns in the individual regions, $\{x_o^m\}$, can be easily determined via backsubstitution. Specifically,

$$\{x_o^m\} = [A_{oo}^m]^{-1}\{\bar{b}_o^m\} \tag{3.3.8}$$

where

$$\{\bar{b}_o^m\} = \{b_o^m\} - [A_{oa}^m]\{x_a^m\}. \tag{3.3.9}$$

With this procedure, often called ‘‘substructured solution’’, none of zero blocks present in the block-banded approach, ever enter into the formulation. This greatly reduces the disk storage requirements, particularly for large problems, and also eliminates unnecessary operations. In addition, by reordering operations during integration and assembly, further savings can be realized.

If all interfaces are assumed smooth, both the potential and the flux at each interface node are unique. In defining the size of A^m in equation (3.3.4) a similar assumption was made. More generally, there are N_a^m components in $\{p_a^m\}$, however $\{q_a^m\}$ contains a component for each node of each interface element. Only for the case in which a unique interfacial normal exists at each interface node will the number of components in $\{q_a^m\}$, say Q_a^m, reduce to N_a^m. Thus, in general, $Q_a^m \geq N_a^m$, and, of course, Q_a^m interface relationships are needed to eliminate the unknowns $\{q_a^m\}$.

This can be accomplished by writing a resistance relationship at each node of each interface element. For example, for a connection between region 1 and 2, the equations

$$q^1 = -\frac{1}{R}(p^2 - p^1) \tag{3.3.10a}$$

$$q^2 = -\frac{1}{R}(p^1 - p^2) \tag{3.3.10b}$$

can be utilized to eliminate q^1 and q^2 in terms of the nodal potentials. In the case of perfect contact, the resistance R is set to a suitably small value to ensure that $p^1 \cong p^2$ while avoiding ill-conditioning. This element-based approach permits the interconnection of any number of substructures at a node.

3.4 SOLUTION OF SYSTEM EQUATIONS

3.4.1 LINEAR ANALYSIS

The first step in the solution 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 re-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. The process is then 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.

A new solver was written for **GPBEST.** 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 behavior, is highly efficient.

3.4.2 NONLINEAR ANALYSIS

In the inelastic analysis, two algorithms have been implemented: An initial stress iteration algorithm and a Newton-Raphson algorithm. The two algorithms are presented as parallel procedures in order to make clear comparisons.

Initial Stress Iteration Algorithm

The final equations for static and transient BEM plasticity analysis are assembled to the following equations:

$$\mathbf{A}\dot{\mathbf{X}} = \dot{\mathbf{R}} + \mathbf{B}\dot{\boldsymbol{\sigma}} \qquad \text{Boundary equation,} \tag{3.4.1}$$

$$\dot{\boldsymbol{\sigma}} = -\mathbf{A}^{\boldsymbol{\sigma}}\dot{\mathbf{X}} + \dot{\mathbf{R}}^{\boldsymbol{\sigma}} + \mathbf{B}^{\boldsymbol{\sigma}}\dot{\boldsymbol{\sigma}}^{0} \qquad \text{Stress equation,} \tag{3.4.2}$$

$$\dot{\mathbf{U}} = -\mathbf{A}^{\mathbf{u}}\dot{\mathbf{X}} + \dot{\mathbf{R}}^{\mathbf{u}} + \mathbf{B}^{\mathbf{u}}\dot{\boldsymbol{\sigma}}^{0} \qquad \text{Displacement equation.} \tag{3.4.3}$$

The above equations are based on the concept of initial stresses introduced earlier. In the elastic region, the initial stresses are zero. But for materials in yielding, the initial stresses are not known *a priori*. Then, the solution of the above equations for plasticity has to be sought by iteration.

Taking $\dot{\mathbf{X}}$ as an example, the standard linearization of variables in iteration can be written as:

$$\dot{\mathbf{X}}_{n+1} = \dot{\mathbf{X}}_n + \Delta\dot{\mathbf{X}}_n, \qquad n = 1, 2, 3, \ldots \tag{3.4.4}$$

Where, the superior dot represents the loading step, Δ means the increment of iteration within a loading step, and subscript n indicates the nth iteration.

The first procedure of the initial stress iterative method is to obtain an approximate solution by assuming the material's remaining elastic during one iteration. Then, by integrating the constitutive relationship of plasticity, the correction is made and iterations are established. Referring to Figure 3.4.1, the details of this method are discussed below.

In Figure 3.4.1, position 1 defines the initial yielding status for which the highly stressed node is at yield. Position 1 can be easily found by the elastic solution. Starting from point 1, a loading step $\dot{R}_1$ is applied. Supposing position Q is the real solution to the current loading step, our goal is to find a convergent position which is close to position Q within a given tolerance. Figure 3.4.1 only plots the first iteration of the loading step, since the subsequent iterations are merely repetitions. The initial stress iteration method can be presented by the following procedures.

1. Setting the Initial Increments

The initial increments are all set to zero at the first iteration of the current loading step:

$$\dot{\mathbf{X}}_1 = 0,\ \dot{\boldsymbol{\sigma}}_1 = 0,\ \dot{\mathbf{U}}_1 = 0,\ \dot{\boldsymbol{\varepsilon}}_1^{m} = 0,\ \dot{\boldsymbol{\sigma}}_1^{0} = 0 \tag{3.4.5}$$

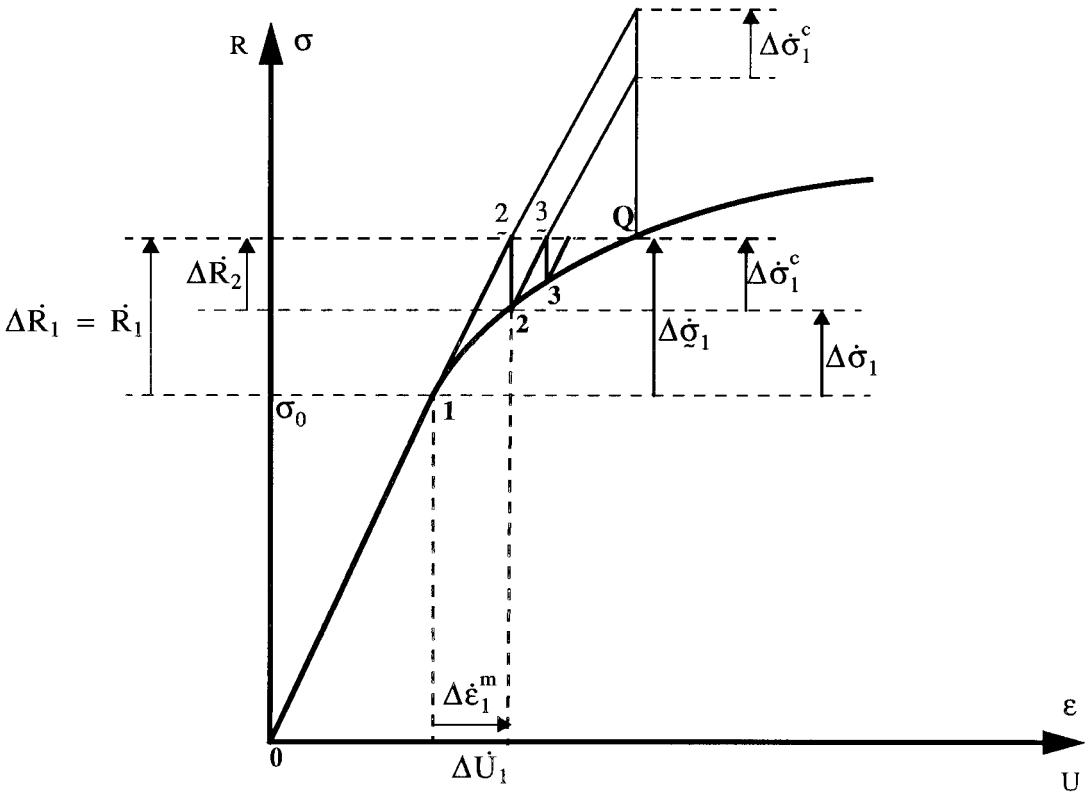


Figure 3.4.1 Illustration of the Initial Stress Iteration Method for BEM Plasticity

2. Approximation by Elastic Solution

For the current loading step, position $\underset{\sim}{2}$ is obtained by assuming that the materials still remain elastic. This assumption is equivalent to allowing the increment of initial stresses from position 1 to position $\underset{\sim}{2}$ be zero:

$$\Delta\dot{\underset{\sim}{\sigma}} = 0 \tag{3.4.6}$$

and other increments at position $\underset{\sim}{2}$ are obtained by solving equations (3.4.1) through (3.4.3):

$$\begin{aligned} \Delta\dot{\mathbf{X}}_1 &= \mathbf{A}^{-1}\dot{\mathbf{R}}_s \\ \Delta\dot{\underset{\sim}{\sigma}}_1 &= -\mathbf{A}^{\sigma}\Delta\dot{\mathbf{X}}_1 + \dot{\mathbf{R}}_1^{\sigma} \\ \Delta\dot{\mathbf{U}}_1 &= -\mathbf{A}^{u}\Delta\dot{\mathbf{X}}_1 + \dot{\mathbf{R}}_1^{u} \end{aligned} \tag{3.4.7}$$

Since the materials are assumed to remain elastic from position 1 to position $\underset{\sim}{2}$, the increment of mechanical strains from position 1 to $\underset{\sim}{2}$ is given through the elastic constitutive relationship:

$$\Delta\dot{\varepsilon}_1^m = (\mathbf{D}^e)^{-1}\Delta\dot{\underset{\sim}{\sigma}}_1$$

3. Runge-Kutta Integration of the Constitutive Tensors $\mathbf{D}^{ep}$ and $\mathbf{D}^{T}$

It is obvious that position $\underset{\sim}{2}$ is not on the stress-strain curve of plasticity. The stresses related to $\Delta\dot{\varepsilon}_1^m$ should be $\Delta\dot{\sigma}_1$ instead of the approximated $\Delta\dot{\underset{\sim}{\sigma}}_1$. And $\Delta\dot{\sigma}_1$ is obtained by the integration of constitutive relationship. Thermoplasticity is illustrated here:

$$\Delta\dot{\sigma}_1 = \int_0^{\Delta\dot{\varepsilon}_1} \mathbf{D}^{ep} d\dot{\varepsilon}_1^m + \int_0^{\Delta\dot{T}_1} \mathbf{D}^T d\dot{T}_1 \tag{3.4.9}$$

Where the increment of temperature $\Delta\dot{T}_1$ is included in $\Delta\dot{\mathbf{U}}_1$. Since $\mathbf{D}^{ep}$ and $\mathbf{D}^T$ are also dependent on stresses, (i.e., state related), special integral schemes must be employed for (3.4.9). The classical *fourth-order Runge-Kutta formula* [Abramowitz and Stegun (1964)] is selected.

After obtaining $\Delta\dot{\sigma}_1$, the difference between $\Delta\dot{\underset{\sim}{\sigma}}_1$ and $\Delta\dot{\sigma}_1$ is defined as:

$$\Delta\dot{\sigma}_1^c = \Delta\dot{\underset{\sim}{\sigma}}_1 - \Delta\dot{\sigma}_1 \tag{3.4.10}$$

$\Delta\dot{\sigma}_1^c$ is the plastic correction of stresses for the elastic assumption. Then, referring to Figure 3.4.1, the increment of initial stresses of the first iteration is:

$$\Delta\dot{\sigma}_1^0 = \Delta\underset{\sim}{\sigma}_1^0 + \Delta\dot{\sigma}_1^c = \Delta\dot{\sigma}_1^c \tag{3.5.11}$$

4. Summation of Variables

Position 2, which is on the stress-strain curve of plasticity, is obtained by the summation update as:

$$\begin{aligned}
\dot{X}_2 &= \dot{X}_1 + \Delta\dot{X}_1 \\
\dot{\sigma}_2 &= \dot{\sigma}_1 + \Delta\dot{\sigma}_1 \\
\dot{U}_2 &= \dot{U}_1 + \Delta\dot{U}_1 \\
\dot{\varepsilon}_2^m &= \dot{\varepsilon}_1^m + \Delta\dot{\varepsilon}_1^m \\
\dot{\sigma}_2^0 &= \dot{\sigma}_1^0 + \Delta\sigma_1^0
\end{aligned} \tag{3.4.12}$$

5. The Right-hand Side (RHS) Terms for the Next Iteration

Generally, position 2 is not the plastic solution desired for the current load step. Let $\dot{R}_2$, $\dot{R}_2^\sigma$ represent the unbalanced RHS terms between position 2 and the real solution at position Q. The following relations must hold:

$$\begin{cases}
A\Delta\dot{X}_1 = (\dot{R}_1 - \dot{R}_2) + B\Delta\dot{\sigma}_1^0 \\
\Delta\dot{\sigma}_1 = -A^\sigma\Delta\dot{X}_1 + \left(\dot{R}_1^\sigma - \dot{R}_2^\sigma\right) + B^\sigma\Delta\dot{\sigma}_1^0 \\
\Delta\dot{U}_1 = -A^u\Delta\dot{X}_1 + \left(\dot{R}_1^u - \dot{R}_2^u\right) + B^u\Delta\dot{\sigma}_1^0
\end{cases} \tag{3.4.13}$$

Combining (3.4.13) with the solutions (3.4.7), and noticing relations (3.4.10) and (3.4.11), we get the unbalanced RHS terms which are also the RHS terms for the next iteration.

$$\dot{R}_2 = B\Delta\dot{\sigma}_1^c$$

$$\dot{R}_2^{\sigma} = \Delta\dot{\sigma}_1^c + B^{\sigma}\Delta\dot{\sigma}_1^c \tag{3.4.14}$$

$$\dot{R}_2^{u} = B^{u}\Delta\dot{\sigma}_1^c$$

While only the formulae of the first iteration have been presented, they are valid for the subsequent iterations by simply changing the subscript of the variables.

6. Iteration and Convergence Checking

If the convergent solution has not been reached, the next iteration will be established by taking the RHS terms obtained in (3.4.14) back to procedure (2) and repeating procedures (2) to (6). Generally, convergence is attained when the norm of $\Delta\dot{\sigma}_n^c$ is less than a given tolerance (normally tolerance = 0.005*yield strength). In other words, the iteration of the current loading step will terminate if $\|\Delta\dot{\sigma}_n^c\| <$ tolerance.

7. Adding the Unbalanced RHS Terms to Next Loading Step

After obtaining the convergent solution for the current loading step, the unbalanced RHS terms obtained by (3.4.14) are small, but may not be zero. They must be added to the corresponding RHS terms of the next loading step, in order to eliminate the deviation of the solution and prevent buildup of equilibrium errors.

Newton-Raphson Algorithm for BEM Plasticity

The derivation of the Newton-Raphson algorithm begins with the procedure of incorporating the $\dot{\lambda}$ to the system equations. It would be better to rewrite equation (2.10.20) in matrix form as:

$$\dot{\lambda} = \mathbf{L}^{\sigma}\dot{\boldsymbol{\sigma}} + \mathbf{L}^{T}\dot{\mathbf{T}} = (\mathbf{L}^{\sigma}\mathbf{L}^{T})\begin{Bmatrix}\dot{\sigma}\\ \dot{T}\end{Bmatrix} \tag{3.4.15}$$

where matrix $\mathbf{L}^{\sigma}$ and $\mathbf{L}^{T}$ were given in (2.10.21) and (2.10.22), respectively.

By the definition of initial stresses and the plastic flow rule, the initial stresses can be expressed as:

$$\dot{\boldsymbol{\sigma}}^0 = \mathbf{K}\dot{\boldsymbol{\lambda}} \tag{3.4.16}$$

where:

$$K_{ij} = D^{e}_{ijkl}\frac{\partial F}{\partial \sigma_{kl}} \tag{3.4.17}$$

For thermoplasticity, the temperature is treated as an additional degree of freedom and is included in displacement equation (3.4.3). If the temperature equation is written separately, it has the form of:

$$\dot{\mathbf{T}} = -\mathbf{A}^{T}\dot{\mathbf{X}} + \dot{\mathbf{R}}^{T} + \mathbf{B}^{T}\dot{\boldsymbol{\sigma}}^0 \quad \ldots \text{ Temperature equation} \tag{3.4.18}$$

where, $\mathbf{A}^T$, $\dot{\mathbf{R}}^T$ and $\mathbf{B}^T$ are the corresponding terms for temperature in $\mathbf{A}^u$, $\dot{\mathbf{R}}^u$ and $\mathbf{B}^u$. Since no temperature variations caused by deformation are considered in this work, matrix B^T should be zero:

$$\mathbf{B}^{T} = \mathbf{0} \tag{3.4.19}$$

Multiplying (3.4.2) and (3.4.18) by $\mathbf{L}^{\sigma}$, $\mathbf{L}^{T}$ respectively, and adding them together, we have:

$$(\mathbf{L}^{\sigma}\mathbf{L}^{T})\left\{\begin{array}{c}\dot{\boldsymbol{\sigma}}\\ \dot{\mathbf{T}}\end{array}\right\} = (\mathbf{L}^{\sigma}\mathbf{L}^{T})\left\{\begin{array}{c}-\mathbf{A}^{\sigma}\\ -\mathbf{A}^{T}\end{array}\right\}\dot{\mathbf{X}} + (\mathbf{L}^{\sigma}\mathbf{L}^{T})\left\{\begin{array}{c}\dot{\mathbf{R}}^{\sigma}\\ \dot{\mathbf{R}}^{T}\end{array}\right\} + (\mathbf{L}^{\sigma}\mathbf{L}^{T})\left\{\begin{array}{c}\mathbf{B}^{\sigma}\\ \mathbf{0}\end{array}\right\}\dot{\boldsymbol{\sigma}}^{0} \tag{3.4.20}$$

Substituting (3.4.17) into (3.4.1) and (3.4.20), and considering (3.4.15), we arrive at the system equations of the Newton-Raphson algorithm:

$$\left[\begin{array}{cc}\mathbf{A} & -\mathbf{C}\\ \mathbf{A}^{\lambda} & -\mathbf{C}^{\lambda}\end{array}\right]\left\{\begin{array}{c}\dot{\mathbf{X}}\\ \dot{\boldsymbol{\lambda}}\end{array}\right\} = \left\{\begin{array}{c}\dot{\mathbf{R}}\\ \dot{\mathbf{R}}^{\lambda}\end{array}\right\} \tag{3.4.21}$$

where:

$$\begin{aligned}\mathbf{C} &= \mathbf{B}\mathbf{K}\\ \mathbf{A}^{\lambda} &= \mathbf{L}^{\sigma}\mathbf{A}^{\sigma} + \mathbf{L}^{T}\mathbf{A}^{T}\\ \mathbf{C}^{\lambda} &= \mathbf{L}^{\sigma}\mathbf{B}^{\sigma}\mathbf{K} - \mathbf{I}\\ \dot{\mathbf{R}}^{\lambda} &+ \mathbf{L}^{\sigma}\dot{\mathbf{R}}^{\sigma} + \mathbf{L}^{T}\dot{\mathbf{R}}^{T}\end{aligned} \tag{3.4.22}$$

Compared to the initial stress iteration method, $\dot{\lambda}$ is included in the system equations as an unknown vector. The advantage is that the convergence of (3.4.21) is much faster than (3.4.1). There are two features should also be noted. First, since $\dot{\lambda}$ is already known as zero for unyielded nodes, the size of $\dot{\lambda}$ included in (3.4.21) is the number of yielded nodes. Second, as matrix $\mathbf{A}$ in (3.4.21) is the same in elastic equations, Gaussian eliminations of matrix $\mathbf{A}$ are needed only once to solve Equations (3.4.21).

In order to compare with the initial stress iteration algorithm, Newton-Raphson algorithm is also expressed through seven procedures. Some of the procedures are the same for both methods and are not duplicated here. The Newton-Raphson algorithm is illustrated in Figure 3.4.2. Position 1 is a known balanced stage, such as the solution of the last loading step. Loading step $\dot{R}_1$ is applied, and position Q is the real solution for it. The seven stages of the Newton-Raphson method are presented as follows.

1. Setting the Initial Values

(Same as initial stress iteration method.)

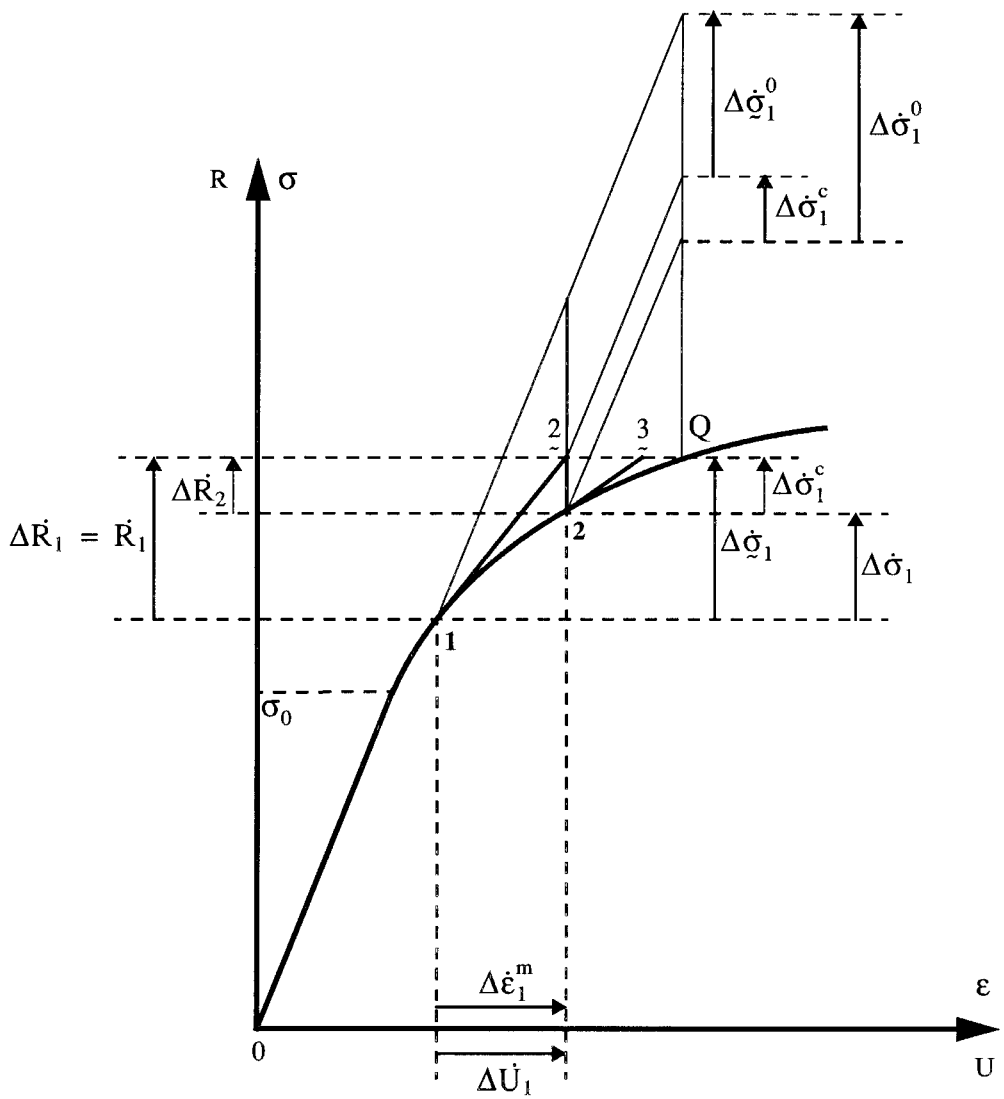


Figure 3.4.2 Illustration of the Newton-Raphson Algorithm for BEM Plasticity

2. Approximation by Tangential Slope

(Different to initial stress iteration method.)

Starting from position 1 in Figure 3.4.2, position $\underset{\sim}{2}$ is obtained based on the assumption that the matrices $\mathbf{K}$, $\mathbf{L}^{\sigma}$ and $\mathbf{L}^{T}$ are constant from position 1 to position $\underset{\sim}{2}$. This assumption is much weaker than the assumption enforced in the initial stress iteration method. Graphically, position $\underset{\sim}{2}$ is obtained by the tangential slope at position 1. Let $\mathbf{A}_1^{\lambda}$, $\mathbf{C}_1$, $\mathbf{C}_1^{\lambda}$, $\mathbf{K}_1$ represent the corresponding matrices at position 1. Position $\underset{\sim}{2}$ is obtained by solving:

$$\begin{bmatrix} \mathbf{A} & -\mathbf{C}_1 \\ \mathbf{A}_1^{\lambda} & -\mathbf{C}_1^{\lambda} \end{bmatrix} \begin{Bmatrix} \Delta\dot{\mathbf{X}}_1 \\ \Delta\dot{\lambda}_1 \end{Bmatrix} = \begin{Bmatrix} \dot{\mathbf{R}}_1 \\ \dot{\mathbf{R}}_1^{\lambda} \end{Bmatrix} \tag{3.4.23}$$

After obtaining $\Delta\dot{\mathbf{X}}_1$ and $\Delta\dot{\lambda}_1$, the following relations hold for the state variables at position $\underset{\sim}{2}$

$$\Delta\underset{\sim 1}{\dot{\sigma}^0} = \mathbf{K}_1\Delta\dot{\lambda}_1 \tag{3.4.24}$$

$$\begin{cases} -\mathbf{A}\Delta\dot{\mathbf{X}}_1 + \dot{\mathbf{R}}_1 + \mathbf{B}\Delta\underset{\sim 1}{\dot{\sigma}^0} = 0 \\ \Delta\underset{\sim 1}{\dot{\sigma}} = -\mathbf{A}^\sigma\Delta\dot{\mathbf{X}}_1 + \dot{\mathbf{R}}_1^\sigma + \mathbf{B}^\sigma\Delta\underset{\sim 1}{\dot{\sigma}^0} \\ \Delta\dot{\mathbf{U}}_1 = -\mathbf{A}^u\Delta\dot{\mathbf{X}}_1 + \dot{\mathbf{R}}_1^u + \mathbf{B}^u\Delta\underset{\sim 1}{\dot{\sigma}^0} \end{cases} \tag{3.4.25}$$

According to the definition of initial stresses, the increment of mechanical strains from position 1 to position $\underset{\sim}{2}$ is given by:

$$\Delta\dot{\varepsilon}_1^m = (\mathbf{D}^e)^{-1}\left(\Delta\underset{\sim}{\dot{\sigma}} + \Delta\underset{\sim 1}{\dot{\sigma}^0}\right). \tag{3.4.26}$$

3. Runge-Kutta Integration of the Constitutive Tensors $\mathbf{D}^{ep}$ and $\mathbf{D}^T$

(Similar to initial stress iteration method.)

The stresses at corrected position 2 are obtained by the *Runge-Kutta* integration of the constitutive relationship of plasticity:

$$\Delta\dot{\sigma}_1 = \int_0^{\Delta\dot{\varepsilon}_1}\mathbf{D}^{ep}d\dot{\varepsilon}_1^m + \int_0^{\Delta\dot{T}_1}\mathbf{D}^T d\dot{T}_1. \tag{3.4.27}$$

The difference between $\Delta\underset{\sim 1}{\dot{\sigma}}$ and $\Delta\dot{\sigma}_1$ is also defined as:

$$\Delta\dot{\sigma}_1^c = \Delta\underset{\sim 1}{\dot{\sigma}} - \Delta\dot{\sigma}_1. \tag{3.4.28}$$

Then, the corrected increment of initial stresses from position 1 to position 2 is:

$$\Delta\dot{\sigma}_1^0 = \Delta\underset{\sim 1}{\dot{\sigma}^0} + \Delta\dot{\sigma}_1^c. \tag{3.4.29}$$

4. Summation of Variables

(Same as initial stress iteration method.)

5. The Right-hand Side Terms for Next Iteration

(Similar to initial stress iteration method.)

Let $\dot{\mathbf{R}}_2$, $\dot{\mathbf{R}}_2^\sigma$ and $\dot{\mathbf{R}}_2^u$ represent the unbalanced RHS terms between position 2 and the real solution, position Q. The following relations must hold:

$$\begin{cases} \mathbf{A}\delta\dot{\mathbf{X}}_1 = (\dot{\mathbf{R}}_1 - \dot{\mathbf{R}}_2) + \mathbf{B}\Delta\dot{\sigma}_1^0 \\ \Delta\dot{\sigma}_1 = -\mathbf{A}^\sigma\Delta\dot{\mathbf{X}}_1 + (\dot{\mathbf{R}}_1^\sigma - \dot{\mathbf{R}}_2^\sigma) + \mathbf{B}^\sigma\Delta\dot{\sigma}_1^0 \\ \Delta\dot{\mathbf{U}}_1 = \mathbf{A}^u\Delta\dot{\mathbf{X}}_1 + (\dot{\mathbf{R}}_1^u - \dot{\mathbf{R}}_2^u) + \mathbf{B}^u\Delta\dot{\sigma}_1^0 \end{cases}. \tag{3.4.30}$$

Combining (3.4.30) with (3.4.25) and noting (3.4.28) and (3.4.29), we get the unbalanced RHS terms which are also the RHS terms for the next iteration.

$$\dot{\mathbf{R}}_2 = \mathbf{B}\Delta\dot{\sigma}_1^c$$

$$\dot{\mathbf{R}}_2^{\sigma} = \Delta\dot{\sigma}_1^c + \mathbf{B}^{\sigma}\Delta\dot{\sigma}_1^c \tag{3.4.31}$$

$$\dot{\mathbf{R}}_2^{u} = \mathbf{B}^{u}\Delta\dot{\sigma}_1^c$$

6. Iteration and Convergence Checking

(Same as initial stress iteration method.)

7. Adding the Unbalanced RHS Terms to Next Loading Step

(Same as initial stress iteration method.)

3.5 ACCURACY AND EFFICIENCY

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 (discontinuous) or partially discontinuous 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 comparing the relative efficiency and accuracy of conforming and nonconforming elements shows that conforming elements are significantly more accurate for the same number of quadratic elements. Additionally, computing costs are much higher for nonconforming elements [30].

Generally speaking, a BEM analysis is much more efficient when a single homogeneous region is sub-divided into several Geometric Modeling Regions. This is more efficient, not only from the point of view of integration but also for the final solution of the system equations. The cost of a single-region BEM analysis increases proportionally to N^3 where N is the total number of degrees of freedom for that region. On the other hand, the cost of a multi-region analysis is proportional to $\sum N_i^3$ (with $N = \sum N_i$), where N_i is the total number of degrees of freedom for each region.

It is therefore clearly advantageous to analyze a large problem by artificially subdividing the body into several regions, even though a natural partitioning may not exist. Since BEM solutions for such problems are also more accurate, there are compelling arguments for such multi-region analyses for a homogeneous body. In **GPBEST** , a limit on the total degrees of freedom for a single region as well as on the residual interface degrees of freedom have been imposed, such that a user is encouraged to use multi-region idealization for large problems. (see page 5.06)

For example, a single-region BEM analysis involving 12000 degrees of freedom takes approximately 30 hours on a typical workstation. In comparison, the same problem analyzed using an eight-region model, resulting in 16000 total degrees of freedom but with only 4000 interface degrees of freedom takes only about 8 hours with the same level of accuracy.

4.0 GUIDE TO USING GPBEST

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

4.1 GETTING STARTED

The following steps may allow a new user to get familiar with **GPBEST** :

I. 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.

II. 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.

III. A number of verification problems are supplied with **GPBEST** code in the 'examples' directory. The naming convention has been discussed in Section 1.4. Some of these examples are explained in more detail in Section 6 along with the entire listing of the input data and sample output. Consult these two sections for more information on the example problems.

A convenient method to locate a particular keyword or feature covered in problem data is to use the 'grep' command within the UNIX environment. For instance, in order to search all the verification problems for axisymmetric analyses that may exist in a directory, the following command may be executed at the system prompt:

```
prompt > grep -n  'AXIS'  *.DAT
```

or

```
prompt > grep -n  'AXIS'  *.DAT > filename
```

Note that while ''axisymmetry'' or ''plane'' problems can be identified this way, for three-dimensional problems there exist no such cards since the default analysis is a three-dimensional one (see Section 5.1). However, the user can also study the EX.summary file in examples directory to find features covered in the examples.

IV. It is possible to view any **GPBEST** data set (such as the ones in the example directories) by first creating a **GPBEST** Neutral file and then using the interface post processors such as BESTPAT and BESTIDEAS. This can be accomplished by including first, the NEUTral-file card in **CASE input (if it is not there already) and then either completing a full execution of **GPBEST** or simply running a data CHECk-data only (see the CHECk-data card in the **CASE input section).

V. The user may finally want to build his own models using PATRAN or IDEAS. This can be accomplished very easily if the user first creates three and four-sided, fully compatible patches to describe the correct geometrical details of the surface of the model. If the geometry is extremely complicated he may be well advised to sub-divide the model into several smaller (like substructuring in a FEM analysis) components. Each of these substructured components are then completely covered with three and four-sided, fully compatible patches. One must ensure however, that the common surfaces where the components join up have identical patches. In order to convert the patches so developed into elements, the user now can invoke the appropriate commands available in PATRAN and IDEAS to break up all patches into 1×1, 2×2, 3×3 (in 3D problems, for example) element patterns.
For an efficient three-dimensional BEM analysis it is also desirable to restrict the nodes per region to about 1500 to 2000 and the total number of interface nodes to less than 2000.

VI. Armed with the information outlined above the user should devote some time to read the Section 5.0 of this manual first and then the rest of this manual.

4.2 GRAPHICS INTERFACE

A graphics interface is essential to generate data for a realistic practical example and the subsequent processing of the results. A schematic representation of the various interfaces available currently and under development with **GPBEST** is shown in Fig. 4.1.

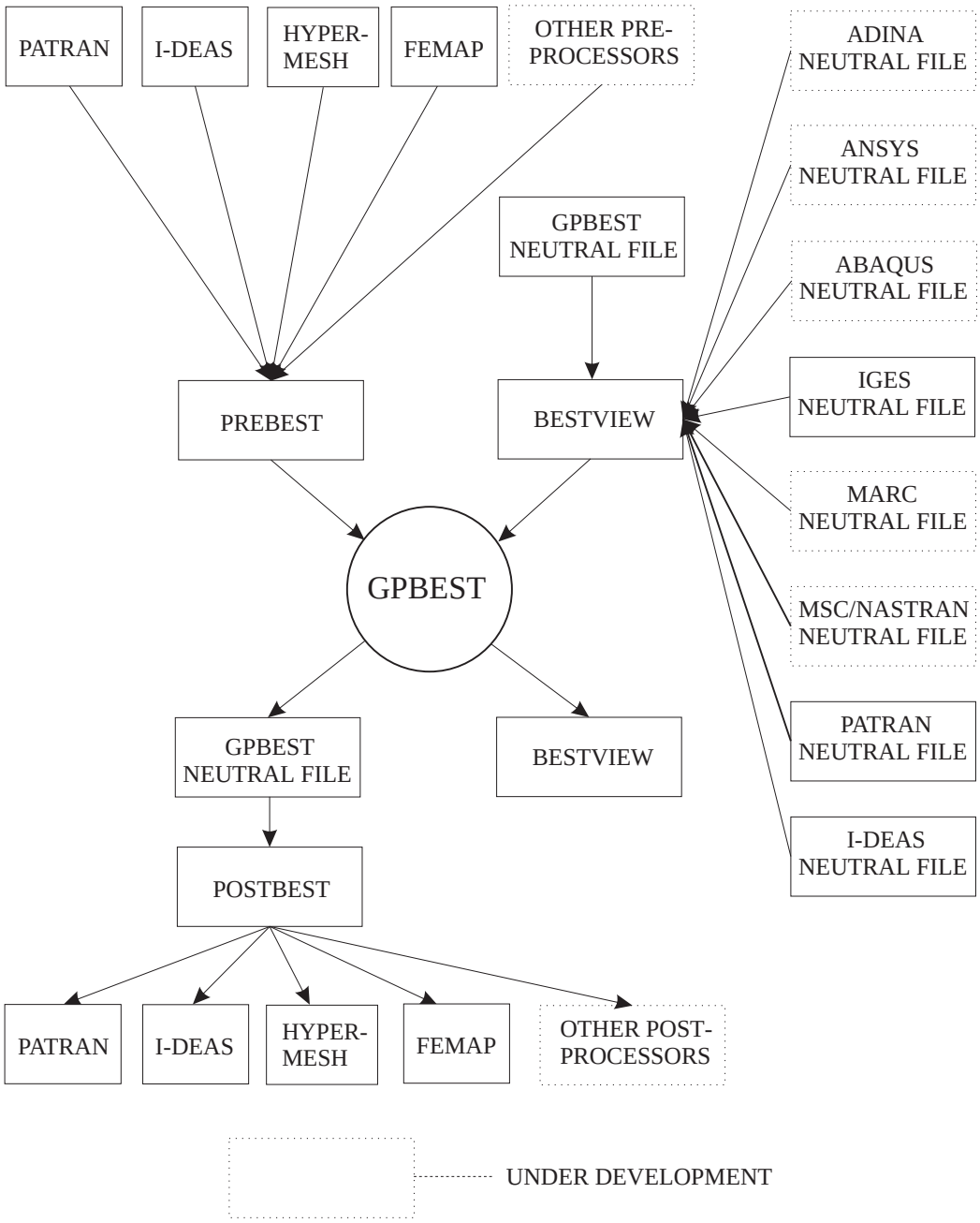


Fig. 4.1 Graphics Interfaces with GPBEST

BESTVIEW is an integrated graphics system for **GPBEST** which allows both pre- and post-processing of data for **GPBEST** as an integrated interface or in a stand-alone mode. The user interaction takes place through a simple X-Windows graphics interface. Additionally, **BESTVIEW** also contains translators to and from **IGES** neutral files and therefore, allows for the interfacing and integration of any other program that creates **IGES** neutral files with **GPBEST.** Translators are also being developed for popular finite element programs such as **ABAQUS, MARC, MSC/NASTRAN, ANSYS and ADINA.** Chapter 9 briefly describes the capabilities and features of **BESTVIEW.** For more information on **BESTVIEW,** please refer to the separate **BESTVIEW** User's Manual. New users of **BESTVIEW** can effectively start with Section 1.6 ''Quick Start Section'' and Chapter 8 ''A Tutorial on BESTVIEW.''

A **GPBEST** neutral file is designed as a general interface for post-processing of **GPBEST** results. Included in the neutral file are material and geometrical properties of the model, the interface and boundary conditions, all body force definitions and the results at each time step. Fortran source code of a standard translator is available to the user so that other graphics packages can easily be integrated with **GPBEST.**

GPBEST also interfaces with **PATRAN** and **I-DEAS**^{TM} for graphics applications. **PATRAN** is a general-purpose graphics input and output system which allows the user to interactively prepare input data for the surface and volume discretizations. It essentially generates the nodal coordinates and connectivities of a given discretization scheme. After the analysis, it allows the user to display the results in a graphics-oriented mode. **PATRAN** was developed and maintained by **PDA**^{TM} Engineering of California. Currently, **PATRAN** is being marketed by **MSC,** the owners of **MSC/NASTRAN,** a very popular finite element analysis program. **I-DEAS** is a CAD package for Mechanical Computer Aided Engineering and is developed and maintained by Structural Dynamics Research Corporation of Ohio.

PREBEST, which is the data preparation translator for **GPBEST,** takes the output from the **PATRAN** neutral file or **I-DEAS** universal file and translates the nodal coordinates and connectivities of the model generated in **PATRAN** or **I-DEAS** into a format of nodal coordinates and connectivities consistent with **GPBEST** input data.

Post-processing of results is possible through the reverse translator, **POSTBEST.** This translator converts the geometry data and results stored in the **GPBEST** neutral file to a **PATRAN** neutral file or an **I-DEAS** universal file. This allows users to display results in the environment of the CAD program they are using.

For more information on the **GPBEST** translators, please refer to Chapter 9 of the **BESTVIEW** manual.

4.3 AVAILABILTY OF GPBEST

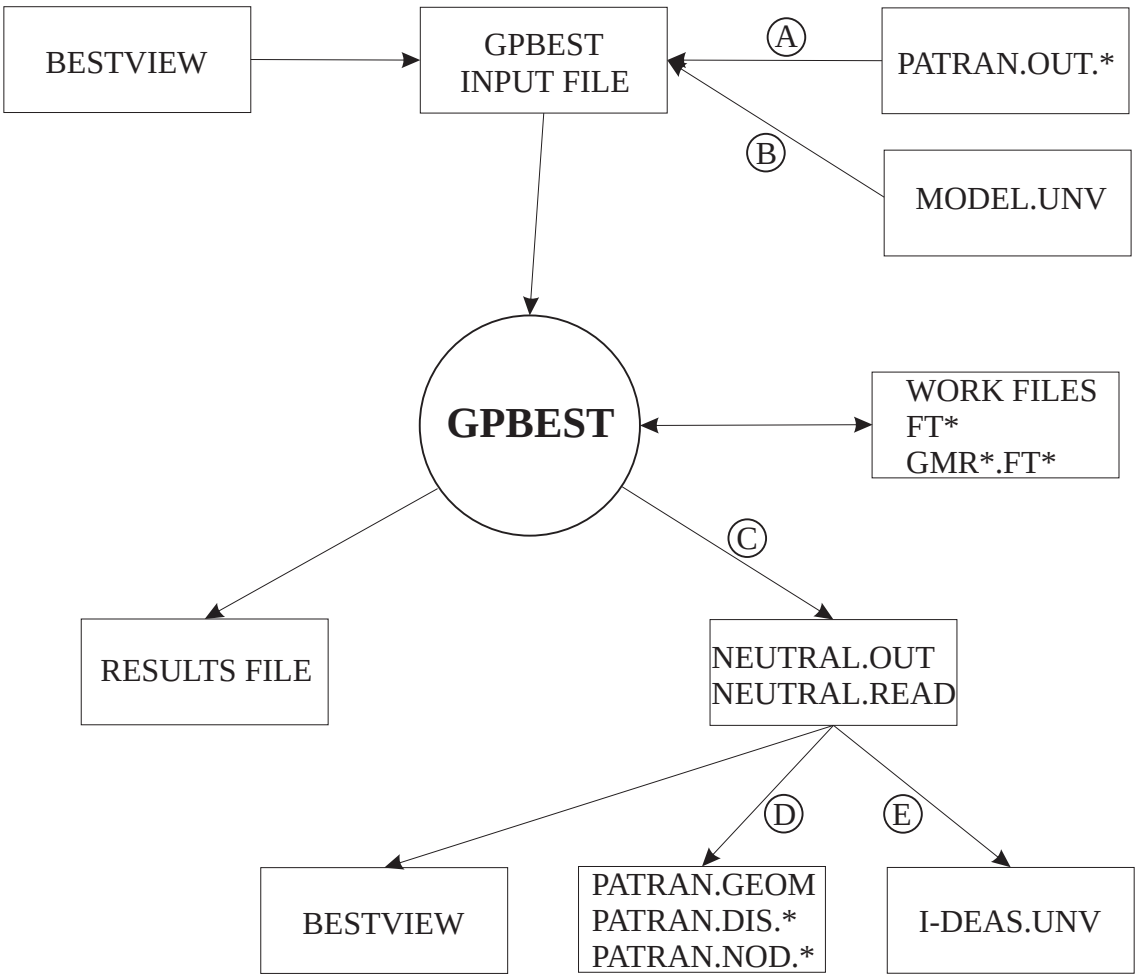
GPBEST is written in FORTRAN 77 and is therefore adaptable to any computer which has such a compiler. An executable version of the code is available on IBM^{TM} and CRAY^{TM} mainframes, HP 9000^{TM} (Series 700 and 800), SUN-4^{TM}, SUN Sparcstation^{TM} systems, IBM RISC System/6000^{TM}, and Silicon Graphics IRIS^{TM}. Depending upon the demand, it is intended to add all other systems to this list.

4.4 FILE SYSTEM IN GPBEST

GPBEST makes use of unit 5 as its input data file and unit 6 as its output (or result) 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 **GPBEST** 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 disk files is made. The files are either of sequential or direct access type and are defined as FT*** based on IBM terminology. The general layout of the file structure in **GPBEST** is shown in Fig. 4.2.

For the efficient execution of **GPBEST,** it is desirable to have at least 32 megabytes of system memory. Additionally, **GPBEST** makes extensive use of disk files during execution of the code. While most of these files are of temporary nature, some are required for restart analyses. In any case, it is recommended that workstation-based users have at least 300 MB of disk space free in order to run practical three-dimensional problems.

BESTVIEW is an integrated graphics program suitable for pre- and post-processing with **GPBEST.** Pre-processing is also possible using other graphical packages such as **PATRAN** and **I-DEAS** through translators. **GPBEST** stores post-processing data in a neutral file upon request. These are either in the form of **GPBEST** neutral files namely, a direct access file NEUTRAL.OUT or its sequential counterpart NEUTRAL.READ which is suitable for viewing in an editing environment. Further details of the **GPBEST** neutral files files are provided in Chapter 10 of the BESTVIEW manual. Translators are available between the **GPBEST** neutral file and **PATRAN** and **I-DEAS** result post-processing files.



		Section in BESTVIEW Manual
A	PREBEST Translator	9.2 & 9.3
B	PREBEST Translator	9.2 & 9.4
C	Using NEUTRAL Card	10
D	POSTBEST Translator	9.5 & 9.6
E	POSTBEST Translator	9.5 & 9.7

Fig. 4.2 GPBEST File Structure - General Layout

A brief description of the function and content of each of the FT*** files used by **GPBEST** is provided in the following pages.

GPBEST Version 6.0 - FT File Description

```
Solver Files:
  FT002         ISV2    D     Decomposed system matrix for Residual
  FT004         ISV4    D     Stores IPVT arrays

Input/Output Files:
  -----         IR            Input
  -----         IW            Output

Data Base for Input:
  FT011         IM1     S     Material property control
  FT012         IE1     S     Material property data
  FT013         IM2     S     Geometry control
  FT014         IE2     S     Geometry data
  FT015         IM3     S     Boundary condition control
  FT016         IE3     S     Boundary condition data

Boundary Surface Assembly Files:
  gmrnm.FT017   IU1     D     Decomposed A-matrix for
                              boundary displacement
  gmrnm.FT018   IU2     S     Assembled B-matrix for
                              boundary displacement

Results Files:
  gmrnm.FT019   IU3     D     Sampling point displacement
                              solution vector
  gmrnm.FT020   IU4     D     Sampling point displacement initial
                              condition vector
  gmrnm.FT021   IU5     D     Sampling point stress solution vector
  gmrnm.FT022   IU6     D     Sampling point stress initial
                              condition vector

Boundary Surface Convolution Files:
  gmrnm.FT023   IU7     D     G-coefficients in convolution
                              for boundary displacement
  gmrnm.FT024   IU8     D     F-coefficients in convolution
                              for boundary displacement
  gmrnm.FT025   IU9     D     G-coefficients in convolution
                              for interior displacement
  gmrnm.FT026   IU10    D     F-coefficients in convolution
                              for interior displacement
  gmrnm.FT027   IU11    D     G-coefficients in convolution
                              for interior stress
  gmrnm.FT028   IU12    D     F-coefficients in convolution
                              for interior stress

Restart File:
  FT032         IRSI    S     Master restart database

Boundary Surface Integration Files:
  gmrnm.FT033   IRS1    D     G-coefficients for boundary displacement
  gmrnm.FT034   IRS2    D     F-coefficients for boundary displacement
```

```
  gmrnm.FT035   IRS3    D     G-coefficients for interior displacement
  gmrnm.FT036   IRS4    D     F-coefficients for interior displacement
  gmrnm.FT037   IRS5    D     G-coefficients for interior stress
  gmrnm.FT038   IRS6    D     F-coefficients for interior stress

Assembly Control File:
  FT039         IRS7    D     Stores IALLOC arrays

Body Force Files:
  gmrnm.FT056   IBFU    D     Particular integral terms
  FT057         INE1    S     Thermal nodal input
  FT058         INE2    S     Inhomogeneity nodal input

Output File:
  FTOUT         IWALT   -     Alternate output file for no print option

Debug File:
  FT079         IWBUG   S     Debug printouts

Results Files:
  gmrnm.FT080   IXV     D     Solution vector at each time step

Post-processing Files:
  FT100         IUNU    D     Neutral file (NEUTRAL.OUT)

Thermal Distortion Files:
  gmrnm.FT121   ITHDS   D     Temperature and flux solution at
                              each time step
  gmrnm.FT122   ITHD0   D     Thermal G and F-coefficients
                              for boundary displacement
  gmrnm.FT123   ITHD1   D     Thermal G and F-coefficients
                              for interior displacement
  gmrnm.FT124   ITHD2   D     Thermal G and F-coefficients
                              for interior stress

Transient Initial Condition Files:
  gmrnm.FT128   IUTHIC  D     Boundary solution for transient
                              initial conditions

Restart Solver Files:
  FT129         IM3O    S     Old boundary condition control
  FT130         IE3O    S     Old boundary condition data
  FT131         IM3N    S     New boundary condition control
  FT132         IE3N    S     New boundary condition data

Restart Hole Files:
  FT133         IRS1H   D     Temporary G-coefficients
  FT134         IRS2H   D     Temporary F-coefficients

General Anisotropy File:
  FT139         IRSANI  S     Stores anisotropic spline information

Post-processing Files:
  FT161         IUNU2   S     Neutral file (NEUTRAL.READ)

(Note: D stands for Direct Access and S stands for Sequential File)
```

4.5 SPECIAL FEATURES OF GPBEST

4.5.1 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 and hole discretizations, or they may be used to define the location of a sampling point. Points are usually user defined, however, **GPBEST** may create additional points through automated surface generation (see the argument REFNAME on the SURFace card in **GMR input in chapter 5). Additional nodes may also be generated by **GPBEST** for quartic surface elements. 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 or hole element. Geometric points also may or may not be source points.

Source Points - refers to boundary source points in an analysis. Source points are used in the functional representation of variables across a boundary element. 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 **GPBEST** 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 SURFace, TYPE-line, TYPE-surface and TYPE-volume 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 SAMPling-points card in **GMR input in Chapter 5) and are totally independent of the point list used for boundary discretization and independent of the point list for holes. A sampling point may coincide with a boundary discretizations point (but not with a hole). 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 convienient way to model a tubular hole in a 3-D body.

Geometric Modeling Region (GMR) - in a boundary element analysis the body under investigation may be fictitously divided in a number of smaller parts for convienence in mesh modeling and efficiency in computation. Each part is called a geometric modeling region and is modeled 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 geometric 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.)

Residual - in a multi-GMR analysis, the collection of all boundary source points that lie on an interface forms the residual.

Enclosing Elements - These are elements used in GMRs of infinite extent in order to create a fictitious boundary required for correct calculation of the matrix coefficients. 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 unmodeled infinite boundary can be taken into account in the calculation of the diagonal terms of the **F** matrix (Rigid Body Translation Technique).

4.5.2 MESH SIZE

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

4.5.3 SPECIFICATION OF BOUNDARY RESTRAINTS

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

Since the specification of the boundary element condition needs to be very precise in a boundary based solution scheme it is often possible to introduce high stress and strain gradients purely by specifying inappropriate 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. The user may refer to Section 5.5 for further details of these boundary conditions.

It is therefore strongly recommended that a user specifies roller support to simulate the necessary restraints. In most cases, this is the proper idealization of the physical problem.

4.5.4 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)

$$\beta \mathbf{u}^{\mathrm{p}} = \sum_{\mathrm{q}=1}^{\mathrm{N}} \left[\left(\int_{\Delta \mathrm{S}} \mathbf{G}^{\mathrm{pq}} \mathbf{N}^{\mathrm{q}} \mathrm{ds} \right) \mathbf{t}_{\mathbf{n}} - \left(\int_{\Delta \mathbf{S}} \mathbf{F}^{\mathrm{pq}} \mathbf{N}^{\mathrm{q}} \mathrm{ds} \right) \mathbf{u}_{\mathbf{n}} \right] \tag{4.1}$$

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

$$\mathbf{G}\mathbf{t} - \mathbf{F}\mathbf{u} = 0 \tag{4.2}$$

where

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

and

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

The additional columns in $\mathbf{G}$ 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. 4.3a-b) at the corner also presents no difficulty if the final system matrix involving all the unknowns is square and of order $2m \times 2m$ for two-dimensional problems and $3m \times 3m$ for three-dimensional ones. If the displacements alone are specified at the corner (Fig. 4.3c) it is not possible to solve Eq. (4.2) exactly and an approximation will be utilized at this node.

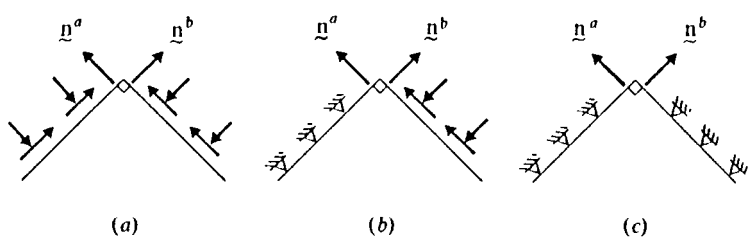


Fig. 4.3 The problems at a corner of the boundary

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

4.5.5 LOAD BALANCE AS A MEASURE OF ACCURACY

As a part of the standard **GPBEST** printed output and neutral file output, element loads and/or heat rates are provided. These quantities are determined by integrating the known tractions and/or heat flux over the surface of the element. In axisymmetry, a circumferential integration is also performed. Hence, the quantities output are total loads and/or heat rates. On the other hand, for two-dimensional problems, the quantities are reported per unit depth.

In addition to the individual element loads, the printed output contains a line labeled LOAD BALANCE for each GMR. The quantity reported is simply a summation of the loads on the individual elements of that region. For many elastic problems, the magnitude of this number should be small compared to that of the applied load. As a result, that ratio can be used as an overall indicator of the quality of the results. Typically, the ratio decreases with increasing mesh refinement or the use of higher order elements, and the LOAD BALANCE approaches zero indicating a converged solution. However, some caution is advised since body forces are not included in this summation. For example, in a problem with centrifugal loading the LOAD BALANCE should not approach zero. Additionally, in problems with planar symmetry, since the LOAD BALANCE does not include the contribution from the reflected elements, in general, a non-zero value will be reported in directions perpendicular to any plane of symmetry.

Similarly, TOTAL HEAT RATE is determined for problems involving heat conduction, however within each GMR, separate totals are provided for contributions of surface and hole elements. The combined TOTAL HEAT RATE for each GMR should approach zero for steady-state problems which do not involve planar symmetry. A ratio of the magnitude of the combined total to that of the applied heat rate may then be used as an overall measure of accuracy. However, this ratio will not approach zero for transient heat conduction nor for problems with planar symmetry.

4.5.6 GETTING THE MOST FROM GPBEST

A BEM analysis is much more efficient when a single homogeneous region is sub-divided into several Geometric Modeling Regions. This is more efficient, not only from the point of view of integration but also for the final solution of the system equations. The cost of a single region BEM analysis increases proportional to N^3 where N is the total number of degrees of freedom for that region. On the other hand, the cost of a multiregion analysis is proportional to $\sum N_i^3$ (with $N = \sum N_i$) where N_i is the total number of degrees of freedom for each region.

It is therefore clearly advantageous to analyze a large problem by artificially subdividing the body into several regions even though a natural partitioning may not exist. Since BEM solutions for such problems are also more accurate, there is compelling arguments (see Figure 4.4) for such multi-region analyses for homogeneous body. In **GPBEST** , a limit on the total degrees of freedom for a single region as well as on the residual interface degrees of freedom have been imposed such that a user is encouraged to use multi-region idealization for large problems.

For example, a single region BEM analysis involving 12000 degrees of freedom takes several hours on a typical workstation. In comparison, the same problem analyzed using an four region model, resulting in 4000 interface degrees of freedom takes nearly half the time with no difference in accuracy.

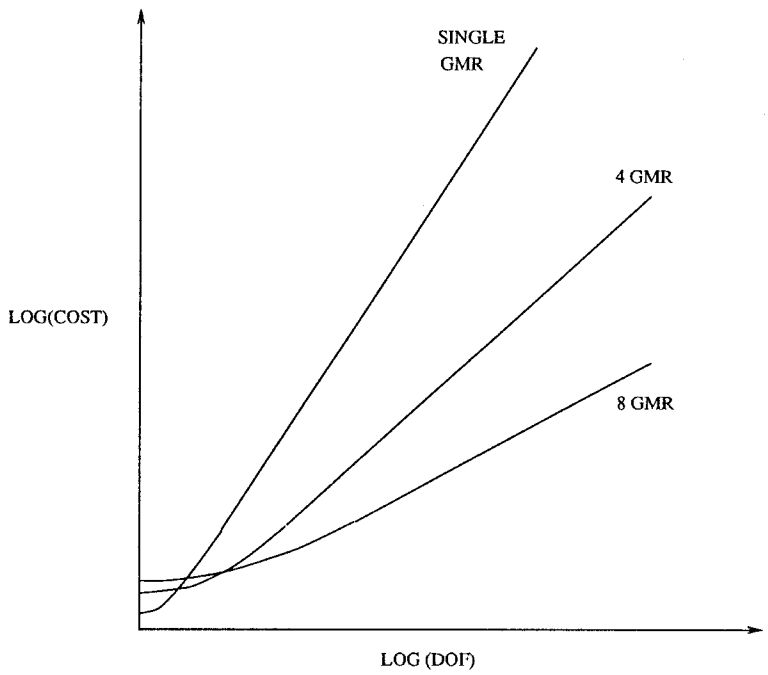


Fig. 4.4 Schematic diagram for cost against degrees of freedom

The following example demonstrates how a large model could be substructured for an efficient analysis using **GPBEST.** Figure 4.5 shows a slice of a turbine disk, modeled to be analysed using the cyclic symmetry option in **GPBEST.** Note that this figure shows the surface patches which almost look like a coarse mesh comprised of higher order elements. The patches are made small in regions with more geometrical complexity. For ease of meshing the patches should be compatible

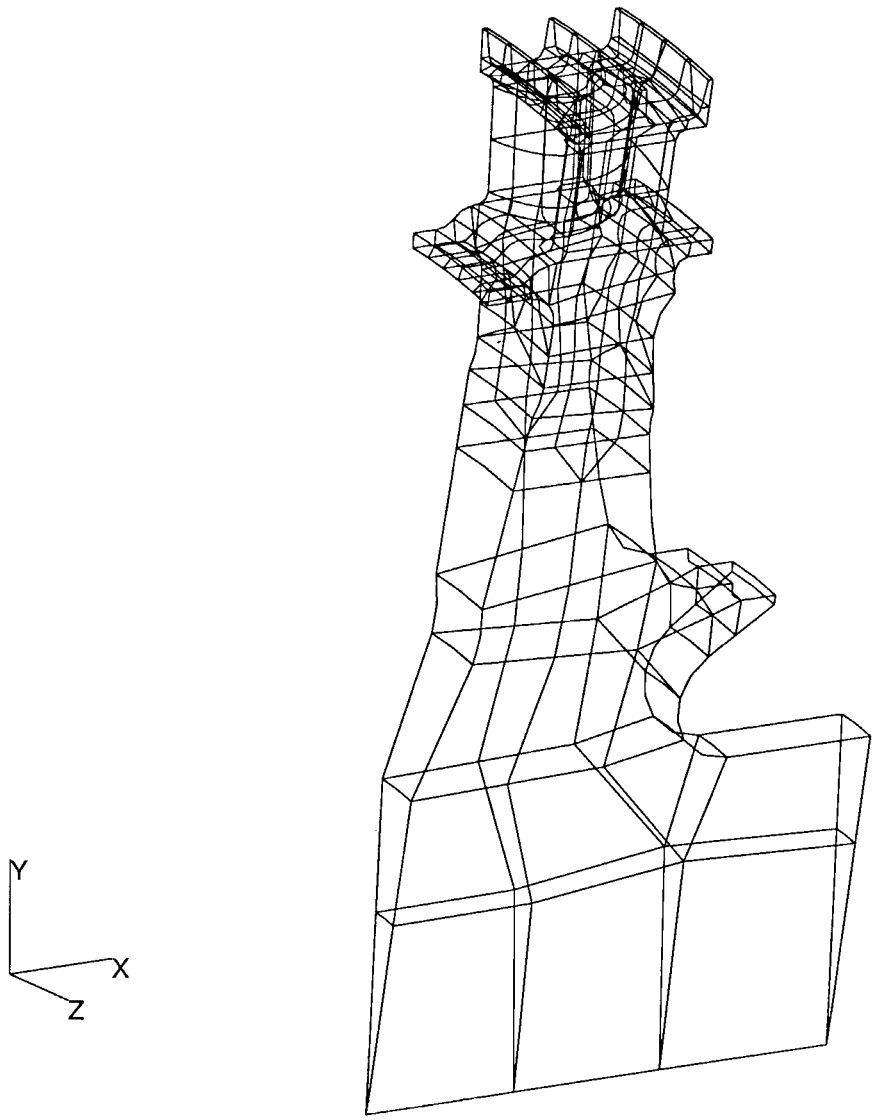


Fig. 4.5 Cyclic Symmetry model of a Turbine Disk

Figure 4.6 shows the four regions into which the model is partitioned. The components are suitably displayed to highlight the interface compatibility between the regions, so that the surface patches match exactly at the interfaces. The substructuring should be done with an intent to restrict the number of nodes per region to less than 2000. For this model the upper two regions could have been further subdivided.

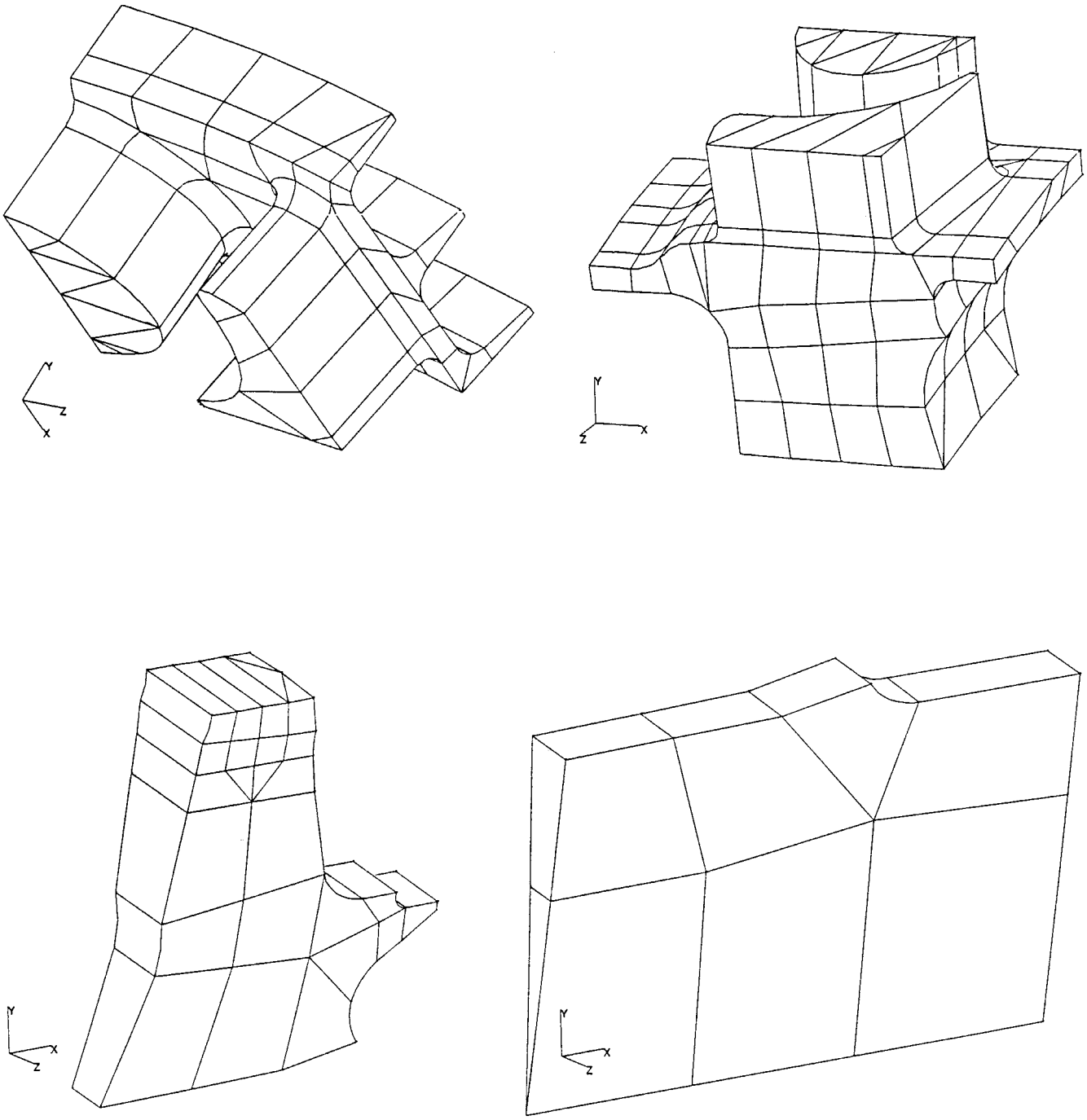


Fig. 4.6 Four regions showing interface compatibility
Note: All components are not displayed to the same scale

The surface patches are then meshed so that each side is divided into two. Figure 4.7 shows the final mesh thus produced.

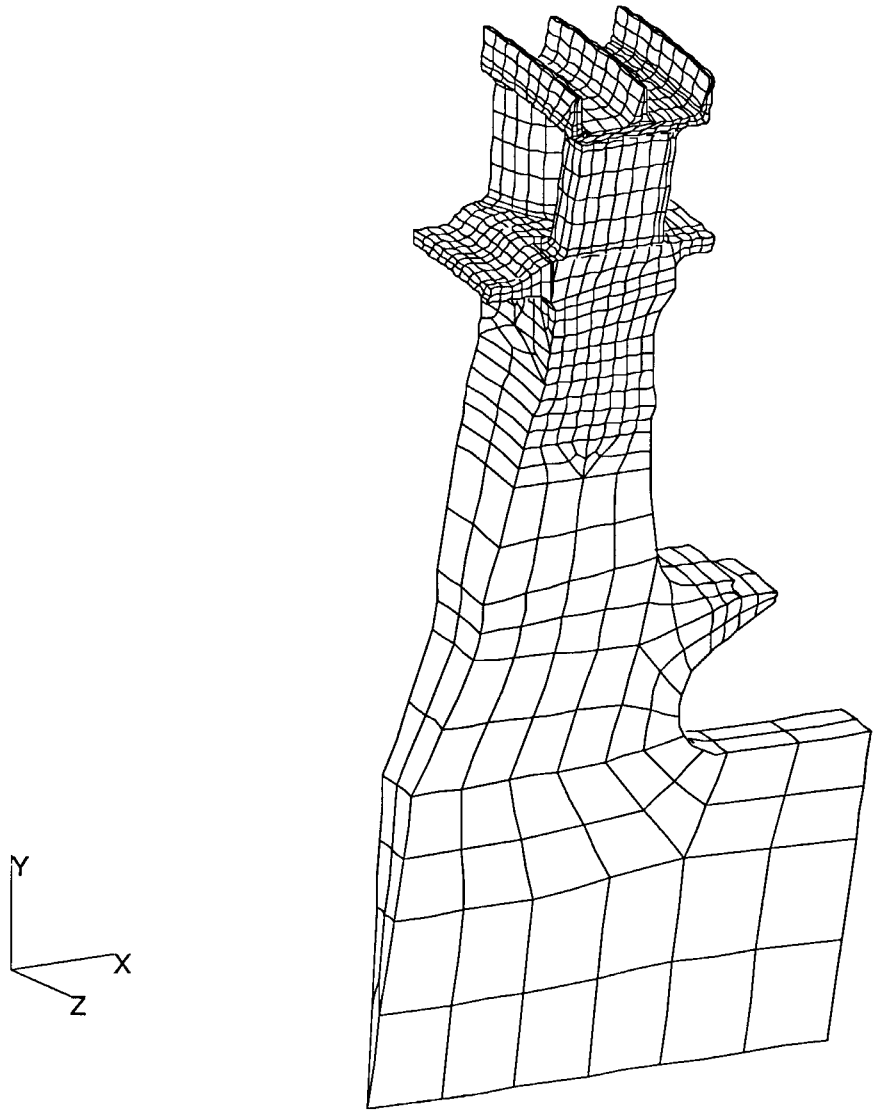


Fig. 4.7 Fine Mesh for Cyclic Symmetry Model

Figures 4.8 to 4.10 show a number of large problems involving 18000 to 24000 degrees of freedom (d.o.f), suitably substructured to provide efficient BEM analyses such that the final solution involves only 3000 to 7000 residual degrees of freedom.

It is frequently expressed that BEM are not good for anything but "potato" like bodies. These problems for which elastic stress analyses were carried out show that BEM is capable of analysing bodies which contains sufficiently difficult geometrical complexities.

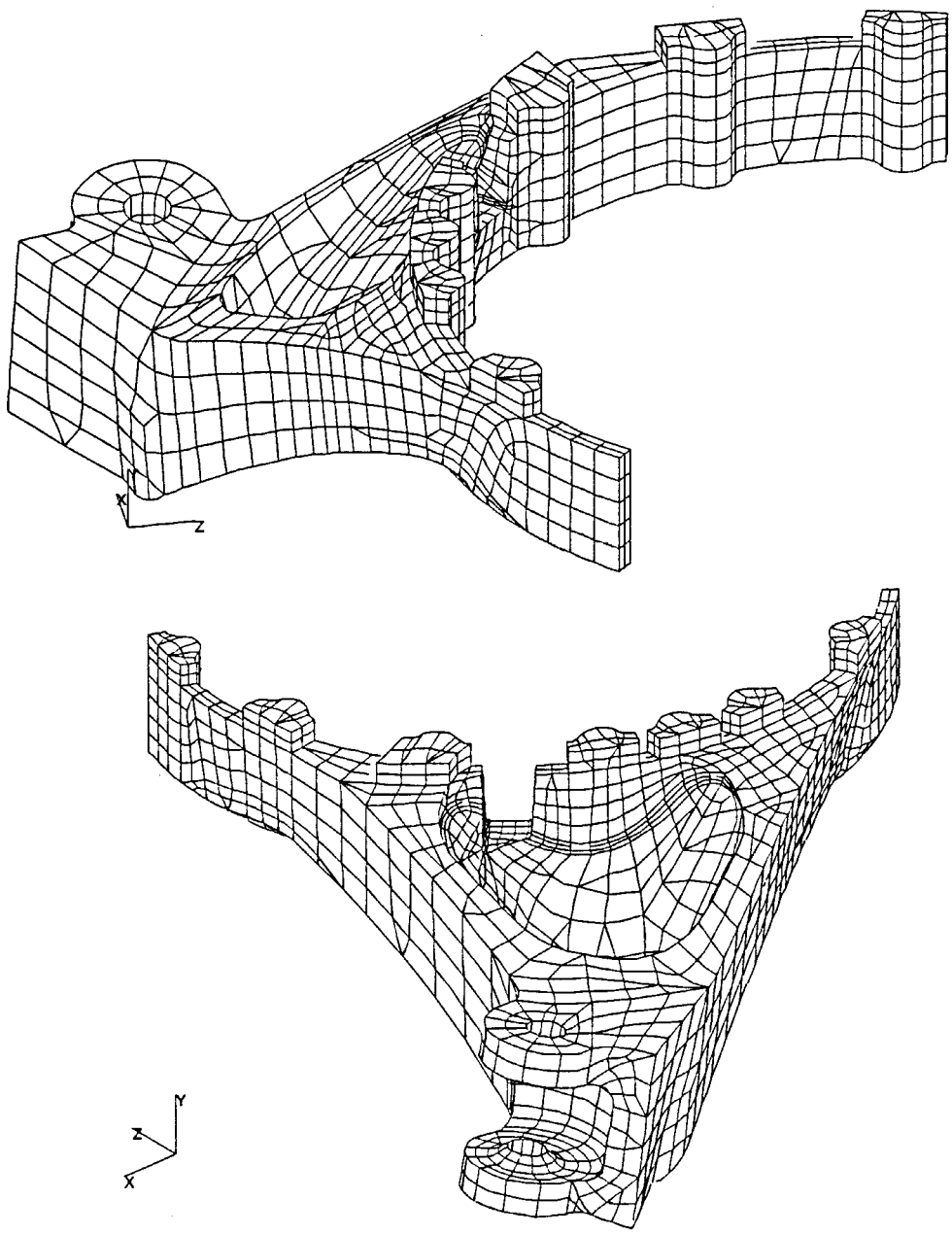


Fig. 4.8 Seven region model(Total d.o.f. 24000; approx. 3500 to 4000 d.o.f per region, leading to a final residual d.o.f. of 5500)

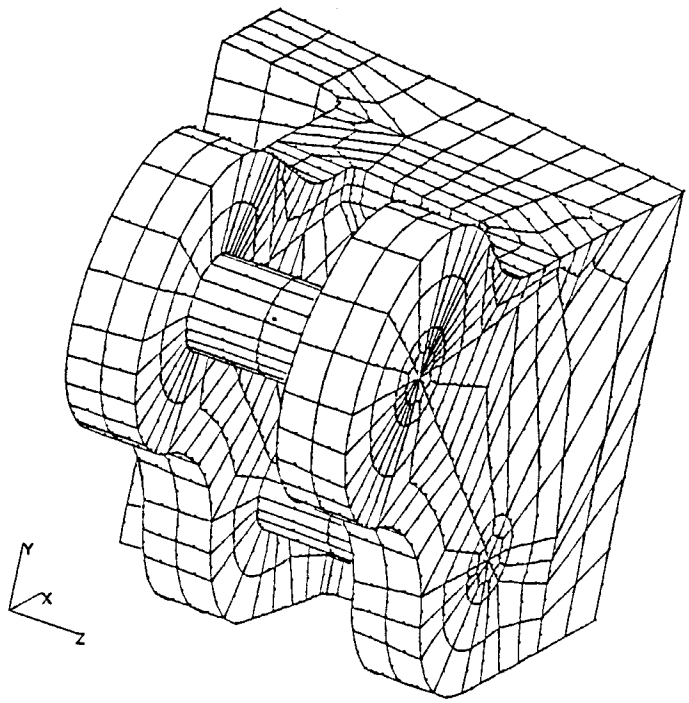


Fig. 4.9 Four region model(Total d.o.f. 18000 with the largest d.o.f. per region 6000 with residual d.o.f. of 3000)

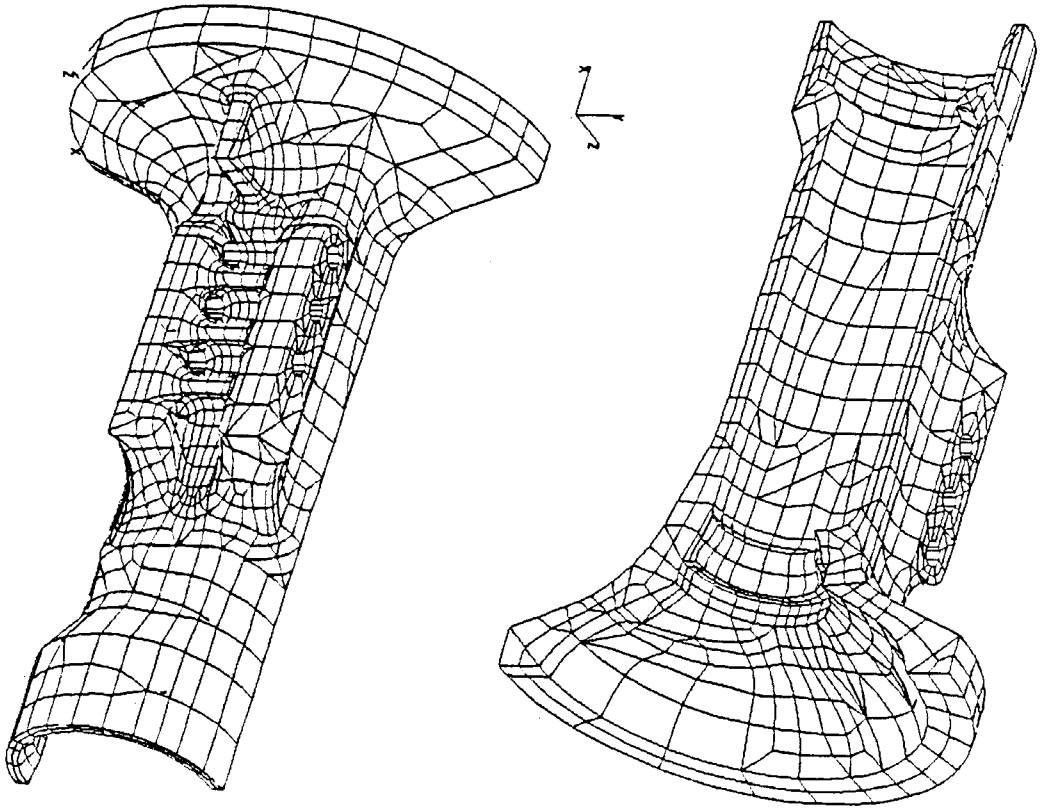


Fig. 4.10 Four region model(Total d.o.f. 23700; approx. 4000 to 4500 d.o.f. per region, leading to a final residual d.o.f. of 7000)

4.5.7 DISK SPACE REQUIREMENTS

In the previous section, the computational advantages of a multi-region approach for large BEM problems was emphasized. There are also significant benefits in terms of disk space utilization. In any case, whether a single or multi-region model is constructed, it is often important to estimate the total storage required for a **GPBEST** analysis, particularly for large problems. The estimate given here are for real arithmatic analyses. For complex arithmatic analyses (e.g. periodic analysis) the storage requirement is doubled.

Consider first an elastic or steady-state heat conduction analysis modeled using a single GMR. Assume at least a moderate size problem with N degrees of freedom (dof), where N $>$ 1000. For a one time step analysis, without restart flags, the only large storage file will be a temporary file named gmrnm.FT017. This file contains the system A-matrix which is assembled and then decomposed in-place. Since single precision storage is used, the total number of bytes required for gmrnm.FT017 is 4N(N+1). The right hand side vector is included in the N+1 column. All of the other temporary disk files utilized by **GPBEST** increase at most linearly with N. (A table describing the contents of the disk files appears in Section 4.4). Generally, the sum total of storage for these other files is less than 25% of that needed for gmrnm.FT017. (The exact size depends upon many factors including the number of points and elements, the number and type of boundary conditions and the type of analysis.) Thus, a rough estimate may be approximately $5N^2$ bytes of disk needed to run the anlaysis. For a single region problem with 6000 dof, this amounts to 150 Mbytes.

On the other hand, in a similar multi-region analysis, a file gmrnm.FT017 is generated for each region. Let N_i represent the total degrees of freedom for the ith GMR while M_i is the portion of these that lie on an interface. Usually, M_i is small compared to N_i. In any case, a file of size $4N_i(N_i + M_i + 1)$ is constructed for each region. Additionally, a global residual system matrix is formed and stored as FT002. The size of FT002 is 4M(M+1), where $M = \sum M_i$. Assuming once again, that the remaining files require an additional 25%, one can estimate the total disk storage space as :

$$D \approx 1.25\left\{\sum 4N_i(N_i + M_i) + 4M^2\right\}.$$

While the computation is a little more cumbersome, in general, considerably less space is needed for a mutli-region analysis. For example, consider a model composed of six equal sized regions. Each region contains 1200 dof, of which 200 define the interface of that region with the rest of the model. Thus, N_i = 1200, M_i = 200 for i = 1,..,6 and M = 1200. One can visualize this as the single region model defined above, sliced into six individual pieces with a total of 1200 dof added to represent the interfaces. However, for the multi-region model, only 60 Mbytes of storage is required.

The above estimates are applicable only to the most basic of the **GPBEST** analysis options. When restart options are invoked, body forces included or transient analyses conducted, additional

storage space is required. The primary purpose of the restart facility is to permit efficient re-analysis with a different set of boundary conditions or with geometric changes in one or two regions. In the restart analysis, numerous operations are skipped, including in many cases, integration and assembly. While this may significantly reduce execution time, unfortunately, additional information must be saved on disk as permanent files. In particular, gmrnm.FT017 becomes a permanent file along with several new files including gmrnm.FT033 and gmrnm.FT034. The first of these new files contains the GMR-based G-coefficients, while the latter stores the F-coefficients. Let L_i represent the total number of traction and/or flux dof in region i counted on an element-by-element basis. Then, the size of gmrnm.FT033 and gmrnm.FT034 is $4N_iL_i$ and $4N_iN_i$ respectively. Consequently, gmrnm.FT034 is of comparable size to gmrnm.FT017, however gmrnm.FT033 is somewhat larger since $L_i > N_i$. Note that since gmrnm.FT033 and gmrnm.FT034 contain matrix coefficients for an individual GMR, their size is not influenced by the interface dof M_i. For an elastic or steady-state heat conduction analysis, which includes a restart request, the total disk space is approximately

$$D \approx 1.25\left\{\sum 4N_i(2N_i + M_i + L_i) + 4M^2\right\}$$

The same two coefficient files gmrnm.FT033 and gmrnm.FT034 are needed for the right-hand side computation when body forces are included in the analysis. Consequently, the above formula is appropriate for that case, with or without a restart flag. Returning to the six region hypothetical example, assume now that a restart option has been selected. In three-dimensional problems, generally $L_i \approx 3N_i$. (In 2-D and axisymmetry, $L_i \approx 1.5N_i$). Thus, for 3-D from the above formula, approximately 230 Mbytes of storage is required.

The final case to be considered involves transient analyses. In order to retain a boundary-only approach, a convolution in time must be performed. Consequently, a set of G and F-coefficients must be stored for each time step. The files gmrnm.FT023 and gmrnm.FT024 are constructed from these coefficients. In a heat conduction analysis involving K time steps, these two files are of the size $4(K-1)N_iL_i$ and $4(K-1)N_iN_i$, respectively. Then, the total storage, assuming no restart option is selected, becomes

$$D \approx 1.25\left\{\sum 4N_i(N_i + M_i) + 4M^2 + \sum 4(K-1)N_i(L_i + N_i)\right\}$$

For a ten time step, three-dimensional transient heat conduction analysis, approximately 1600 Mbytes of storage is required. While this is a very large requirement, it must be remembered that the model under consideration would involve 7200 source points. This represents an extremely refined boundary element model. On the other hand, for a J-dimensional transient concurrent thermoelastic analysis the convolution is only needed over the thermal degrees of freedom. Consequently, the size of gmrnm.FT023 and gmrnm.FT024 becomes $4(K-1)N_iL_i/(J+1)$ and $4(K-1)N_iN_i/(J+1)$, respectively. Returning to the ten time step 3-D example, but now assuming transient concurrent thermoelastic analysis, one determines a disk storage requirement of 450 Mbytes. Note that this model would contain 7200/(J+1) or 1800 source points, which still represents a detailed model by present boundary element standards.

The simple formulae and discussion provided in this section are intended to provide some guidance to the user not only in determining disk space requirements for an existing model, but also for the conceptualization of boundary element models. For example, the advantages of using multi-region models, even for a single homogeneous component, should be kept in mind. In PATBEST and IDEASBEST interfaces the disk space requirements are printed out in the journal file for the benefit of the user.

4.6 USEFUL UNIX-BASED OPERATIONS

Some useful hints for carrying out different operations such as compiling, executing and results processing within a UNIX system, are provided below for a SUN operating system. The users, however, must be fimiliar with the shell (which may differ from those given here for their own operating system) as well as editors on their system for creating and editing files. These UNIX commands are to be executed at the system prompt, henceforth denoted by the symbol: 'prompt >'.

(1.) To run a job using the **GPBEST** executable file (for example, best-s) and a **GPBEST** data set (for example, datafile) and storing the results in a results file (for example, resultfile):

```
prompt > < datafile  best-s > resultfile
```

(2.) To run multiple jobs using a batch file (e.g., Runproc) where all **GPBEST** data sets are of form *.dat and the results files created are of the form *.r. First, a file (e.g., Rundata) containing all data sets without the extension '.dat' is created. In the above, the character asterisk (*) stands for a wild-card encompassing all files fitting the pattern. To run the datasets elas001.dat, frac003.dat and heat040.dat using the batchfile, the file Rundata is constructed as follows:

```
elas001
frac003
heat040
```

Note, NEUTral-file post-processing files are created for each problem listed in Rundata. These need to be moved to other locations to prevent them from being overwritten. The batch file Runproc is constructed for a SUN operating system as follows:

```
#
#  Use :    (nohup time csh Runproc) > & Runtot &
#  Rundata must contain data file names sans extension
#
set Runtemp = ‘cat Rundata‘
foreach i ($Runtemp)
echo "$i.dat  ‘hostname‘  ‘date‘ " > $i.r
(time <$i.dat nice best-s >> $i.r )>& b.err
cat b.err >> $i.r
rm b.err
echo "$i.dat  ‘hostname‘  ‘date‘ " >> $i.r
mv NEUTRAL.OUT  $i.nout
mv NEUTRAL.READ  $i.nread
end
```

As seen above, in addition to executing the job with each datafile the above batch file also carries out the operation of moving all post-processing files created at the completion of

each of the runs. Thus, these files are not overwritten, but stored along with name of the corresponding data file for easy reference. The procedure Runproc is then executed using the following command :

```
prompt > (nohup time csh Runproc) > & Runtot &
```

All system messages are stored in the file called Runtot.

(3.) To process the data from the result file ; for instance, in order to search the result file for a pattern or string of characters which may relate a particular element, node or time step value, the following commands are very useful:

```
prompt > grep 'pattern'  resultfile(s) >  storefile
```

The command described above produces a list of all lines from the resultfile which contain the prescribed pattern. This may be used for compiling a list of results pertaining to a certain node or time value. The users should refer to the help information provided by their respective systems for more information on the 'grep' command.

Another command that is useful within UNIX is the 'awk' command. It allows for further manipulation of data after searching for a particular string of characters. For example, a starting pattern and an ending pattern may be specified and all lines between these two patterns will then be stored in a file by using the 'awk' command as follows:

```
prompt > awk '/pattern1/, /pattern2/'  resultfile(s) >  storefile
```

The users should refer to the help information provided by their respective systems for more information on the 'awk' command.

Examples of Use:

(i) To search for all results data pertaining to Node ID 201 in the result file elas001.r and store in a file called data.201:

```
prompt > grep '  201  '  elas001.r >  data.201
```

(ii) To search for all lines of data starting with Node ID 101 and ending with the pattern JOB in the result file heat040.r and store in a file called data.101:

```
prompt > awk '/ 101 /, /JOB/'  heat040.r >  data.101
```

5.0 GPBEST INPUT

The basic input required by **GPBEST** 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 **GPBEST** 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.

Some simple example problems with sample **GPBEST** input data is provided in Section 6. In addition several hundred verification problems are supplied with the program. The user is encouraged to study and understand these problems as well, to gain a clearer picture of the different keywords used in **GPBEST** input.

5.0.1 General Rules for Input Data

1. **Upper Case**

   All alphanumeric input must be provided in upper case.

   Proper Usage:

```
**CASE
  TITLe  HEAT TRANSFER - TEST CASE
  HEAT
  SYMMetry QUARTER
```

   Improper Usage:

```
  heat
  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. Tab characters are not permitted.

   Proper Usage:

```
CTHErmal STEADY
ELEMents-surface 1 6 8
```

   Improper Usage:

```
CTHERMALSTEADY
ELEMents-surface, 1, 6, 8
```

3. **Length of Input Line**

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

4. Keyword Truncation

Any keywords that are longer than four letters may be truncated to the first four letters. This has been emphasized throughout this manual by lowercase letters following the first four capital letters of a keyword.

Proper Usage:

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

Improper Usage:

```
SYMM QUA
```

5. Floating Point Numbers

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

Proper Usage:

```
EMODulus 30.E+7
ALPHa-temp-dep 1.E-06
POINts-bc
  0.004 1.110 0.0
```

Improper Usage:

```
EMODulus 300000000.0
ALPHa-temp-dep 1.-6
POINts-bc
  4.0-3 1.110 0
```

6. **Comments**

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

Proper Usage:

```
ELEMents-surface 1 6 8 $ ELEMENTS ON THE OUTER RIM
$
$ MODIFIED 03/08/88 GFD
  POINts 25 26 27
```

7. **Blank Lines**

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

8. **Include Files**

The INCLude-file key word can be inserted anywhere in the data file. This permits additional lines of data to be stored in separate files. See the full description for this card in Section 5.1.11 of this manual.

9. **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.

10. ** 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
**DCE

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

5.0.2 General Recommendations for Input Data

1. **Ordering of Input Items**

   While there is some flexibility in the ordering of lines within a **GPBEST** 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 $ or # keywords are provided to permit comments anywhere in the input data set. This should be used generously to fully document the analysis. Blank spaces can also be used to improve readability. The format, displayed in the examples of this chapter and in Section 6.0, is recommended.

5.0.3 General Limits of GPBEST

It should be noted that there are certain limits which must be observed in the preparation of input for **GPBEST** . The present limits are summarized below. Also refer to Section 6.1.2 of this manual, which describes how to determine limits for specialized versions of **GPBEST** .

ENTITY[1]	best-s best-t	best-s80 best-t80
GLOBAL PARAMETERS		
total geometric modeling regions	20	80
total degrees of freedom[2]	24000	80000
total residual degrees of freedom (dof on interfaces)	10000	20000
total geometric points (includes non-source points)	16000	54000
total boundary source points	8000	20000
total boundary elements	4000	18000
total line elements	400	400
total enclosing elements	100	100
total discontinuity elements	100	100
REGION (GMR) PARAMETERS		
surfaces in any region	15	15
degrees of freedom in any region[2]	8000	12000[3]
point sources/forces in any region	25	25
OTHER PARAMETERS		
time points for elastic analysis[4]	20	20
table points for boundary condition and body force definition	20	20
temperature points for material properties	21	21
USER SPECIFIED NUMBERING		
points[5]	99999	99999
elements[5]	99999	99999

1 Definition of the terminology used in this table can be found in Section 4.5.

2 For analyses involving complex arithmetic, the total degrees of freedom is half.

3 For an exterior region in periodic acoustic analysis, the region degree of freedom is 8000 total for best-t80.

4 For periodic analyses, instead of time points, frequencies have been used.

5 For generalised axisymmetry, the limit is 9999

5.0.4 Individual Data Items

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

5.1 CASE CONTROL INFORMATION (**CASE)
5.2 MATERIAL PROPERTY DEFINITION (**MATERIAL)
5.3 GEOMETRY DEFINITION (**GMR)
5.4 INTERFACE DEFINITION BETWEEN SUBREGIONS (**INTERFACE)
5.5 BOUNDARY CONDITION DEFINITION (**BCSET)
5.6 BODY FORCE DEFINITION (**BODY)
5.7 BOUNDARY CONDITION CHANGE DEFINITION (**BCCHANGE)
5.8 GPBEST-DCE CONTROL (**DCE)

5.0.5 GPBEST Keyword Input Card Quick Reference

****CASE**
TITLe
AXISymmetry
PLANe
MIXEd
ACOUstic
CONSolidation
CTHErmal
DIFFusion
DISTortion
ELAStic
FREE-vibration
FORCed-vibration
HEAT
PLASticity
SACOustic
FRACture
TIMEs-output
TIMEs
FREQuency-output
FREQuency
SYMMetry
RESTart
SAVE
INITial
PRECision
STORage-precision
SOLVer-precision
SOLVer-algorithm
QUICk-solver
STANdard-solver
MEMOry
TOLErance-convergence
MAXImum-iteratons
ECHO-input
PRINtout-control
NEUTral-file
HISTory-quantities
CONTour-plot
DECIbel-output
VECTor-intensity
EIGEnvalue
CHECk-data
LOAD
BCFIle
INCLude-file
COORdinate-system
EIGEnvalue-option-1
EIGEnvalue-option-2
ITERative
NEWTon-raphson

****MATERIAL**
ID-Material
DENSity
EMODulus
ALPHa
POISson
SPEEd
CONDuctivity
SPECific
TEMPeratures
EMODulus-temp-dep
ALPHa-temp-dep
ANISotropy
STIFfnesses
COMPliances
TECHnical
ALPHa-axes
CONDuctivity-axes
ORIEntation
FDENsity
PERMeability
SBULk
UPOIsson
DAMPing
INELastic
VON-mises
YIELd
CURVe
(NONE)
TMELt
TPEAk
MODIfied
ISOII
KAPPa
LAMBda
VOID-ratio
PHI-soil
CURRent-stress
PAST-maximum-stress
COHEsion

****GMR**
ID-Gmr
AXISymmetry-gmr
PLANe-gmr
THICkness
MATErial
RESTart-gmr
SAVE-gmr
TINTegration
TREFerence
EXTErior
HALF
CONTour-gmr
POINts
(NONE)
SURFace
TYPE-surface
GEOMetry
ELEMents-surface
(NONE)
TRANslate
REFErence
DIREction
ROTAtion
CRACk
NORMal
ENCLosing-elements
(NONE)
DISContinuity-elements
(NONE)
LINE-element-definition
POINts-line
(NONE)
TYPE-line
HOLE
RUNNer
CLINe
TREFerence-line
CONVection-line
RESIstance-line
ELEMents-line
(NONE)
SAMPling-points
(NONE)
SAMPling-surfaces
ELEMents-sampling-surf
(NONE)
HISTory-points
VOLUme
TYPE-volume
CELLs
(NONE)
FULL
PROPerty
MASS
OVDAmping
ECCEntricity

****INTERFACE**
ID-Interface
GAP-interface
TOLErance
SANDwich
GMR-1
SURFace-gmr-1
CLINe-gmr-1
ELEMents-gmr-1
POINts-gmr-1
GMR-2
SURFace-gmr-2
CLINe-gmr-2
ELEMents-gmr-2
POINts-gmr-2
BONDed
SLIDing
FIT-value
CONTact
SPRIng-value
CONVection-value
RESIstance-value
GENEralized-interface
IMPEdance-value
ADMIttance-value
FRICtion
EQUIvalence
OFFSet
O-VAlue
CYCLic
ANGLe
DIREction-cyclic

****BODY**
CENTrifugal
POINt-centrifugal
DIREction-centrifugal
TIMEs-centifugal
SPEEd-centrifugal
INERtia-force
DIREction-inertia-force
TIMEs-inertia-force
ACCEleration-inertia
THERmal
TIMEs-thermal
GMR-thermal
TEMPeratures-thermal
(NONE)
(TERM)
GENEralized
TIMEs-generalized
GMR-generalized
CONStant
XVALue
YVALue
CONCentrated
TIMEs-concentrated
FREQuencies-concentrated
GMR-concentrated
POINts-concentrated
SOURce-concentrated
FORCe-concentrated
T-COncentrated
DISContinuity-surface
TIMEs-disc-surf
FREQuencies-disc-surf
GMR-disc-surf
DISPlacement-disc-surf
(NONE)

****BCSET**
ID-Bcset
VALUe
RELAtion
VARIable
MOVIng
INTErface
LOCAl
DIREction-bc
(NONE)
GMR-bc
SURFace-bc
ELEMents-bc
POINts-bc
TIMEs-bc
FREQuency-bc
TRACtion-bc
LOAD-bc
GLOAd-bc
TORQue-bc
DISPlacement-bc
FLUX-bc
TEMPeratures-bc
PRESsure-bc
VELOcity-bc
GMR-line-bc
HOLE-line-bc
CLINe-line-bc
ELEMents-hole-cline
POINts-line-bc
TIMEs-line-bc
FLUX-line-bc
TEMPeratures-line-bc
SPLIst
T-VAlue
SPRIng
CONVection
RESIstance
IMPEdance
ADMIttance
SOURce
TIMEs-moving
ENERgy-rate
LOCAl-fem
POINts-fem
DISPlacement-fem
STIFfness-fem
FORCe-fem
DROP-fem

****BCCHANGE**
TIMEs-bcch

****DCE**
HOST
MFLOp
SCRAtch
COURtesy
LIMIt
MAINdir
MAXTask
MEMOry-dce
MULTiple
PASSword
PRIOrity
RESULt

5.0.6 GPBEST Keyword Input Card Quick Reference

Alphabetically Sorted

****CASE**
ACOUstic
AXISymmetry
BCFIle
CHECk-data
CONSolidation
CONTour-plot
COORdinate-system
CTHErmal
DECIbel-output
DIFFusion
DISTortion
ECHO-input
EIGEnvalue
EIGEnvalue-option-1
EIGEnvalue-option-2
ELAStic
FORCed-vibration
FRACture
FREE-vibration
FREQuency
FREQuency-output
HEAT
HISTory-quantities
INCLude-file
INITial
ITERative
LOAD
MAXImum-iteratons
MEMOry
MIXEd
NEUTral-file
NEWTon-raphson
PLANe
PLASticity
PRECision
PRINtout-control
QUICk-solver
RESTart
SACOustic
SAVE
SOLVer-algorithm
SOLVer-precision
STANdard-solver
STORage-precision
SYMMetry
TIMEs
TIMEs-output
TITLe
TOLErance-convergence
VECTor-intensity

****MATERIAL**
(NONE)
ALPHa
ALPHa-axes
ALPHa-temp-dep
ANISotropy
COHEsion
COMPliances
CONDuctivity
CONDuctivity-axes
CURRent-stress
CURVe
DAMPing
DENSity
EMODulus
EMODulus-temp-dep
FDENsity
ID-Material
INELastic
ISOII
KAPPa
LAMBda
MODIfied
ORIEntation
PAST-maximum-stress
PERMeability
PHI-soil
POISson
SBULk
SPECific
SPEEd
STIFfnesses
TECHnical
TEMPeratures
TMELt
TPEAk
UPOIsson
VOID-ratio
VON-mises
YIELd

****GMR**
(NONE)
(NONE)
(NONE)
(NONE)
(NONE)
(NONE)
(NONE)
(NONE)
(NONE)
AXISymmetry-gmr
CELLs
CLINe
CONTour-gmr
CONVection-line
CRACk
DIREction
DISContinuity-elements
ECCEntricity
ELEMents-line
ELEMents-sampling-surf
ELEMents-surface
ENCLosing-elements
EXTErior
FULL
GEOMetry
HALF
HISTory-points
HOLE
ID-Gmr
LINE-element-definition
MASS
MATErial
NORMal
OVDAmping
PLANe-gmr
POINts
POINts-line
PROPerty
REFErence
RESIstance-line
RESTart-gmr
ROTAtion
RUNNer
SAMPling-points
SAMPling-surfaces
SAVE-gmr
SURFace
THICkness
TINTegration
TRANslate
TREFerence
TREFerence-line
TYPE-line
TYPE-surface
TYPE-volume
VOLUme

****INTERFACE**
ADMIttance-value
ANGLe
BONDed
CLINe-gmr-1
CLINe-gmr-2
CONTact
CONVection-value
CYCLic
DIREction-cyclic
ELEMents-gmr-1
ELEMents-gmr-2
EQUIvalence
FIT-value
FRICtion
GAP-interface
GENEralized-interface
GMR-1
GMR-2
ID-Interface
IMPEdance-value
OFFSet
O-VAlue
POINts-gmr-1
POINts-gmr-2
RESIstance-value
SANDwich
SLIDing
SPRIng-value
SURFace-gmr-1
SURFace-gmr-2
TOLErance

****BODY**
(NONE)
(NONE)
(TERM)
ACCEleration-inertia
CENTrifugal
CONCentrated
CONStant
DIREction-centrifugal
DIREction-inertia-force
DISContinuity-surface
DISPlacement-disc-surf
FORCe-concentrated
FREQuencies-concentrated
FREQuencies-disc-surf
GENEralized
GMR-concentrated
GMR-disc-surf
GMR-generalized
GMR-thermal
INERtia-force
POINt-centrifugal
POINts-concentrated
SOURce-concentrated
SPEEd-centrifugal
T-COncentrated
TEMPeratures-thermal
THERmal
TIMEs-centifugal
TIMEs-concentrated
TIMEs-disc-surf
TIMEs-generalized
TIMEs-inertia-force
TIMEs-thermal
XVALue
YVALue

****BCSET**
(NONE)
ADMIttance
CLINe-line-bc
CONVection
DIREction-bc
DISPlacement-bc
DISPlacement-fem
DROP-fem
ELEMents-bc
ELEMents-hole-cline
ENERgy-rate
FLUX-bc
FLUX-line-bc
FORCe-fem
FREQuency-bc
GLOAd-bc
GMR-bc
GMR-line-bc
HOLE-line-bc
ID-Bcset
IMPEdance
INTErface
LOAD-bc
LOCAl
LOCAl-fem
MOVIng
POINts-bc
POINts-fem
POINts-line-bc
PRESsure-bc
RELAtion
RESIstance
SOURce
SPLIst
SPRIng
STIFfness-fem
SURFace-bc
TEMPeratures-bc
TEMPeratures-line-bc
TIMEs-bc
TIMEs-line-bc
TIMEs-moving
TORQue-bc
TRACtion-bc
T-VAlue
VALUe
VARIable
VELOcity-bc

****BCCHANGE**
TIMEs-bcch

****DCE**
COURtesy
HOST
LIMIt
MAINdir
MAXTask
MEMOry-dce
MFLOp
MULTiple
PASSword
PRIOrity
RESULt
SCRAtch

5.1 CASE CONTROL

This input section provides **GPBEST** with information controlling the overall execution of the problem. 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 only one time for each analysis and must be input before any other data.

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

SECTION	KEYWORD	PURPOSE	PAGE
5.1.1	Case Control Input Card		5.1.6
	**CASE	Start of case control input	5.1.6
5.1.2	Title		5.1.7
	TITLe	Title of job	5.1.7
5.1.3	Dimensionality of the Problem	(default is 3-D)	5.1.8
	AXISymmetry	Axisymmetric flag	5.1.8
	AXISymmetry GENE	Axisymmetric body with general (non-axisymmetric) loading	5.1.8
	PLANe	2-D flag	5.1.11
	PLANe STRE	Plane stress flag	5.1.11
	PLANe STRA	Plane strain flag	5.1.11
	MIXEd	Dimensionality is region-dependent	5.1.12
5.1.4	Type of Analysis		5.1.13
	ACOUstic	Acoustic analysis	5.1.13
	CONSolidation	Consolidation (poroelastic and) poroplastic analysis	5.1.19
	CTHErmal	Concurrent thermal analysis (thermoelastic and thermoplastic) of an isotropic solid	5.1.20
	DIFFusion	Diffusion analysis	5.1.21
	DISTortion	Thermal distortion analysis	5.1.22
	ELAStic	Elastic analysis	5.1.23
	FREE-vibration	Free vibration analysis	5.1.24
	FORCed-vibration	Forced vibration analysis	5.1.25

SECTION	KEYWORD	PURPOSE	PAGE
	HEAT	Heat transfer analysis	5.1.26
	HEAT TRAN	Transient heat transfer analysis	5.1.26
	PLASticity	Plastic analysis	5.1.27
	SACOustic	Structural acoustics	5.1.28
5.1.5	Analysis-Type Modifiers		5.1.31
	FRACture	Fracture Mechanics analysis	5.1.31
5.1.6	Times/Frequencies for Solution and Output		5.1.32
	TIMEs-output	Times for solution output	5.1.32
	TIMEs STEP	Time-step for transient solution algorithms	5.1.33
	TIMEs VARI	Time variation assumption for transient analysis	5.1.35
	FREQuency-output	Frequencies for full output of results	5.1.36
	FREQuency SWEEP	Frequency sweep (minimal output of results)	5.1.37
5.1.7	Geometric and Loading Symmetry Control		5.1.39
	SYMMetry HALF	Half Symmetry	5.1.39
	SYMMetry QUAR	Quarter Symmetry	5.1.39
	SYMMetry OCTA	Octal Symmetry	5.1.39
5.1.8	Restart Facility		5.1.46
	RESTart WRIT	Save integration files for future runs	5.1.46
	RESTart READ	Use integration files from previous run	5.1.46
	RESTart HOLE	Use surface integration files from previous run, but permit changes in hole elements	5.1.46
	RESTart SOLV	Use the decomposed system matrix from previous run	5.1.46
	RESTart GMR	Use a GMR-based restart facility	5.1.46
	SAVE SOLV	Save the decomposed system matrix and right hand side matrices for future runs	5.1.50
	INITial STEA	Use the results from a previous steady state analysis as an initial condition	5.1.51

SECTION	KEYWORD	PURPOSE	PAGE
5.1.9	Efficiency and Precision Control		5.1.52
	PRECision	Precision of analysis	5.1.52
	STORage-precision	Precision for storage of equation systems	5.1.53
	SOLVer-precision SING	Use single precision variables in solver	5.1.54
	SOLVer-precision DOUB	Use double precision variables in solver	5.1.54
	SOLVer-algorithm ELIM	Use a solver algorithm based on gaussian elimination	5.1.55
	SOLVer-algorithm ITER	Use an efficient iteration-based solver algorithm	5.1.55
	QUICk-solver	Use Quick (block) elimination solver	5.1.57
	STANdard-solver	Use Standard (line) elimination solver	5.1.58
	MEMOry	Specify dynamic memory allocation for use in the system equation solver	5.1.59
	TOLErance-convergence	Set convergence tolerance	5.1.60
	MAXImum-iterations	Set maximum number of iterations	5.1.61
5.1.10	Output Options		5.1.62
	ECHO-input	Produce echo of input data	5.1.62
	PRINtout-control BOUN	Print displacement and traction results	5.1.63
	PRINtout-control NODA	Print boundary displacement, stress, strain at nodal points	5.1.63
	PRINtout-control LOAD	Print load calculation	5.1.63
	PRINtout-control DATA	Print input data	5.1.63
	PRINtout-control ALL	Provide the maximum printed output file	5.1.63
	PRINtout-control NONE	Provide only a minimal printed output file	5.1.63
	PRINtout-control LIMI	Print current GPBEST limits	5.1.63
	PRINtout-control FEAT	Print current implementation status of GPBEST special features	5.1.63

SECTION	KEYWORD	PURPOSE	PAGE
	NEUTral-file	Produce formatted GPBEST neutral file (all quantities)	5.1.65
	NEUTral-file BOUN	Print boundary quantities in neutral file	5.1.65
	NEUTral-file NODA	Print nodal quantities in neutral file	5.1.65
	NEUTral-file LOAD	Print load balance in neutral file	5.1.65
	NEUTral-file DATA	Print input data in neutral file	5.1.65
	NEUTral-file ALL	Print all quantities in neutral file	5.1.65
	NEUTral-file READ	Write a sequential neutral file	5.1.65
	NEUTral-file MODI	Write the neutral file in a modified form	5.1.65
	HISTory-quantities	Define which quantities will be output in the history file	5.1.68
	CONTour-plot	Create output in neutral file for contour plots of results in post-processing 2D problems	5.1.70
	DECIbel-output	Define reference values for decibel output of results	5.1.74
	VECTor-intensity	Restrict the output of Intensity vectors in the neutral file (for BESTVIEW)	5.1.76
	EIGEnvalue OUTPUT	Specify the number of eigenvalue and eigenvectors to be output	5.1.77
5.1.11	Miscellaneous Control Options		5.1.78
	CHECk-data	Check input data only	5.1.78
	LOAD COMP	Identifies complex valued boundary condition in acoustic/periodic dynamic analysis	5.1.79
	BCFIle	Specify a file that contains nodal based displacement or velocity boundary conditions for periodic acoustic analysis	5.1.81
	INCLude-file	Include an external file in the GPBEST data set	5.1.84
	COORdinate-system	Specifies coordinate system for input data	5.1.86
	EIGEnvalue-option-1 SAMP	Specify eigenvalue option	5.1.88
	EIGEnvalue-option-2 SHIFT	Specify eigenvalue option	5.1.90

SECTION	KEYWORD	PURPOSE	PAGE
5.1.12	Nonlinear Algorithm Control Options		5.1.91
	ITERative	Select the iteration algorithm	5.1.91
	NEWTon-raphson	Select the Newton-Raphson algorithm	5.1.92

5.1.1 CASE CONTROL INPUT CARD

**CASE

Status - REQUIRED

Full Keyword - **CASE**

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

Input Variables - NONE

Additional Information - NONE

Examples of Use -

1. Request a plane stress elastic analysis.

```
**CASE
  TITLe  PLANE STRESS ANALYSIS OF A BAR
  PLANe STRESS
  ELAStic
```

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

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

5.1.2 TITLE

TITLe CASETITLE

Status - REQUIRED

Full Keyword - **TITLe**

Function - Defines title for analysis.

Input Variables -

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

Additional Information - NONE

Examples of Use -

1. Describe the analysis.

```
**CASE
  TITLe  TURBINE BLADE A7311 - THERMOELASTIC ANALYSIS
  ELAStic
```

5.1.3 DIMENSIONALITY OF THE PROBLEM

(Default is 3-D geometry)

AXISymmetry ETYPE

Status - OPTIONAL

Full Keyword - **`AXISymmetry`**

Function - Identifies an axisymmetry problem.

Input Variables -

ETYPE (Alphanumeric) - OPTIONAL

Allowable value is GENERAL

GENERAL - Specifies a generalized axisymmetric problem (axisymmetric body under non-axisymmetric loading) Generalized axisymmetry is available only for periodic acoustic and forced vibration analysis.

Additional Information -

In axisymmetry, the first coordinate represents the radial direction (referred to as the r-axis in this manual) and the second coordinate represents the axial direction (referred to as the z-axis). The loading is assumed to be axisymmetric if the variable GENERAL is not specified.

In axisymmetric analysis, the axis of symmetry should not be modeled. Therefore, boundary elements should not lie along the axis of symmetry.

In axisymmetric analysis, only the discretization of the generator of axisymmetric body is needed.

The use of axisymmetry (with axisymmetric boundary conditions) automatically invokes the condition of zero radial displacement at the origin. Therefore, displacement in the radial direction does not have to be set to zero at the origin or at any other point for the purpose of preventing arbitrary ‘rigid body’ motion in the radial direction. The displacement in the axial direction, however, must still be fixed at some point in order to prevent rigid body translation in this direction.

In **general axisymmetric** analysis, the variation of boundary conditions in circumferential direction has to be specified (see definition of Space/Time Variation under Section 5.5.8). Presently only $180°$ loading condition is available and circumferential orders 0,1,2 and 3 are implemented. Also, it is not possible at this stage to apply FIT-value, CONTact and OFFSet cards with the AXISymmetry GENERAL card.

Examples of Use -

1. Request an axisymmetric elastic analysis of a pressure vessel.

```
**CASE
  TITLe  PRESSURE VESSEL - INTERNAL PRESSURE 30 PSIG
  AXISymmetry
  ELAStic
```

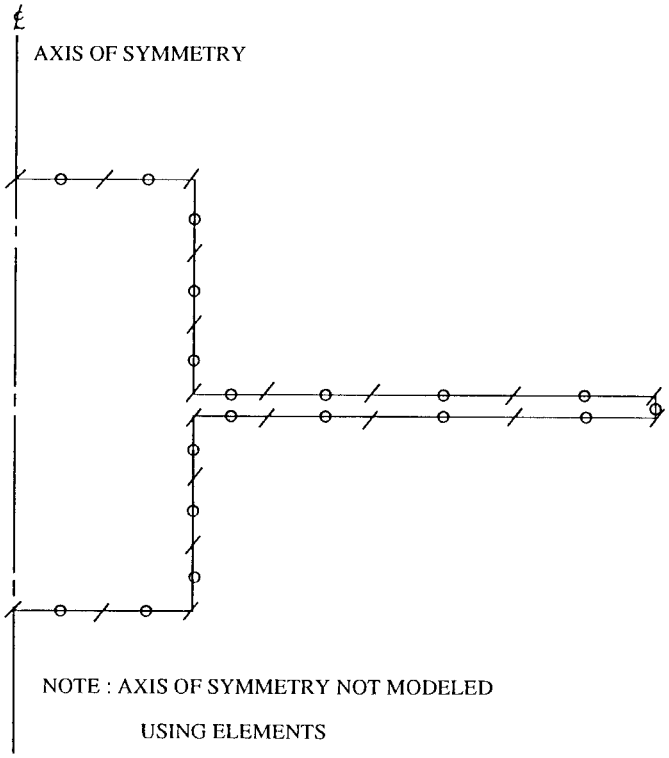


FIGURE FOR AXIS CARD

2. Model the bending of a round cantilever beam using generalized axisymmetry.

```
**CASE
  TITLe  CANTILEVER BEAM - CIRCULAR CROSS-SECTION
  AXISymmetry GENERAL
  ELAStic
```

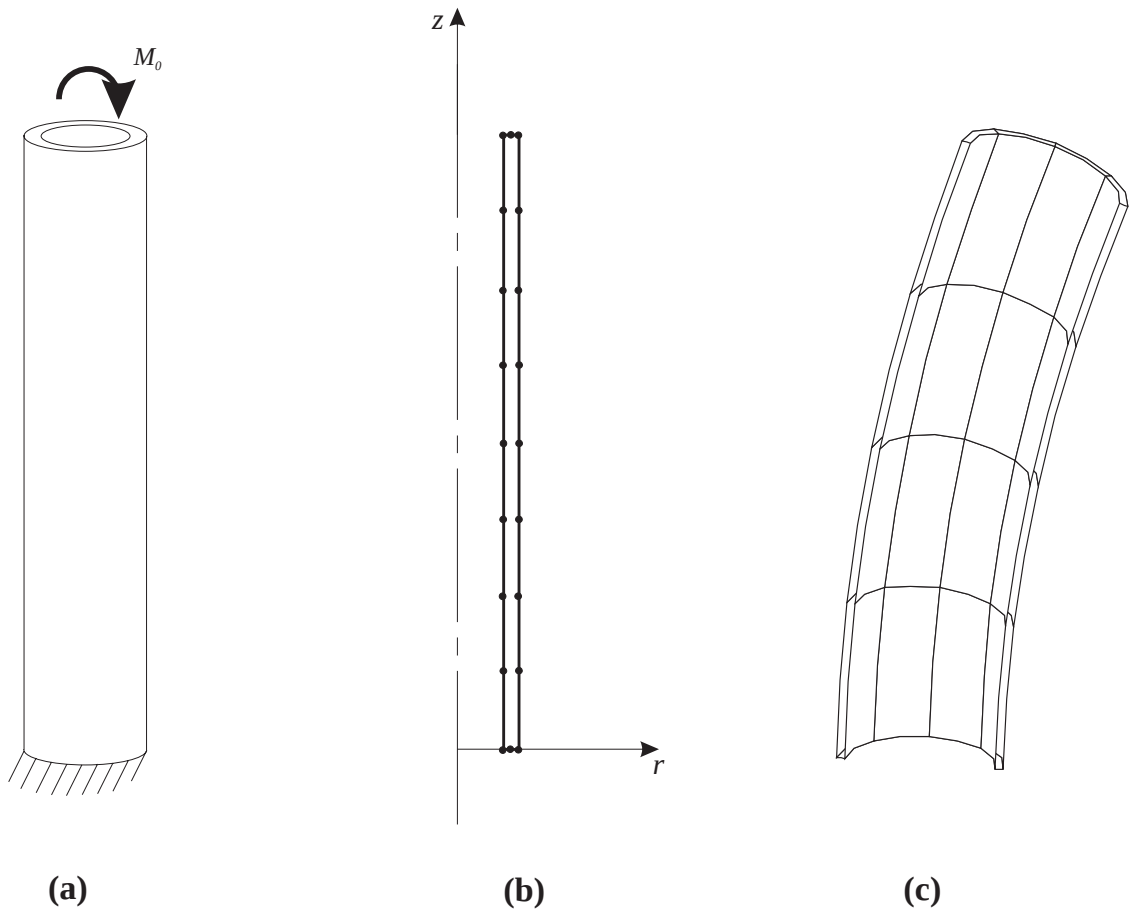


Figure for AXISymmetry GENE Card: (a) Cylinder With End-Moment
(b) Axisymmetric BE Model (c) Post-Processed Deformation

PLANe ETYPE

Status - OPTIONAL

Full Keyword - **PLANe**

Function - Identifies a two-dimensional problem. Default is a three-dimensional analysis.

Input Variables -

ETYPE (Alphanumeric) - OPTIONAL

Allowable values are STRE or STRA.

STRESS - specifies a plane stress problem.

STRAIN - specifies a plane strain problem.

Additional Information -

For problems other than that of stress analysis (e.g., heat conduction), only the first keyword PLANe needs to be specified.

If ETYPE is not specified, STRAIN is assumed.

Examples of Use -

1. Request a plane strain elastic analysis of a dam.

```
**CASE
  TITLe  KOYNA DAM - PLANE STRAIN ELASTIC ANALYSIS OF A DAM
  PLANe STRAIN
  ELAStic
```

2. Request a two-dimensional steady-state heat conduction analysis of a cylinder.

```
**CASE
  TITLe  CYLINDER - HEAT CONDUCTION
  PLANe
  HEAT
```

MIXEd

Status - OPTIONAL

Full Keyword - **`MIXEd`**

Function - specifies that the dimensionality is region dependent.

Input Variables - None

Additional Information -

If the MIXEd card is used then the dimensionality type must be specified in **GMR input for all regions.

Region Mixtures are only allowed in 2D analysis, i.e., only axisymmetry, plane stress, and plane strain regions can be defined under **GMR.

The MIXEd card is optional, even for MIXEd region problems. If analysis type modifiers (AXISymmetry, PLANe STRE, PLANe STRA) appear in each GMR, then GPBEST assumes this is a MIXEd region problem.

Three dimensional regions and Generalized Axisymmetric regions are not allowed in MIXEd problems.

Examples of Use -

1. Request heat conduction analysis of a finned tube which involves both axisymmetric and planar regions.

```
**CASE
  TITLe  FINNED TUBE - THERMAL ANALYSIS
  MIXEd
  HEAT STEADY
```

5.1.4 TYPE OF ANALYSIS

ACOUstic ATYPE

Status - OPTIONAL

Full Keyword - **`ACOUstic`**

Function - Identifies an acoustic analysis.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are:

PERIODIC - identifies a periodic (steady-state) acoustic analysis.

EIGENFREQUENCY - identifies an acoustic eigenfrequency analysis.

Additional Information for Periodic Acoustics-

A modified Burton/Miller formulation is used for exterior periodic acoustics. Exterior periodic acoustics is not available for 2D plane analysis.

The results output in a periodic analysis is controlled using the FREQuency-output, FREQuency SWEEP, NEUTral-file, and HISTory-quantities cards which are all documented in the **CASE input section.

In periodic acoustic analysis, boundary conditions and body forces are generally given as complex quantities with real and imaginary parts, see the LOAD COMPLEX card documented under **CASE input for details.

Periodic Acoustic quantities of pressure, intensity, energy density, and power are reported in both decimal and decibel units. Decibel units are reported with respect to reference quantities. These reference quantities may be defined using the DECIbel card documented under **CASE input. The default values are those commonly used for air. See the DECIbel card for details.

Boundary conditions in the form of nodal displacements or velocities may be input in a periodic acoustic analysis using the BCFIle input. See the BCFIle input card under **CASE input for details.

Additional Information for Acoustic Eigenfrequency analysis-

Acoustic Eigenfrequency analysis is not available for axisymmetric or general axisymmetric analysis.

The MAXImum-iterations card no longer controls the number of iterations. The algorithm currently used to extract eigenvalues is a complex, self-contained program, with an iteration process with multiple branches of execution. The iteration control is sophisticated and works best without user intervention. The user will have some control over the time of execution using the EIGEnvalue OUTPUT card or with the TOLErance card, however,

these commands are treated only as suggestions to the iteration control routine. The routine has the final decision as to how many iterations are required to produce a good solution.

Refer to the following cards documented under **CASE input for additional information on Acoustic Eigenfrequency analysis: EIGEnvalue OUTPUT, EIGEnvalue-option-1 SAMPLING, and EIGEnvalue-option-2 SHIFT.

Examples of Use -

1. Request a periodic acoustic analysis (direct boundary element formulation) with five frequencies.

```
**CASE
  TITLe  CAVE
  ACOUstic PERIODIC
  FREQuency RADPS 1.5 2.5 3.0 4.0 5.0
```

2. Request an acoustic eigenfrequency analysis.

```
**CASE
  TITLe  ACOUSTIC EIGENFREQUENCIES OF A TRUCK CAB
  ACOUstic EIGENFREQUENCY
  EIGEnvalue OUTPUT 20 10
  EIGEnvalue-option-1 SAMP
```

3. Request a periodic acoustic analysis with a frequency sweep of 20 Hz to 40 Hz at 2 Hz intervals with a history recording information of some sampling points:

```
**CASE
  TITLe  PULSATING SPHERE
  ACOUstic PERIODIC
  FREQuency SWEEP   20.0   40.0   2.0
  HISTory-quantities   1   2   5   6
  NEUTral-file
  ECHO-input
```

ACOUSTIC ANALYSIS USING PEAK ACOUSTIC QUANTITIES

The boundary element acoustic analysis in GPBEST is based on the assumption that the solution for pressure and velocity exist as a product of a function of space and a harmonic function of time.

$$p(x,t) = Re[p(x)e^{-i\omega t}] \tag{1a}$$

$$v_k(x,t) = Re[v_k(x)e^{-i\omega t}] \tag{1b}$$

where $p(x)$ and $v_k(x)$ are spatially dependent, complex quantities of pressure and velocity vector

$$p(x) = (p_r + ip_i) \tag{2a}$$

$$v_k(x) = ((v_k)_r + i(v_k)_i) \tag{2b}$$

Since the functional variation in time is known, GPBEST only determines the spatial variation of the complex acoustic quantities for an appropriate set of complex boundary conditions (and body forces) for a specified frequency.

Equation (1) can also be expressed in the following form:

$$p(x,t) = p_a \cos(\omega t + \phi_p) \tag{3a}$$

$$v_k(x,t) = (v_k)_a \cos(\omega t + \phi_v) \tag{3b}$$

where p_a and $(v_k)_a$ are the amplitude or peak values of pressure and velocity, respectively and ϕ_p and ϕ_v are their respective phase angles. The amplitude is the modulus of the complex quantities

$$p_a = (p_i^2 + p_r^2)^{1/2} \tag{4a}$$

$$(v_k)_a = \left((v_k)_r^2 + (v_k)_i^2\right)^{1/2} \tag{4b}$$

and the phase angles are

$$\phi_p = \tan^{-1}\left(\frac{-p_i}{p_r}\right) \tag{5a}$$

$$\phi_v = \tan^{-1}\left(\frac{-(v_k)_i}{(v_k)_r}\right) \tag{5b}$$

The instantaneous acoustic intensity vector is defined as

$$I_k(x_i,t) = p(x_i,t)v_k(x_i,t) \tag{6}$$

A time independent definition for intensity which is representative of the instantaneous intensity over a period is defined as

$$I_k(x_i) = Re\left(p(x_i)v_k^*(x_i)\right) \tag{7}$$

where $v_k^*(x)$ is the complex conjugate of the velocity vector defined in Eq. (2b). More specifically Eq. (7) can be written

$$I_k = p_r(v_k)_r + p_i(v_k)_i \tag{8}$$

Equation (8) is the intensity vector that is output in GPBEST. The magnitude of the intensity vector is defined as

$$I_m = (I_1^2 + I_2^2 + I_3^2)^{\frac{1}{2}}$$

The pressure amplitude and the intensity vector are output in both decimal and decibel output. These values are converted to sound pressure level, SPL, and intensity level, $(\mathrm{IL})_m$, using

$$\mathrm{SPL} = 20 \cdot \log_{10}\left(\frac{p_a}{p_{\mathrm{ref}}}\right) dB \tag{9}$$

$$(\mathrm{IL})_m = 10 \cdot \log_{10}\left(\frac{|I_m|}{I_{ref}}\right) dB \tag{10}$$

where p_{ref} and I_{ref} are the reference pressure and reference intensity, respectively. See the DECIbel card for more information on reference values. If the reference values are greater than the numerator in these equations, then a value of zero is reported.

The intensity level may be expressed as a vector $(IL)_k$ in which the components are defined as

$$(IL)_k = \frac{I_k}{I_m}(IL)_m$$

Energy Density E is defined

$$E = \frac{1}{2}\rho\left(\overline{v}^2 + p_a^2/\rho^2 c^2\right) \tag{11}$$

where $\overline{v}$ is the modulus of the magnitude of the velocity vector, i.e.,

$$\overline{v} = \left[\left((v_x)_r^2 + (v_y)_r^2 + (v_z)_r^2\right) + \left((v_x)_i^2 + (v_y)_i^2 + (v_z)_i^2\right)\right]^{1/2}.$$

Energy Density can be expressed in decibels as sound energy decibel level (SEDL)

$$\mathrm{SEDL} = 10 \cdot \log_{10}\left(\frac{E}{E_{ref}}\right) dB \tag{12}$$

where E_{ref} is the reference energy density. See the DECIbel card for details.

Radiated Power or Power Output $\mathrm{W_o}$ is defined as

$$W_o = \int_S Re(pv_n^*)dS \tag{13}$$

where S represents a surface, v_n is the velocity normal to this surface, and the superscript indicates the complex conjugate. More specifically

$$Re(pv_n^*) = (p)_r(v_n)_r + (p)_i(v_n)_i \tag{14}$$

This quantity can be positive or negative.

Applied Power $\mathbf{W_i}$ is defined as

$$W_i = \rho c \int_S (v_n)_a^2 dS \tag{15}$$

where $(v_n)_a$ is the amplitude of the velocity normal to the surface and S represents a surface. Only elements with a non-zero velocity boundary condition or a pressure boundary condition will contribute. This value is assumed to be zero for elements on a fluid-fluid interface.

The radiated power and the applied power are reported in the GPBEST result file for every individual boundary element and sampling surface element. The power output and the applied power are summed over elements and the total for each region, for each sampling surface, and for the entire boundary surface are output for each frequency in the GPBEST result file and in the RADIATION.DAT file.

NOTE: Radiated acoustic power is calculated for each sampling surface. Since the sign of $Re(pv_n^*)$ is dependent on the orientation of the outward normal, the normal of each element of the sampling surface must be consistent with all the other elements in the same sampling surface. The outward normal of an element is defined according to the right-hand-rule with respect to the connectivity.
The radiated power and the applied power are also output in decibels, as sound power level, (SWL_o)

$$(\mathrm{SWL}_o) = 10 \cdot \log_{10} \left(\frac{|W_o|}{W_{ref}} \right) dB \tag{16}$$

$$(\mathrm{SWL}_i) = 10 \cdot \log_{10} \left(\frac{|W_i|}{W_{ref}} \right) dB \tag{17}$$

where W_{ref} is the reference power (see DECIbel card for details). If the reference value is greater than the numerator in the power equations above, then a value of zero is reported for that quantity.,

The Acoustic Radiation Efficiency σ is defined as the total power output over the total applied power

$$\sigma = \frac{W_o^{\mathrm{Total}}}{W_i^{\mathrm{Total}}} \tag{18}$$

The total power output W_o^{Total} is defined as

$$W_o^{\mathrm{Total}} = \int_{S_{\mathrm{Total}}} Re(pv_n^*) dS \tag{19}$$

and the total applied power W_i^{Total} is defined as

$$W_i^{\mathrm{Total}} = \rho c \int_{S_{\mathrm{Total}}} (v_n)_a^2 dS \tag{20}$$

where S_{Total} is the entire boundary (i.e., the sum of all boundary elements excluding the fluid-fluid interface elements).

ACOUSTIC ANALYSIS USING EFFECTIVE ACOUSTIC QUANTITIES

Effective quantities or root-mean-square quantities are widely used particularly in experimental acoustics.

Effective values for pressure, velocity, intensity and energy density are defined as

$$p_{eff} = \left[\frac{1}{T}\int_o^T p^2(x,t)dt\right]^{1/2} = \frac{1}{\sqrt{2}}[p_r^2 + p_i^2]^{1/2} \tag{21}$$

$$(v_k)_{eff} = \left[\frac{1}{T}\int_o^T v_k^2(x,t)dt\right]^{1/2} = \frac{1}{\sqrt{2}}[(v_k)_r^2 + (v_k)_i^2]^{1/2} \tag{22}$$

$$(I_k)_{eff} = \frac{1}{T}\int_o^T p(x,t)v_k(x,t)dt = \frac{1}{2}[p_r(v_k)_r + p_i(v_k)_i] \tag{23}$$

$$E_{eff} = \frac{1}{T}\int_o^T \frac{1}{2}\rho\left(\overline{v}^2(x,t) + \frac{p(x,t)}{\rho^2 c^2}\right)dt = \frac{1}{4}\rho\left(\overline{v}^2 + \frac{p_a^2}{\rho^2 c^2}\right) \tag{24}$$

where $\overline{v}(x,t)$ is the magnitude of the velocity vector, and T is the period of one complete cycle $(T = 2\pi/\omega)$.

We now introduce definitions for complex effective values of pressure and velocity in terms of the complex peak values

$$p_{r,eff} + ip_{i,eff} = \frac{1}{\sqrt{2}}(p_r + ip_i) \tag{25a}$$

$$(v_k)_{r,eff} + i(v_k)_{i,eff} = \frac{1}{\sqrt{2}}\left((v_k)_r + i(v_k)_i\right) \tag{25b}$$

Substituting Eq. (15) into Eq. (21) through Eq. (24) yields

$$p_{eff} = \left[p_{r,eff}^2 + p_{i,eff}^2\right]^{1/2} \tag{26}$$

$$(v_k)_{eff} = \left[(v_k)_{r,eff}^2 + (v_k)_{i,eff}^2\right]^{\frac{1}{2}} \tag{27}$$

$$(I_k)_{eff} = p_{r,eff}(v_k)_{r,eff} + p_{i,eff}(v_k)_{i,eff} \tag{28}$$

$$E_{eff} = \frac{1}{2}\rho\left(\overline{v}_{eff}^2 + p_{eff}^2/\rho^2 c^2\right) \tag{29}$$

If we compare Eq. (4a) with Eq. (26), Eq. (4b) with Eq. (27), Eq. (8) with Eq. (28) and Eq. (11) with Eq., (29), we see these equations have the exact same form, the only difference is the former equations are expressions containing maximum (or peak) values and the later equations are expressions containing effective (or root-mean-square) values. Similar relations hold for the power equations. Hence, the acoustic quantities used in a GPBEST periodic acoustic analysis can represent either the maximum (or peak) quantities or the effective quantities. Boundary conditions and body forces of all quantities input, may be specified as complex values of either the maximum values or the effective values of those quantities. All results reported by GPBEST will then be output consitient with the type of values that were input. The user, however, must be consistent and specify all input using the same type of values (maximum or effective values) and interpret results in the same manner.

CONSolidation ITYPE JTYPE

Status - OPTIONAL

Full Keyword - **CONSolidation**

Function - Identifies a consolidation (or poroelastic) analysis.

Input Variables -

ITYPE (Alphanumeric) - OPTIONAL

Allowable values are STEA, and TRAN

Default is steady-state analysis.

STEADY - Identifies a steady-state (drained) analysis.

TRANSIENT - Identifies a transient analysis.

JTYPE (Alphanumeric) - OPTIONAL

Allowable values are ELAS, and PLAS

Default is linear analysis.

ELASTIC - Identifies linear analysis.

PLASTIC - Identifies a nonlinear analysis.

Additional Information -

Displacements, tractions, pore pressure and flux are determined at each time step.

Refer to the following cards documented under **CASE input for additional information: TIMEs-ouput and TIMEs STEP.

Examples of Use -

1. Perform a consolidation analysis of the soil under a strip footing. Include ten time steps for a total time of 5.0.

```
**CASE
  TITLe  CONSOLIDATION UNDER A STRIP FOOTING
  PLANe STRAIN
  CONSolidation TRANSIENT
  TIMEs STEP  10  0.5
```

2. Perform a nonlinear consolidation analysis of the soil under a strip footing.

```
**CASE
  TITLe  NONLINEAR CONSOLIDATION UNDER A STRIP FOOTING
  PLANe STRAIN
  CONSolidation TRANSIENT
  TIMEs STEP  10  0.5
```

CTHErmal ITYPE JTYPE

Status - OPTIONAL

Full Keyword - **`CTHErmal`**

Function - Identifies a concurrent thermoelastic analysis.

Input Variables -

ITYPE (Alphanumeric) - OPTIONAL

Allowable values are STEA and TRAN.

Default is steady-state analysis.

STEADY - Identifies a steady-state analysis.

TRANSIENT - Identifies a transient analysis.

JTYPE (Alphanumeric) - OPTIONAL

Allowable values are ELAS, and PLAS

Default is linear analysis.

ELASTIC - Identifies linear analysis.

PLASTIC - Identifies a nonlinear analysis.

Additional Information -

Displacements, tractions, temperatures and heat flux are determined at each time step. The problems of heat conduction and thermoelasticity are solved simultaneously.

Refer to the following cards documented under **CASE input for additional information: TIMEs-ouput and TIMEs STEP.

Examples of Use -

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

```
**CASE
  TITLe  PRESSURE VESSEL AG-9 THERMAL LOAD
  AXISymmetry
  CTHErmal  TRANS
  TIMEs STEP  14  0.05
```

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

```
**CASE
  TITLe  TURBINE BLADE A7311 - THERMOELASTIC
  CTHErmal STEADY
```

3. Determine the steady-state thermoplastic response of a turbine blade.

```
**CASE
  TITLe  TURBINE BLADE AT311 - THERMOPLASTIC
  CTHErmal STEADY PLASTIC
```

DIFFusion ITYPE

Status - OPTIONAL

Full Keyword - **DIFFusion**

Function - Identifies a diffusion analysis.

Input Variables -

ITYPE (Alphanumeric) - OPTIONAL

Allowable value is STEA and TRAN.

Default is steady-state analysis.

STEADY - Identifies a steady-state potential flow analysis.

TRANSIENT - Identifies a transient analysis.

Additional Information -

Refer to the following cards documented under **CASE input for additional information: TIMEs-ouput and TIMEs STEP.

NONE

Examples of Use -

1. Determine the transient temperature distribution in a printed circuit board by using eight time steps of duration 0.01.

```
**CASE
  TITLe  PRINTED CIRCUIT BOARD - TRANSIENT CONDITION
  DIFFusion  TRANSIENT
  TIMEs STEP  8  0.01
```

2. Request a steady-state potential flow analysis of an incompressible, inviscid fluid.

```
**CASE
  TITLe  POTENTIAL FLOW OVER AN AIRFOIL
  DIFFusion STEADY
```

DISTortion

Status - OPTIONAL

Full Keyword - **`DISTortion`**

Function - Identifies a thermal distortion analysis.

Input Variables - NONE

Additional Information -

Thermal distortion analysis is run by starting with a known boundary temperature and flux solution, which must be determined from a previous HEAT analysis. That HEAT analysis generates a results file FT121, containing boundary temperature and flux. FT121 must then be available for the DISTortion analysis.

Stress results are not calculated when holes are present in the model. This is due to the approximate nature of the DISTortion analysis relating to hole element contributions.

DISTortion is only available for steady-state analysis. For transient thermal distortion analysis, concurrent thermoelasticity (CTHErmal) must be used.

For steady-state thermoelastic analysis, DISTortion is more effcient than CTHErmal STEA when running large problems. If the hole elements are not present, it will give the same answer.

Refer to the following cards documented under **CASE input for additional information: TIMEs-ouput and TIMEs STEP.

Examples of Use -

1. Request a steady-state thermal analysis of a casting mold.

```
**CASE
  TITLe CASTING MOLD - THERMAL ANALYSIS
  HEAT
  RESTart WRITE
```

2. Request a subsequent thermal distortion analysis of the same casting mold.

```
**CASE
  TITLe CASTING MOLD - THERMAL DISTORTION
  DISTortion
```

ELAStic

Status - OPTIONAL

Full Keyword - **ELAStic**

Function - Identifies an elastic analysis.

Input Variables - NONE

Additional Information -

Elastic analysis is the default analysis type. Therefore, if an analysis type is not specified, an elastic analysis will be performed.

Examples of Use -

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

```
**CASE
  TITLe  REACTOR PRESSURE VESSEL - LOAD CASE 2B
  AXISymmetry
  ELAStic
  TIME   1.0   2.0   3.0
```

FREE-vibration

Status - OPTIONAL

Full Keyword - **`FREE-vibration`**

Function - Identifies a free-vibration analysis.

Input Variables - NONE

Additional Information -

Free vibration analysis is not available for axisymmetric or generalized axisymmetric problems.

The MAXImum-iterations card no longer controls the number of iterations. The algorithm currently used to extract eigenvalues is a complex, self-contained program, with an iteration process with multiple branches of execution. The iteration control is sophisticated and works best without user intervention. The user will have some control over the time of execution using the EIGEnvalue OUTPUT card or with the TOLErance card, however, these commands are treated only as suggestions to the iteration control routine. The routine has the final decision as to how many iterations are required to produce a good solution.

Refer to the following cards documented under **CASE input for additional information: EIGEnvalue OUTPUT, EIGEnvalue-option-1 SAMPLING, and EIGEnvalue-option-2 SHIFT.

Examples of Use -

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

```
**CASE
  TITLe  TURBINE BLADE A7311 - MODE SHAPES
  FREE-vibration
  EIGEnvalue OUTPUT 20 10
  EIGEnvalue-option-1 SAMP
```

FORCed-vibration W1 W2 W3 ... WN

Status - OPTIONAL

Full Keyword - **`FORCed-vibration`**

Function - Identifies a steady-state forced vibration analysis of solids.

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
Defines additional frequecies at which output is required.

Additional Information -

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

In periodic analysis, boundary conditions and body forces are generally given as complex quantities with a real and imaginary part. See the LOAD COMPLEX card documented under **CASE input for details.

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-vibration 15.  30.  60.
```

HEAT ITYPE

Status - OPTIONAL

Full Keyword - **HEAT**

Function - Identifies a heat conduction analysis.

Input Variables -

ITYPE Alphanumeric - OPTIONAL

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

STEADY - Identifies a steady-state analysis.

TRANSIENT - Identifies transient analysis.

Additional Information -

HOLE elements may be included in a three-dimensional HEAT analysis.

Refer to the following cards under **CASE input for additional information: TIMEs-output and TIMEs STEP.

Examples of Use -

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

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

2. Run a 2-D transient problem with 15 time steps of 2 seconds each.

```
**CASE
  TITLe  TRANSIENT HEAT TRANSFER IN A CIRCLE
  PLANe
  HEAT TRANSIENT
  TIMEs STEP 15 2.0
  TIMEs VARIATION CONSTANT
```

PLASticity

Status - OPTIONAL

Full Keyword - **PLASticity**

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

Input Variables - NONE

Additional Information -

If this card is input, an appropriate volume cell discretization must be defined in the geometry input section.

Refer to the following cards under **CASE input for additional information: TOLErance-convergence, MAXImum-iterations, ITERative, and NEWTon-raphson.

Examples of Use -

1. Perform elastoplastic analysis of an axisymmetric pressure vessel utilizing the Newton-Raphson plasticity algorithm.

```
**CASE
  TITLe  PRESSURE VESSEL - PLASTICITY
  TIMEs-output  1.0  2.0  3.0  4.0
  AXISymmetry
  PLASticity
  NEWTon-raphson
```

SACOustic ATYPE BTYPE CTYPE

Status - OPTIONAL (applicable only for 3-D structural-acoustics problems)

Full Keyword - **`SACOustic`**

Function - Identifies a 3-D structural-acoustic analysis.

Input Variables -

ATYPE (alpha) - REQUIRED (Only value currently available is PERIODIC)

BTYPE (alpha) - REQUIRED (Allowable value is either DIRECT or INDIRECT)

CTYPE (alpha) - REQUIRED (Available options are ONEWAY (in which the structural deformation is unaffected by the acoustic pressure), and COUPLED (in which the structural deformation is influenced by the fluid inertia))

Additional Information -

Currently, the frequency information can only be specified through the FREQuency SWEEP option.

The data file must contain one FE GMR (in a **GMR block) along with one BE GMR (for DBEM and IBEM) or multiple BE GMRs (for DBEM only).

The **BCSET and **BODY (if present) should have complex valued input. The LOAD-bc COMPLEX option is assumed even if the card is not included.

Auxiliary Files for Structural Acoustic Analysis

The following chart outlines the output files that can be created (in addition to the standard result file) in a structural acoustic analysis.

	DBEM	IBEM
1. Single frequency	(a) NEUTRAL.OUT.STR, NEUTRAL.OUT.FLU } Paired neutral files* (b) Radiation file (c) Sweep file	(a) NEUTRAL.OUT.STR, NEUTRAL.OUT.FLU } Paired neutral files* (b) Sweep file
2. Multiple frequencies	(a) NEUTRAL.OUT.STR, NEUTRAL.OUT.FLU } Paired neutral files* (1st frequency only) (b) Radiation file (for all frequencies) (c) Sweep file (for all frequencies)	(a) NEUTRAL.OUT.STR, NEUTRAL.OUT.FLU } Paired neutral files* (1st frequency only) (b) Sweep file (for all frequencies)

*The paired files are:

NEUTRAL.OUT.STR for FE structural solution, and
NEUTRAL.OUT.FLU for BE acoustic solution.

The neutral file and sweep files are created by including the NEUTral-file and HISTory-quantities card, respectively, in **CASE input (see section 5.1.10).

The RADIATION.OUT file is automatically created in a direct boundary element (DBEM) analysis (see the ACOUstic card in this section for a description of the RADIATION.OUT file).

Case Control

Examples of Use -

1. Initiate a one-way, coupled structural-acoustics problem using indirect boundary element formulation being run at 1 Hz.

```
**CASE
  TITLe  P1: A ONE WAY COUPLED STRUCT. ACOU. MODEL
  SACOustic    PERIODIC    INDIRECT    ONEWAY
  FREQuency SWEEP HERTZ 1.0
  LOAD COMPLEX
  ECHO-input
  NEUTral-file
```

2. Define a fully coupled structural acoustic problem using the direct boundary element formulation with a frequency sweep from 100 Hz to 200 Hz in 5 Hz steps.

```
**CASE
  TITLe  FULLY COUPLED STRUCTURAL ACOUSTIC PROBLEM
  SACOustic    PERIODIC    DIRECT    COUPLED
  FREQuency SWEEP HERTZ  100.0  200.0  5.0
  HISTory-quantities  1  2
  LOAD COMPLEX
  ECHO-input
  NEUTral-file
```

5.1.5 ANALYSIS TYPE MODIFIERS

FRACture

Status - OPTIONAL

Full Keyword - **FRACture**

Function - Identifies fracture mechanics analysis. This card is only applicable to ELAStic and CTHErmal analyses.

Input Variables - NONE

Additional Information - NONE

Examples of Use -

1. Elastic analysis of a rectangular block with through crack.

```
**CASE
  TITLe  RECTANGULAR THROUGH CRACK
  ELAStic
  FRACture
  TIMEs-output  1.0 2.0
```

5.1.6 TIMES FOR OUTPUT

TIMEs-output T1 T2 T3 ... TN

Status - OPTIONAL

Full Keyword - **`TIMEs-output`**

Function - Identifies times at which output is desired.

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 TIMEs-output. A maximum of twenty output times may be selected.

If the TIMEs-output card is not input in a static or quasistatic analysis, then results are output at an assumed time of 1.0.

If the TIMEs-output card is not input in a transient analysis, then results are output at every incremental step (see TIMEs STEP card).

In a transient analysis, the times on the time card should be coincident with the end of a load increment.

In a periodic analysis, results are output at the frequencies specified on the analysis card (see ACOUstic or FORCed-vibration card).

This card defines a pseudo time step in plastic analysis.

This card has no significance for the SACOustic analysis type.

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
  ELAStic
  TIMEs-output 1.0 2.5 6.0
```

TIMEs STEP NSTEP DELTA

Status - OPTIONAL (required for transient analysis algorithms)

Full Keyword - **`TIMEs STEP`**

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

Input Variables -

NSTEP (Integer) - REQUIRED

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

DELTA (Real) - REQUIRED

Defines the size of the time step.

Additional Information -

This card may be included as many times as desired with different time step information on each successive card. The primary purpose of including multiple TIMEs STEP cards is to allow the user to use small time steps at the start of the analysis so that short term (start up) effects can be accurately captured, and then switch to a larger time step to complete the analysis in an efficient manner. Usually, the time step increment will increase with successive TIMEs STEP cards. However, if the boundary condition values suddenly change at a later time, it may be desirable to reduce the time step interval in order to capture the short term effect associated with a change in the boundary condition values.

It is important to note that the system equations must be resolved every time the time step increment is changed. This cost, however, can be offset by the time saved in using a larger time step increment. It is therefore necessary to use good judgement when variable time steps are utilized.

Unlike the finite element method, the value of the time step is not critical to the stability of the transient algorithm. Two effects influence the rate of diffusion in a body and both effects must be considered when choosing time step increments for the analysis. The first effect is the rate of heat flux entering and leaving each region. This rate is dependent on the surface coefficient of heat transfer, also known as the convective heat transfer coefficient, for the outer surface and the convective coefficient between the regions. The second consideration is the heat diffusivity of each region. The user may choose a time step as large or as small as he desires depending on whether long or short term effects are important. The user must keep in mind that the temperature and flux variables are assumed to be constant (or linear; see the TIMEs VARI card) in time and that the solution will be in error if too large a time increment is chosen. A user should conduct a convergence study until experience has been gained in choosing a time increment for **GPBEST** .

The user should also realize that **GPBEST** is a boundary method and the default quantities of temperatures that are reported in the **GPBEST** results are for nodes on the surface of the body. This can be deceiving since the boundary temperatures can reach their steady-state values very quickly while the interior of the body is still in transition. The user is encouraged to input sampling points and sampling surfaces in the interior of the body in order to monitor the temperatures in the interior of the body. This will result in a better understanding of the entire diffusive process.

Examples of Use -

1. Conduct a transient thermoelastic analysis of a spherical tank using a constant time step.

```
**CASE
  TITLe  SPHERICAL TANK - SUDDEN PRESSURIZATION
  CTHErmal TRANSIENT
  TIMEs STEP   20  0.01
```

2. Conduct a transient thermoelastic analysis of a spherical tank using variable time stepping (three different time step values).

```
**CASE
  TITLe  SPHERICAL TANK - SUDDEN PRESSURIZATION
  CTHErmal TRANSIENT
  TIMEs STEP   5  0.002
  TIMEs STEP   4  0.01
  TIMEs STEP   3  0.05
```

Note that in both of these examples, the analysis ends at time=0.2. However, in example 2, smaller time steps are used at the start of the analysis in order to accurately capture the short term (start up) effects. Larger time steps are then utilized to complete the analysis in an efficient manner.

TIMEs VARI CTYPE

Status - OPTIONAL (REQUIRED for transient analysis algorithms)

Full Keyword - **`TIMEs VARIATION`**

Function - Identifies type of temporal variation of displacements/temperatures and tractions/fluxes.

Input Variables -

CONSTANT - constant variation of field quantities in time.

LINEAR - linear variation of field quantities in time.

Additional Information -

Linear time variation is sensitive to the value of the TIMEs STEP increment that is used. Furthermore, the linear time variation does not appear to greatly improve the solution. It is therefore recommended that the CONSTANT time variation be used.

A CONSTANT variation is available for heat conduction, thermoelastic and consolidation analyses.

LINEAR is available for transient diffusion and heat transfer analyses.

Examples of Use -

1. Perform a heat conduction analysis of a spherical tank using a linear time variation of field variables.

```
**CASE
  TITLe  SPHERICAL TANK - SUDDEN PRESSURIZATION
  HEAT TRANSIENT
  TIMEs STEP  10  0.01
  TIMEs VARIATION LINEAR
```

FREQuency-output ATYPE F1 F2 ... FN

Status - OPTIONAL

Full Keyword - **FREQuency-output**

Function - Used for periodic analysis to define the units of frequency used in the analysis and to define at which frequency full output of results will be output.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are -

HERTZ - uses cycles per second (Hertz) as the unit of frequency

RADPS - uses radians per second as the unit of frequency

F1 (Real) - REQUIRED

Defines the sinusoidal frequency at which the first results will be output.

F2 ... FN (Real) - OPTIONAL

Defines additional frequencies at which results will be output.

Additional Information -

A maximum of twenty discrete frequency values may be declared with the above syntax.

If a specified frequency is less than 0.000016 Hertz (or 0.0001 rad/sec) then it is reset to 0.000016 Hertz (or 0.0001 rad/sec).

This card can be used by itself or in combination with the FREQuency SWEEP card. A frequency sweep run may contain a large number of frequencies and the amount of results that are generated would be impractical to output. Therefore, a limited amount of information is output to the RADIATION.OUT file and the HISTORY.OUT file during a FREQuency SWEEP analysis. However, a complete set of results can be output for selected frequencies using the FREQuency-output card. These results are output to the GPBEST result file, the NEUTRAL.OUT, the RADIATION.OUT, and the HISTORY.OUT file.

Either the FREQuency-output or FREQuency SWEEP cards (or both) must be included in a periodic analysis. If both are input, then the units specified for frequency must be the same on both cards. The units specified on these cards will be the units used for the result output.

Examples of Use -

1. Request a periodic acoustic analysis with five frequencies.

```
**CASE
  TITLe  PERIODIC ACOUSTIC ANALYSIS
  ACOUstic PERIODIC
  FREQuency HERTZ 1.5 2.5 3.0 4.0 5.0
```

FREQuency SWEEP ATYPE FS FE FI

Status - OPTIONAL

Full Keyword - **FREQuency SWEEP**

Function - Used in periodic analysis to define the units for frequency used in the analysis and to indicate a frequency range and interval for a periodic analysis in which results will be generated and output to the RADIATION.OUT file (and optionally to the HISTORY.OUT file).

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are -

HERTZ - uses cycles per second (Hertz) as the unit of frequency

RADPS - uses radians per second as the unit of frequency

FS (Real) - REQUIRED

Defines the starting sinusoidal frequency at which the analysis is run.

FE (Real) - REQUIRED

Defines the ending sinusoidal frequency at which the analysis is to be performed.

FI (Real) - OPTIONAL

Defines the frequency step used to increment the frequency from FS to FE.

Additional Information -

If the starting frequency FS is less than 0.000016 Hertz (or 0.0001 rad/sec) then it is reset to 0.000016 Hertz (or 0.0001 rad/sec).

If FI is not defined, the default value is 1.0.

Results are calculated and output starting with the first frequency FS, the next frequency output is (FS + FI), followed by (FS + 2FI), etc.

This card can be used by itself or in combination with the FREQuency-output card. A frequency sweep run may contain a large number of frequencies and the amount of results that are generated would be impractical to output. Therefore, a limited amount of information is output to the RADIATION.OUT file and the HISTORY.OUT file during a FREQuency SWEEP analysis. However, a complete set of results can be output for selected frequencies using the FREQuency-output card. These results are output to the GPBEST result file, the NEUTRAL.OUT, the RADIATION.OUT, and the HISTORY.OUT file.

The FREQuency-output card or the FREQuency SWEEP card (or both) must be included in a periodic analysis. If both are included, then the units specified for

frequency must be the same on both cards. The units specified on these cards will be the units used for the result output.

If this card is input without the FREQuency-output card, then complete results are output to the result file and NEUTRAL.OUT file only for the starting frequency. This is done to prevent the size of these files from becoming very large when hundreds of frequencies are calculated during a sweep. From the information learned from the sweep analysis, the user can always rerun the analysis at the specified frequencies of interest by using the FREQuency-output card independent of this card.

For additional information on requesting history information, see the HISTory-quantities card under **CASE input, and the HISTory-points card under **GMR input.

Examples of Use -

1. Request a sweep from 100 Hz to 200 Hz in steps of 20 Hz.

```
**CASE
  TITLe  FREQUENCY SWEEP
  ACOUstic PERIODIC
  FREQuency  SWEEP  HERTZ  100.0  200.0  20.0
```

5.1.7 GEOMETRIC AND LOADING SYMMETRY CONTROL

SYMMetry STYPE AXIS1 AXIS2 AXIS3

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 (2D,3D and Axisymmetry)

QUAR - Quarter symmetry (2D and 3D only)

OCTA - Octal symmetry (3D only)

AXIS1 (Alphanumeric) - REQUIRED for STYPE= HALF, QUAR and OCTA

AXIS2 (Alphanumeric) - REQUIRED for STYPE= QUAR and OCTA

AXIS3 (Alphanumeric) - REQUIRED for STYPE= OCTA

Allowable variables of AXIS1, AXIS2, AXIS3 are X, Y and/or Z.

Additional Information -

The plane(s) of symmetry is(are) defined by specifing the axis(es) perpendicular to the plane(s) of symmetry, i.e. X (meaning X=0) defines the Y-Z plane.

Only half symmetry about the $r-\theta$ plane is allowed in axisymmetric analysis and in analysis in which the data is input using cylindrical COORdinate-system. **GPBEST** accepts either AXIS1=Z or AXIS1=Y as a definition for input for specification of the $r-\theta$ plane.

If AXIS1, AXIS2, AXIS3 are not specified, the following default cases are assumed.

SYMMetry HALF assumes symmetry about Y-Z plane

($r-\theta$ plane in axisymmetry)

SYMMetry QUAR assumes symmetry about X-Z and Y-Z plane

SYMMetry OCTA assumes symmetry about X-Y, X-Z and Y-Z plane

If the SYMMetry card is used the plane of symmetry should not be discretized, i.e. boundary elements should not appear on the plane of symmetry (see the figure associated with this keyword).

The use of the SYMMetry card automatically invokes the condition of zero displacement (and zero flux) on and perpendicular to the plane of symmetry. Since rigid body motion is prevented in the direction perpendicular to an axis of symmetry, the user

is not required to explicitly fix the displacement in this direction for the purpose of preventing rigid body motion.

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

For a structural acoustics problem, the SYMMetry card is applicable for the acoustic GMR. The symmetry condition over the structural GMR must be imposed through the DISPlacement-fem card in **BCSET in the proper directions.

Symmetry Guidelines -

The user should be aware of some of the benefits and consequences of using the SYMMetry options. This will allow an intelligent use of this option.

If the user wishes to model a plate with an internal hole, a 2-D model is sufficient. The geometry appears in Figure 5.1.7.1. The loading is simple tension along the X-axis.

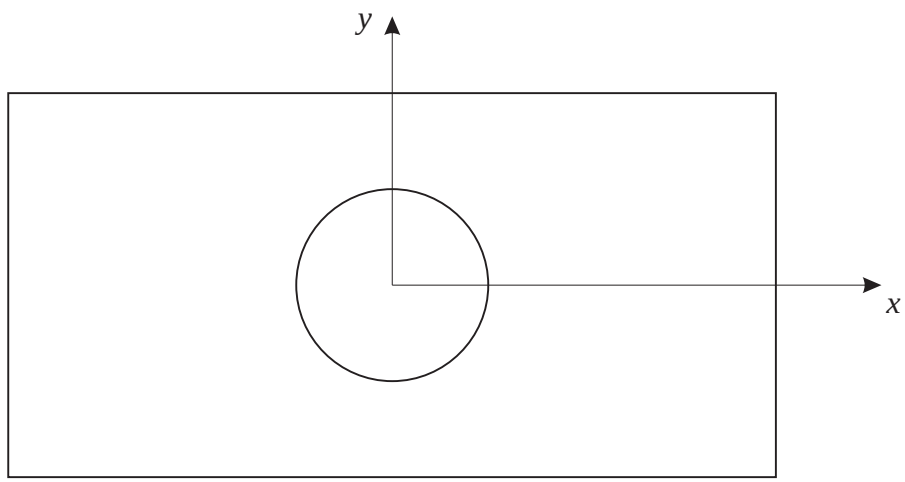


Figure 5.1.7.1 2-D Plate With a Hole

Of course, the entire problem could be modeled. However, since symmetry is present about the X-Z plane and the Y-Z plane, symmetry can be exploited to reduce the modeling effort. In addition, symmetry eases the post-processing of **GPBEST** results since only one section needs to be considered. All other quadrants will exhibit identical results and need not be investigated.

There are basically two methods of incorporating symmetry: 1) using conventional symmetry and 2) using the SYMMetry card. In both cases, only a quarter-model is constructed. These two methods will be described in detail to highlight their differences.

1) Conventional Symmetry

Conventional symmetry is applied purely using boundary conditions along the planes of symmetry. The SYMMetry card is not used in Conventional Symmetry. Boundary elements must be placed along the planes of symmetry (i.e., along the axes). Figure

5.1.7.2 illustrates a quarter-symmetric boundary element model using conventional symmetry. The symmetry conditions on the axes are imposed using **BCSET input. These are typically referred to as "roller-type boundary conditions."

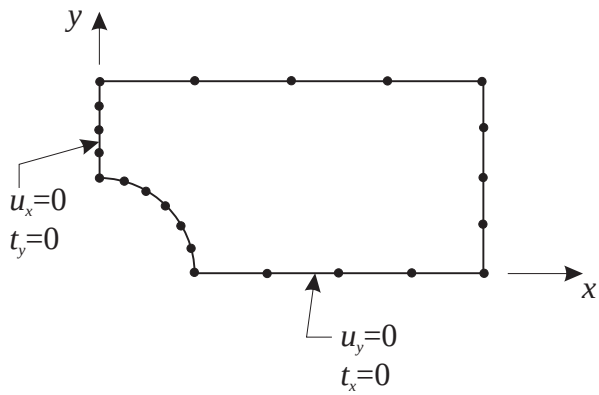


Figure 5.1.7.2 Conventional Symmetry (Roller Boundary Conditions)

2) Symmetry Using the SYMMetry Card

The SYMMetry card in **CASE input permits the user to model one-quarter of the plate with no elements along the planes of symmetry (See Figure 5.1.7.3).

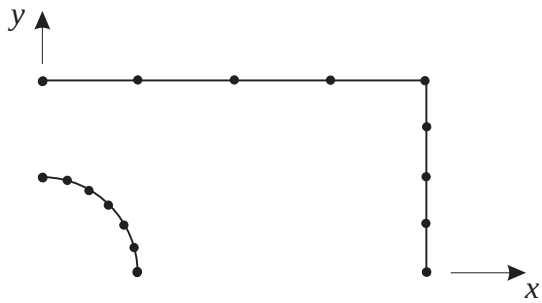


Figure 5.1.7.3 Symmetry Using the SYMMetry Option

However, the user should be aware of what the SYMMetry card does internally within **GPBEST** . While **GPBEST** is running, a model of the remaining three quadrants is being generated internally. Thus, as far as the solution phase of **GPBEST** is concerned, Figure 5.1.7.4 shows the problem **GPBEST** is solving internally. This may significantly add to the solution time.

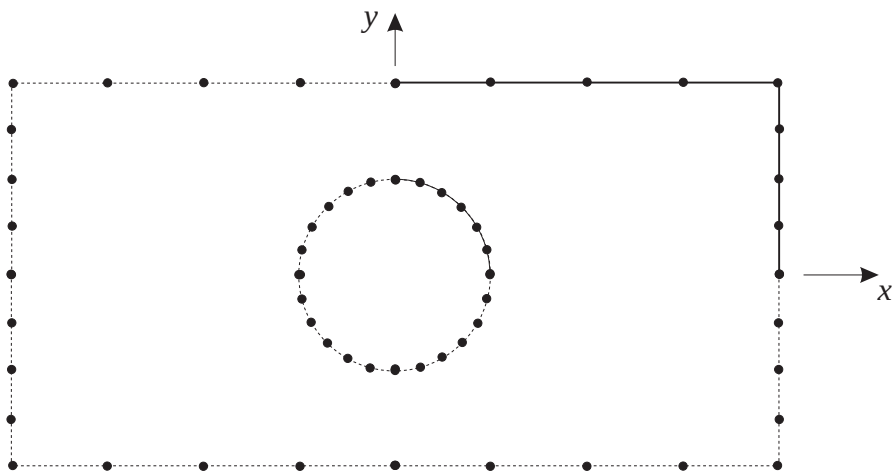


Figure 5.1.7.4 SYMMetry Option: Problem Actually Solved by GPBEST

Comparing the Two Forms of Symmetry

Since the user can choose which symmetry to use, the factors favoring one method over another must be considered. Conventional symmetry requires the user to take the additional steps of modeling elements along the planes of symmetry as well as applying the appropriate boundary conditions on those elements. The SYMMetry option uses a simpler model. However, internally, **GPBEST** solves a much larger problem.

Another difference which is more subtle is that the elements on the planes of symmetry in the Conventional Symmetry model actually constrain displacements and tractions to conform to a quadratic variation. If the stress varies greatly along the Y-axis, as it does in the case of a plate with a hole, then the quadratic functional variation may constrain the solution near the stress concentration (See Figure 5.1.7.5). In using conventional symmetry, the user must model the symmetric edges using enough elements to capture the reactions. However, in the case of SYMMetry Symmetry, no elements appear along the axes so the functional variation is not constrained there.

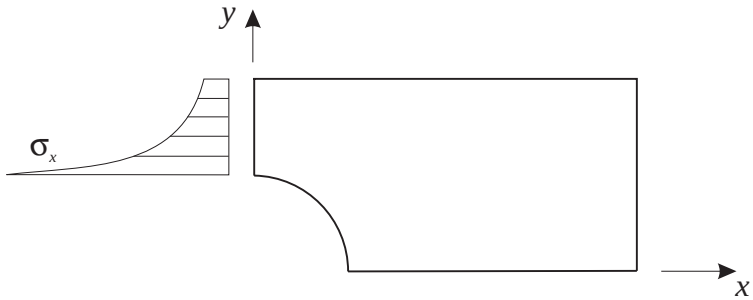


Figure 5.1.7.5 Stress Concentration Near Hole

Other Considerations

Analysis procedures for nonlinear analysis often require internal (volume) elements. For instance, in considering the plastic zone arising in the plate with a hole problem, volume elements must be placed around the hole to capture the plastic behavior (See Figure 5.1.7.6). This problem should be modeled using Conventional Symmetry. The SYMMetry Symmetry option will create the other three quadrants internally. The volume integration from this will multiply the solution time many times over, rendering this method of symmetry impractical. For problems involving plasticity (PLASticity in **CASE), it is NOT recommended to use SYMMetry card.

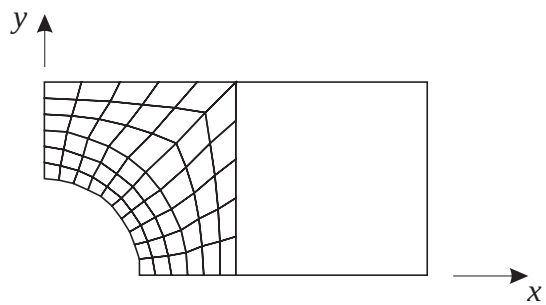


Figure 5.1.7.6 Internal Cells Used to Model Plastic Zone

Figure 5.1.7.7 illustrates a 2-D quarter symmetric model (shaded portion) that would benefit from the use of Conventional Symmetry. Very little modeling effort is required along the planes of symmetry, and the problem will run very quickly in **GPBEST** .

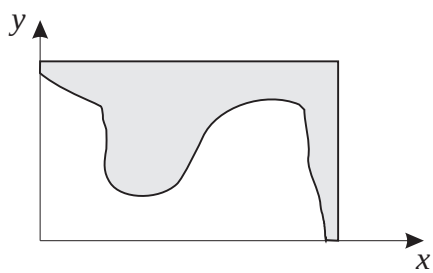


Figure 5.1.7.7 Model Favoring the Use of Conventional Symmetry

NOTE: This model (shaded portion) would be better suited for conventional symmetry since it would only take one or two boundary elements to model the edges of the model along the axes.

Figure 5.1.7.8 would favor the use of SYMMetry card. Since the model is somewhat complex along the planes of symmetry, SYMMetry would save the analyst significant modeling time.

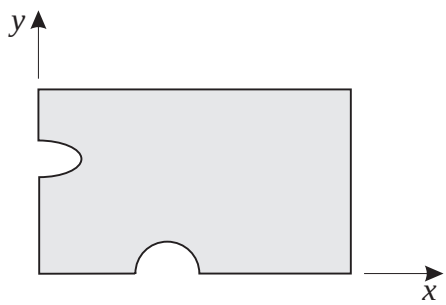


Figure 5.1.7.8 Model Favoring the Use of SYMMetry Option

It should be mentioned that the above examples are merely intended to illustrate a few of the considerations when choosing which symmetry method to use. Although 2-D examples were used, the 3-D symmetry options are similar. And although quarter-symmetry (QUAR) was illustrated, the principles extend to the other forms of symmetry (i.e., HALF and OCTA).

Examples of Use -

1. Perform a heat conduction analysis on an axisymmetric body with half symmetry about the $r - \theta$ plane.

```
**CASE
  TITLe  AXISYMMETRIC VESSEL
  AXISymmetry
  HEAT
  TIMEs-output 1.0 2.0 3.0
  SYMMetry HALF Z
```

2. Perform a 2D elastic analysis on a plate with a hole using quarter symmetry. (See Figure)

```
**CASE
  TITLe  PLATE WITH HOLE
  PLANe STRESS
  ELAStic
  SYMMetry QUAR X Y
```

3. 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
  SYMMetry OCTAL
```

5.1.8 RESTART FACILITY

RESTart RTYPE

Status - OPTIONAL

Full Keyword - **`RESTart`**

Function - Enables the restart facility.

Input Variables -

RTYPE (Alphanumeric) - REQUIRED

Allowable values are WRIT, READ, HOLE, SOLV, GMR

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.

HOLE - Only surface integration coefficients from the restart files are reused. All hole related coefficients are recomputed and stored on the restart files.

SOLVER - Uses the decomposed system matrix from a previous run. Integration and assembly are skipped.

GMR - Enables the GMR-based restart facility.

Additional Information -

Integration is often an expensive part of a 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. A number of FT* files 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 FT* files. It should be noted that in Version 4.0, some of these files use the GMR ID prefix.

RESTart HOLE provides an efficient means for rerunning a model for which only the hole element data has been changed. This option utilizes the standard restart files FT031-FT039. Consequently, RESTart HOLE may follow a RESTart WRITE, READ, or HOLE.

Upon completion of a RESTart HOLE analysis, the standard restart files are retained. Thus, RESTart HOLE may be followed by a RESTart READ or another RESTart HOLE.

Holes need not exist in either the current or previous geometry definition to utilize the RESTart HOLE option.

When employing RESTart READ or HOLE 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 **GPBEST**. This is the user's responsibility.

It is the user's responsibility to be certain that the material properties and surface geometry remain unchanged for RESTart HOLE analyses. Only hole element geometry and boundary conditions may change.

RESTart SOLVER is only available for steady-state versions of heat conduction, thermal distortion, and concurrent thermoelasticity.

The RESTart SOLVER option can only be used if a SAVE SOLVER card has been included in the previous run, along with RESTart WRITE, READ or HOLE. The only items that may be changed for a RESTart SOLVER analysis are the values specified for VARIable boundary condition sets and ambient temperatures for RELAtion bcsets. Geometry, material properties, interface definitions, and VALUe bcsets must be identical to those used for the previous run.

For runs which utilize the RESTart SOLVER option, the only data requirements are the **CASE section and complete **BCSET blocks for any boundary condition sets which change from the previous analysis. (The **BCSETs that are included must appear in the same order that was specified during the last SAVE SOLVER run.) The material (**MATERIAL), geometry (**GMR), and interface (**INTERFACE) sections need not be included in the restart solver input data set.

When RESTart GMR is specified, additional restart information must be supplied in the **GMR data sections. Specifically, within each **GMR data block, a RESTart-gmr WRITE, RESTart-gmr READ, or RESTart-gmr HOLE should be included. If none of these three cards is present, a RESTart-gmr WRITE is assumed.

When a RESTart flag appears in a HEAT analysis, then boundary temperature and flux are saved in FT121. This file can then be used for a subsequent DISTortion (thermal distortion) analysis.

RESTart capability is not currently available for periodic acoustic, structural acoustics, and forced vibration analyses.

RESTart is not available when executing **GPBEST** in a distributed computing environment (DCE) mode. (See **DCE for details on DCE.)

Examples of Use -

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

```
**CASE
  TITLe  AXLE - LOAD CASE 1A
  ELAStic
  TIMEs-output  1.0
  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
  ELAStic
  TIMEs-output  1.0
  RESTart READ
```

3. Save the integration and solver files generated during a steady-state heat conduction analysis of a tank mold.

```
**CASE
  TITLe  TANK MOLD (BOUNDARY CONDITIONS NO. 1)
  HEAT
  RESTart WRITE
  SAVE SOLVER
```

4. Use the decomposed system matrix saved during the previous heat conduction analysis of the tank mold.

```
**CASE
  TITLe  TANK MOLD (BOUNDARY CONDITIONS NO. 2)
  HEAT
  RESTart SOLVER
```

5. Select a GMR-based restart option. Use all the existing integration files from GMR1, but recompute the hole integration in GMR2.

```
**CASE
  TITLe  TANK MOLD  (COOLING LINE CHANGE)
  HEAT
  RESTart GMR
  .
  .
**GMR
  ID-Gmr GMR1
  MATE STEEL
  RESTart-gmr HOLE
  .
  .
**GMR
  ID-Gmr GMR2
  MATE STEEL
  RESTart-gmr HOLE
```

SAVE STYPE

Status - OPTIONAL

Full Keyword - **SAVE**

Function - Requests that certain matrices and problem information be saved on disk for subsequent analyses.

Input Variables -

STYPE (Alphanumeric) - REQUIRED

Allowable value is SOLV

SOLVER - Saves decomposed system matrix and right hand side matrices for subsequent analyses. Material, geometry, and interface information are also retained.

Additional Information -

This option is only available for elastostatic, steady-state heat conduction and steady-state concurrent thermoelastic analyses.

The SAVE SOLVER card must be used in conjuction with RESTart WRITE, READ, HOLE, or GMR.

Once a SAVE SOLVER analysis is completed, the decomposed system matrix and right hand side matrices remain on disk. These can be accessed on the following run by including a RESTart SOLVER card. Only the values specified in VARIable boundary condition sets and ambient temperatures defined in RELAtion bcsets may differ between the two analyses. Geometry, material properties, interface definitions and VALUe bcsets must remain unchanged. Essentially, only a new right hand side vector may be formed.

Examples of Use -

1. Save the integration and solver files generated during the heat conduction analysis of a piston.

```
**CASE
  TITLe  HEAT CONDUCTION IN PISTON A011
  HEAT
  RESTart WRITE
  SAVE SOLVER
```

INITial ITYPE

Status - OPTIONAL

Full Keyword - **`INITial`**

Function - Defines the initial conditions to be used for a transient analysis.

Input Variables -

ITYPE (Alphanumeric) - REQUIRED

Allowable value is STEA

STEADY - Use the results from a previous steady-state analysis as initial condition

Additional Information -

This option is only available for transient heat conduction.

During all steady heat conduction analyses, results are stored in the files gmr-name.FT128. These files must be present in the appropriate directory in order to invoke the INITial STEA option.

Examples of Use -

1. Perform a transient heat conduction analysis using a non-uniform initial temperature distribution.

```
**CASE
  TITLe  TRANSIENT THERMAL ANALYSIS OF CASTING MOLD
  PLANe
  HEAT  TRANSIENT
  TIMEs STEP  10  0.5
  INITial STEADY
```

5.1.9 EFFICIENCY AND PRECISION CONTROL

PRECision PTYPE

Status - OPTIONAL

Full Keyword - **PRECision**

Function - Optimizes the overall numerical efficiency and precision of an analysis. Parameters controlling integration and solution are set in a consistent manner to produce a desired level of accuracy with maximum efficiency.

Input Variables -

PTYPE (Alphanumeric) - OPTIONAL

Allowable values are PREM, HIGH, MEDIUM, and LOW.

Option	Number of Gauss Points used on Subelement*	Storage Precision of System Equation	Precision of Variables used in Solver
LOW	2×2	SINGLE	SINGLE
MEDIUM	3×3	SINGLE	SINGLE
HIGH	4×4	SINGLE	DOUBLE
PREMIUM	5×5	DOUBLE	DOUBLE

Additional Information -

This card sets default parameter for the numerical integration and the overall control of the analysis. See also the STORage-precision card and the SOLVer-precision card described on the following pages.

Default is HIGH precision analysis

In many cases, (such as in heat transfer) MEDIUM or LOW 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, axisymmetric periodic acoustic and axisymmetric forced vibration analyses, PRECision PREMIUM may have to be used. LOW is not recommended for stress analysis.

Examples of Use -

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

```
**CASE
  TITLe  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTart WRITE
  PRECision LOW
```

*An element may be divided into a number of subelements during integration.

STORage-precision STYPE

Status - OPTIONAL (overrides the storage precision defined on the PRECision card)

Full Keyword - **STORage-precision**

Function - Specifies the precision in which the system equation is stored in files prior to solution.

Input Variables -

STYPE (Alphanumeric) - REQUIRED

Allowable values are DOUBLE and SINGLE.

DOUB - Use double precision storage.

SING - Use single precision storage.

Additional Information -

If STORage-precision card is not included, then the default is based on the PRECision (precision) card. (See PRECision card in **CASE input).

In most cases, single precision storage is adequate and the required disk storage is kept to a minimum. However, for analysis of bodies with thin regions, the card PRECision HIGH is recommended. (See PRECision card in **CASE input).

Double precision storage is not allowed in plasticity analysis.

For problems in structural acoustics, the finite element structural part uses a single precision complex storage.

Examples of Use -

1. Use single precision storage for the heat conduction analysis of a piston.

```
**CASE
  TITLe  HEAT CONDUCTION IN PISTON A011
  HEAT
  RESTart WRITE
  PRECision HIGH
  STORage-precision SINGLE
```

SOLVer-precision APREC

Status - OPTIONAL (overrides the solver precision defined on the PRECision card)

Full Keyword - **`SOLVer-precision`**

Function - Specifies the precision (double or single) of the floating-point variables used in the solution of the system equations.

Input Variables -

APREC (Alphanumeric) - REQUIRED

Allowable values are DOUB and SING.

DOUB - Use double precision variables.

SING - Use single precision variables.

Additional Information -

- If this card is not input then the default is based on the PRECision (precision) card. (see PRECision card in **CASE input).
- In general, single precision is more efficient than double precision, and may be as high as 40-50% on some machines. However, some sensitive problems such as nonlinear analysis, or problems with long and slender bodies may require double precision for accurate results.
- This card may be used with the SOLVer-algorithm ELIM or the SOLVer-algorithm ITER cards (see next page).

Examples of Use -

```
**CASE
  TITLe  BRAKE SHOE ANALYSIS
  ELAStic
  SOLVer-algorithm ELIMINATION
  SOLVer-precision SINGLE
  NEUTral-file
```

SOLVer-algorithm STYPE CONV MAXI COND

Status - OPTIONAL

Full Keyword - **`SOLVer-algorithm`**

Function - Permits selection of the algorithm used for the solution of the system equations.

Input Variables -

STYPE (Alphanumeric) - REQUIRED

Allowable values are ELIM and ITER

ELIM - Use an elimination-based algorithm.
ITER - Use an iteration-based algorithm.

CONV (Real) - OPTIONAL (for STYPE=ITER)
Iteration precision - defines the precision of the converged solution. Recommended range is $0.0001 - 0.01$. The value depends on the precision of the analysis (see PRECision card): 0.01 for PRECision LOW; 0.001 for PRECision MEDI; and 0.0001 for PRECision HIGH and PRECision PREM. A larger value will decrease execution time.

MAXIMUM (Integer) - OPTIONAL (for STYPE=ITER)
Defines the maximum number of iterations for the iterative solver. The default value is 50. This card may be used in conjunction with the SOLVer-precision DOUB or the SOLVer-precision SING Cards (see previous input card).

COND (Real) - OPTIONAL (for STYPE=ITER)
Iteration pre-conditioning number. Recommended range is $0.001 - 0.01$. The default value is 0.001. A larger value will decrease execution time, however, for an ill-conditioned problem the solution may not converge.

Additional Information -

The default algorithm is ELIMINATION.

The ITERative algorithm can be an efficient alternative for large single region analyses or for multiregion problems with a large residual set. Reduction in the solver execution time of a factor of three can be obtained for the solution of equations on the order of 3000 degrees of freedom. While convergence of the ITERative algorithm cannot be guaranteed for illconditioned problems, the method is suitable for most large BEM analyses. Examples of ill-conditioned problems are problems of thin regions, regions with large aspect ratios, use of very stiff springs etc.

Iterative solver is only available for static elastic analysis, concurrent thermoelastic analysis under steady state conditions and for steady state heat transfer analysis.

This card may be used with the SOLVer-precision DOUB or SOLVer-precision SING card (see previous page).

The Iterative solver will automatically default to the elimination solver for certain problems. These include: Contact problems, Plasticity problems, problems with less than 2000 dof, and problems in which the material properties (modulus of elasticity and/or conductivity) in a multi-region analysis vary greatly (by a factor of 100 or more) from one region to the next.

Examples of Use -

1. Select the iterative algorithm for the solution of a turbine disk.

```
**CASE
  TITLe  TURBINE DISK (MODEL 4B)
  ELAStic
  RESTart WRITE
  PRECision MEDI
  SOLVer-algorithm ITERATIVE 0.001 0.01
  NEUTral-file
```

QUICk-solver

Status - OPTIONAL (Default)

Full Keyword - **`QUICk-solver`**

Function - Specifies the use of the **GPBEST** block elimination solver called ‘QUICk-solver’.

Input Variables - None

Additional Information -

There are two elimination solvers in **GPBEST** . The first is a block solver called QUICk-solver. The second is called STANdard-solver (see next card), which is based on elimination of one equation at a time. The default is QUICk-solver, which is used if neither QUICk-solver nor STANdard-solver is selected. QUICk-solver is an order of magnitude faster than STANdard-solver. QUICk-solver has limited pivoting, but is stable for most problems. STANdard-solver should be used only if QUICk-solver fails.

If **GPBEST** determines that QUICk-solver cannot be used to successfully solve a problem, it will direct the user to rerun the problem using STANdard-solver.

Examples of Use -

1. Use the QUICk-solver on an elastic problem.

```
**CASE
  TITLe  WHEEL ROTOR
  ELAStic
  QUICk-solver
  NEUTral-file
```

STANdard-solver

Status - OPTIONAL

Full Keyword - **`STANdard-solver`**

Function - Specifies the use of the original **GPBEST** elimination solver (with more extensive pivoting than QUICk-solver).

Input Variables - None

Additional Information -

There are two elimination solvers in **GPBEST** . The first is a block solver called QUICk-solver. The second is called STANdard-solver, which is based on elimination of one equation at a time. The default is QUICk-solver (see previous card), which is used if neither QUICk-solver nor STANdard-solver is selected. QUICk-solver is an order of magnitude faster than STANdard-solver. QUICk-solver has limited pivoting, but is stable for most problems. STANdard-solver should be used only if QUICk-solver fails.

If **GPBEST** determines that QUICk-solver cannot be used to successfully solve a problem, it will direct the user to rerun the problem using STANdard-solver.

STANdard-solver is available for all problems.

Examples of Use -

1. Use the STANdard-solver elimination solver on an acoustics problem.

```
**CASE
  TITLe  ROTOR VIBRATION
  ACOUSTICS
  STANdard-solver
  NEUTral-file
```

MEMOry NMAG

Status - OPTIONAL

Full Keyword - **MEMOry**

Function - Decrease execution time for large problems by increasing the random access memory (RAM) that is dynamically allocated for use during the solution of the system matrix.

Input Variables -

NMAG (Integer) - REQUIRED

The number of MEGABYTES of RAM that is dynamically allocated

during the solution of the system matrix.

Additional Information -

The default setting in **GPBEST** is 24 megabytes.

NMAG is an upper limit. If NMAG is set to a value greater than the amount required to hold the largest substructured matrix in core, then **GPBEST** will allocate only the amount required to hold the largest matrix.

It is recommended not to reduce the size of NMAG below the default setting of 24 megabytes. Execution time of large problems may dramatically increase if NMAG is reduced below the default setting. As mentioned above, **GPBEST** will allocate only the amount required during the execution of a small problem.

For installations with more than 36 megabytes of RAM, increasing the dynamic memory allocation can dramatically reduce the execution time of large problems. In setting NMAG, consideration should be given to other processes running in a shared environment. The optimum setting for maximum efficiency on your installation can be obtained by trial and error. As a rule of thumb, NMAG should not exceed the amount of available RAM minus 12 megabytes that is available for your process.

NOTE: NMAG cannot exceed 1024 megabytes.

Examples of Use -

1. Increase the amount of available RAM for **GPBEST** execution to 256 megabyte on an installation with 512 megabytes.

```
**CASE
  MEMOry 256
```

TOLErance-convergence RTOL

Status - OPTIONAL

Full Keyword - **`TOLErance-convergence`**

Function - Sets the convergence tolerance for nonlinear algorithms, and for the convergence of eigenvalues.

Input Variables -

RTOL (Real) - REQUIRED

Defines the convergence tolerance.

Additional Information -

If the TOLErance-convergence card is not included in CASE control, RTOL defaults to 0.005.

For analysis with nonlinear interface conditions, the convergence is tested at the end of the ith iteration by computing two DNORMs, one for displacement and one for tractions:

$$DNORM = \sqrt{\frac{\sum_{n=1}^{N} ({}_{i}X_n - {}_{i-1}X_n)^2}{\sum_{n=1}^{N} ({}_{i-1}X_n)^2}}$$

where ${}_{i}X_n$ represents the displacement/traction vector solution that is found after the ith iteration. Convergence is assumed when DNORM $\leq$ RTOL for both displacement and traction.

Convergence for eigenvalues is based on a norm of the eigenvectors of a reduced system [63]. If TOLErance-convergence is not input, the machine precision is assumed. A larger tolerance will generally reduce the computational time, but eigenvalues in a tight cluster may be missed.

Examples of Use -

```
**CASE
  TITLe  CONTACT PROBLEM NO.1
  ELAS
  PLANe STRAIN
  TIMEs-output 1.0
  MAXImum-iterations  8
  TOLErance-convergence 1.E-4
```

MAXImum-iterations NITER

Status - OPTIONAL

Full Keyword - **`MAXImum-iterations`**

Function - Define the maximum number of iterations per time step for nonlinear algorithms.

Input Variables -

NITER (Integer) - REQUIRED

Sets the number of maximum iterations per time step.

Additional Information -

The default for nonlinear problems is a maximum of 20 iterations.

This card does not control the number of iterations in an eigenvalue extraction. The algorithm currently used to extract eigenvalues is a complex, self-contained program, with an iteration process with multiple branches of execution. The iteration control is sophisticated and works best without user intervention.

Examples of Use -

1. Example with a limit of 10 iterations.

```
**CASE
  TITLe  CONTACT PROBLEM NO. 1
  ELAS
  PLANe STRAIN
  TIMEs-output 1.0
  TOLErance-convergence  0.02
  MAXImum-iterations  10
```

5.1.10 OUTPUT OPTIONS

ECHO-input

Status - OPTIONAL

Full Keyword - **`ECHO-input`**

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 a plane strain elastic analysis with an echo of the input data set.

```
**CASE
  TITLe  DAM - PLANE STRAIN ASSUMPTION
  PLANe STRAIN
  ELAStic
  RESTart WRITE
  ECHO-input
```

PRINtout-control PTYPE

Status - OPTIONAL

Full Keyword - **`PRINtout-control`**

Function - Requests specific printed output in the result file.

Input Variables -

PTYPE (Alphanumeric) - REQUIRED

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

BOUN - For printing the displacements and tractions, or corresponding quantities such as velocity, 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 elasticity, thermoelasticity and consolidation.)

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

DATA - For printing the input data.

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

Additional Options (not related to printout of results) -

NONE - To produce only a minimal printed output.

LIMI - To printout the current limits of **GPBEST** .

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

Additional Information -

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

If a PRINtout-control (BOUN, NODA, DATA, LOAD or NONE) request does not appear in the case control input then all three types of output (BOUN, NODA, LOAD and DATA) will be printed by default.

Examples of Use -

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

```
**CASE
  TITLe  ROTOR - ELASTIC ANALYSIS
  ELAStic
  TIMEs-output  1.0   2.5   6.0
  PRINtout-control LOAD
```

NEUTral-file NTYPE

Status - OPTIONAL

Full Keyword - **`NEUTral-file`**

Function - Requests the generation of a **GPBEST** neutral file. This neutral file is a **formatted, direct-access file** called NEUTRAL.OUT written in ASCII format. The file contains input data as well as result information. More information on this can be found in the Section entitled ‘‘GPBEST Neutral File Format.’’

Input Variables -

NTYPE (Alphanumeric) - OPTIONAL

Allowable values are BOUN, NODA, LOAD, DATA, ALL, READ and MODI.

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

NODA - For printing the displacements, stresses and strains at all geometry nodes on the boundary and sampling points in the neutral file.

LOAD - For printing the resultant load value on each boundary (and hole) element and the total load equilibrium of each region, excluding resultant body force in the neutral file.

DATA - For printing only the input data to the neutral file.

ALL - For printing BOUN, NODA, DATA and LOAD information in the neutral file with a single request (this is the default option).

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

MODI - Writes the neutral file in a modified form. For special use only. See section 10.3.2 (Nodal Coordinates) of the **GPBEST** User’s Manual - Volume II.

Additional Information -

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

For printing two or more types of output in the neutral file, a separate NEUTral-file request must be included for each option on a separate line.

For forced vibration problems, only neutral file corresponding to NEUTral-file NODA is generated.

For post-processing of results using NEUTRAL.OUT through the translators available with GPBEST , it is necessary to output the geometry data to the neutral file. Therefore, if either BOUN, LOAD or NODA option is selected, the option DATA must also be included.

For problems of acoustics or structural acoustics, NTYPE should not be specified. In the case of a one-way, coupled structural acoustics, instead of a NEUTRAL.OUT file, the following two files are created:

NEUTRAL.OUT.STR , corresponding to the structural information, and

NEUTRAL.OUT.FLU , corresponding to the acoustic fluid information.

In the case of a periodic acoustics using the indirect formulation and one-way, coupled structural acoustics using a multi-frequency sweep, the NEUTral-file file is recorded for the first frequency value only.

Examples of Use -

1. Generate a **GPBEST** neutral file for the steady-state heat conduction analysis of a casting mold.

```
**CASE
  TITLe  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTart HOLE
  PRECision LOW
  NEUTral-file ALL
```

2. Generate a **GPBEST** neutral file for the steady-state heat conduction analysis of a casting mold, but only print the input data and boundary results.

```
**CASE
  TITLe  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTart HOLE
  PRECision LOW
  NEUTral-file DATA
  NEUTral-file BOUN
```

3. Only output **GPBEST** model geometry into a **GPBEST** neutral file for post-processing. A graphics program such as BESTVIEW can be used to display the model before proceeding with the rest of the analysis.

```
**CASE
  TITLe  SIMPLE ELASTIC MODEL
  ELAStic
  NEUTral-file
  CHECk-data
```

HISTory-quantities I1 I2 I3 . . . I9

Status - OPTIONAL

Full Keyword - **`HISTory-quantities`**

Function - Identifies which quantities are to be output in the nodal result history file called HISTORY.OUT

Input Variables -

I1 (integer) - REQUIRED

Identifies the first quantity is to be output.

I2 . . . I9 (Integer) - OPTIONAL

Identifies additional quantity to be output.

Additional Information -

The integer identifiers (I1-I9) are analysis dependent and are defined in the following table(s), on the next page.

Result histories are currently available only in heat transfer, uncoupled thermoelastic (CTHErmal) and acoustic analysis.

The maximum number of quantities that can be output in a single run is 9.

If this card is input, then the HISTory-points card (see **GMR input) should be input in at least one GMR.

Examples of Use -

1. Specify that real and imaginary pressure and real and imaginary normal velocity is to be output in the HISTORY.OUT file.

```
**CASE
  TITLE  Acoustic Analysis of Sphere
  ACOUSTIC PERIODIC
  FREQuency-output  HERTZ  10.0   100.0   500.0
  HISTory-quantities   1   2   4   5
  NEUTral-file
  ECHO-input
```

Table 1. History Integer Identifier Definitions
for Acoustic Analysis

Identifier	Quantity		Identifier	Quantity	
1	Pressure	(Real)	16	Magnitude of Velocity	(Real)
2	Pressure	(Imaginary)	17	Magnitude of Velocity	(Imaginary)
3	Pressure	(Modulus)	18	Magnitude of Velocity	(Modulus)
4	Normal Velocity	(Real)	19	Pressure	(Decimal)
5	Normal Velocity	(Imaginary)	20	Pressure	(dB)
6	Normal Velocity	(Modulus)	21	Normal Intensity	(Decimal)
7	Velocity-x	(Real)	22	Normal Intensity	(dB)
8	Velocity-x	(Imaginary)	23	Intensity-x	(Decimal)
9	Velocity-x	(Modulus)	24	Intensity-x	(dB)
10	Velocity-y	(Real)	25	Intensity-y	(Decimal)
11	Velocity-y	(Imaginary)	26	Intensity-y	(dB)
12	Velocity-y	(Modulus)	27	Intensity-z	(Decimal)
13	Velocity-z	(Real)	28	Intensity-z	(dB)
14	Velocity-z	(Imaginary)	29	Magnitude of Intensity	(Decimal)
15	Velocity-z	(Modulus)	30	Magnitude of Intensity	(dB)
			31	Energy Density	(Decimal)
			32	Energy Density	(dB)

The modulus is the square-root of the sum of the squares of the real and imaginary parts.

The pressure in item No. 19 is identical to the modulus of pressure in items No. 3.

Table 2. History Integer Identifier Definitions
for Heat Transfer and Uncoupled-Thermal Elastic Analysis

Identifier	Quantity
1	Temperature

CONTour-plot TYPE

Status - OPTIONAL

Full Keyword - **`CONTour-plot`**

Function - Requests the generation of additional output in the neutral file that can be used to create contour plots of results in the interior of a two-dimensional or axisymmetric body, or on the surface of a hole, runner, or cooling line in a three- dimensional body.

Input Variables -

TYPE (Alphanumeric) - OPTIONAL (for 2-D analysis only)

Allowable values are QUAD and LINE

QUAD - The results over the 2D contour cells assume a QUADratic variation.

LINE - The results over the 2D contour cells assume a LINEar variation. (Values at the mid-nodes of the cell are linearly interpolated from the values at the corner nodes.)

Additional Information -

If the CONTour-plot card is input, a neutral file will automatically be created. The NEUTral-file card is therefore not required, however, if the NEUTral-file card is included, the options included on the NEUTral-file card (if any) have precedence over the CONTour-plot card.

This card is always assumed in generalized axisymmetry, whether it is specified or not. A 180-degree model will be generated in the **GPBEST** neutral file for post-processing.

Additional Information for 2-D Analysis -

If TYPE is not input, then in most cases **GPBEST** will assume QUAD. If **GPBEST** determines an excess number of nodes are generated relative to the number of boundary nodes, then LINE will be assumed, and the values of the variables of the mid-nodes of the cell are linearly interpolated from the values at the corner nodes. If a fine contour is required the user should input QUAD. If fast execution is important the user should input LINE.

The CONTour-gmr card definition under **GMR, if included in data set, overrides the CONTour-plot definition included under **CASE for that GMR only.

Additional Information for 3-D Analysis -

In the case of HOLE, sampling surfaces are created about the circumference of the hole using four quadratic elements in the circumferential direction and one in the axial direction. In the case of RUNNer and cooling lines (CLINe), sampling surfaces are created for both the hole (containing the runners and cooling lines) and the runner

and cooling line. In order to expose the runners and cooling lines, only half of the circumference of the hole is modeled with sampling surfaces (two quadratic elements in the circumferential direction). Each element of a runner or cooling line is modeled with a single flat quadratic element. The width of the element is made larger for the runner than it is for the cooling line in order to distinguish between the two. See the accompanying diagram.

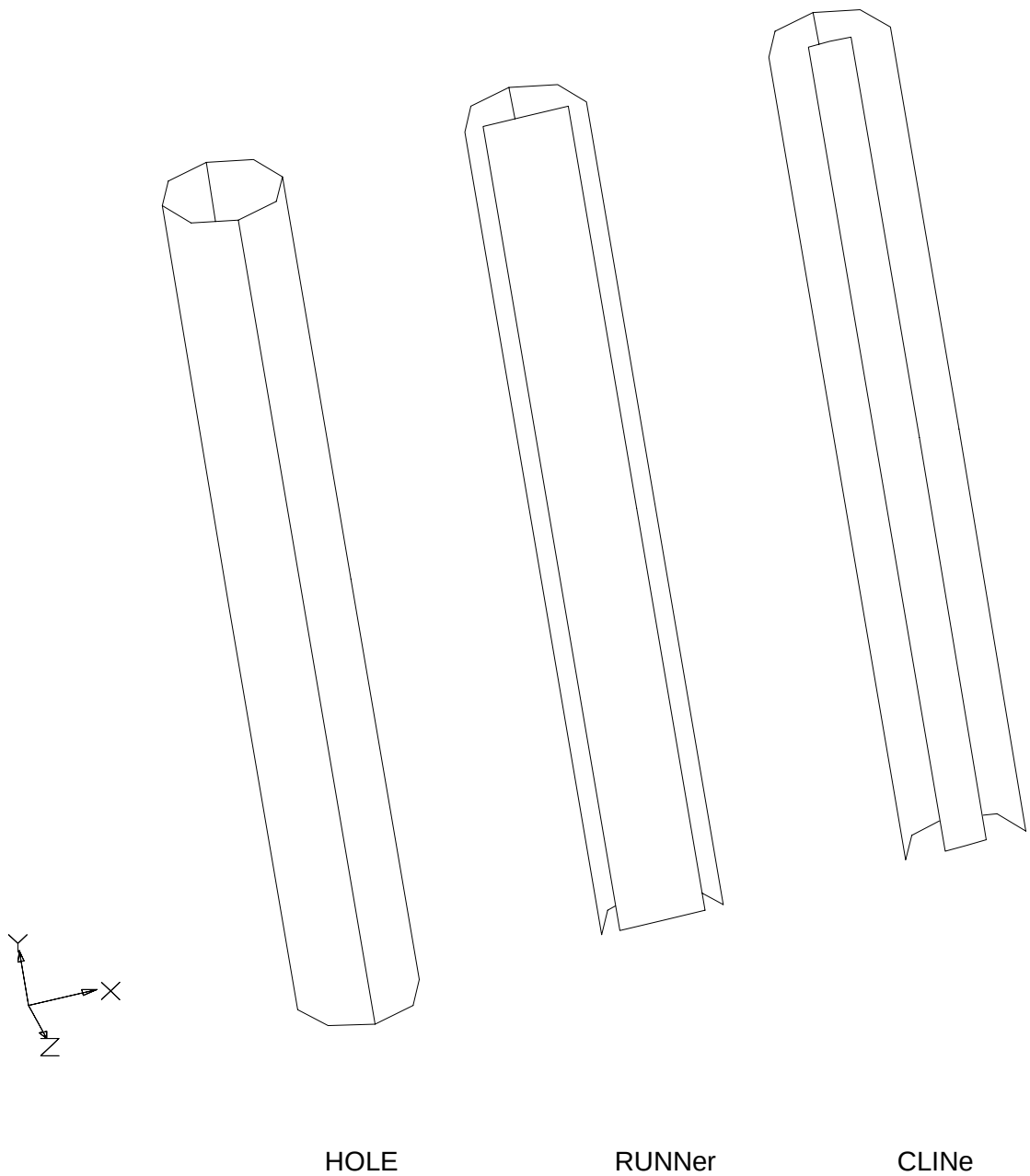


Figure for the CONTour-plot Card: Line Elements

Examples of Use -

1. Request CONTour-plot option for post-processing.

```
**CASE
  TITLe  MOLD COMPONENT
  PLANe STRESS
  NEUTral-file
  CONTour-plot
```

2. Model the bending of a round cantilever beam using generalized axisymmetry. Although the model in Figure (b) is two-dimensional, the CONTour-plot results (Figure (c)) show the 180-degree model for post-processing.

```
**CASE
  TITLe  CANTILEVER BEAM - CIRCULAR CROSS-SECTION
  AXISymmetry GENERAL
  ELAStic
  NEUTral-file
  CONTour-plot    $ THIS CARD IS ALWAYS IN ACTIVE
                   $ FOR GENERALIZED AXISYMMETRY
```

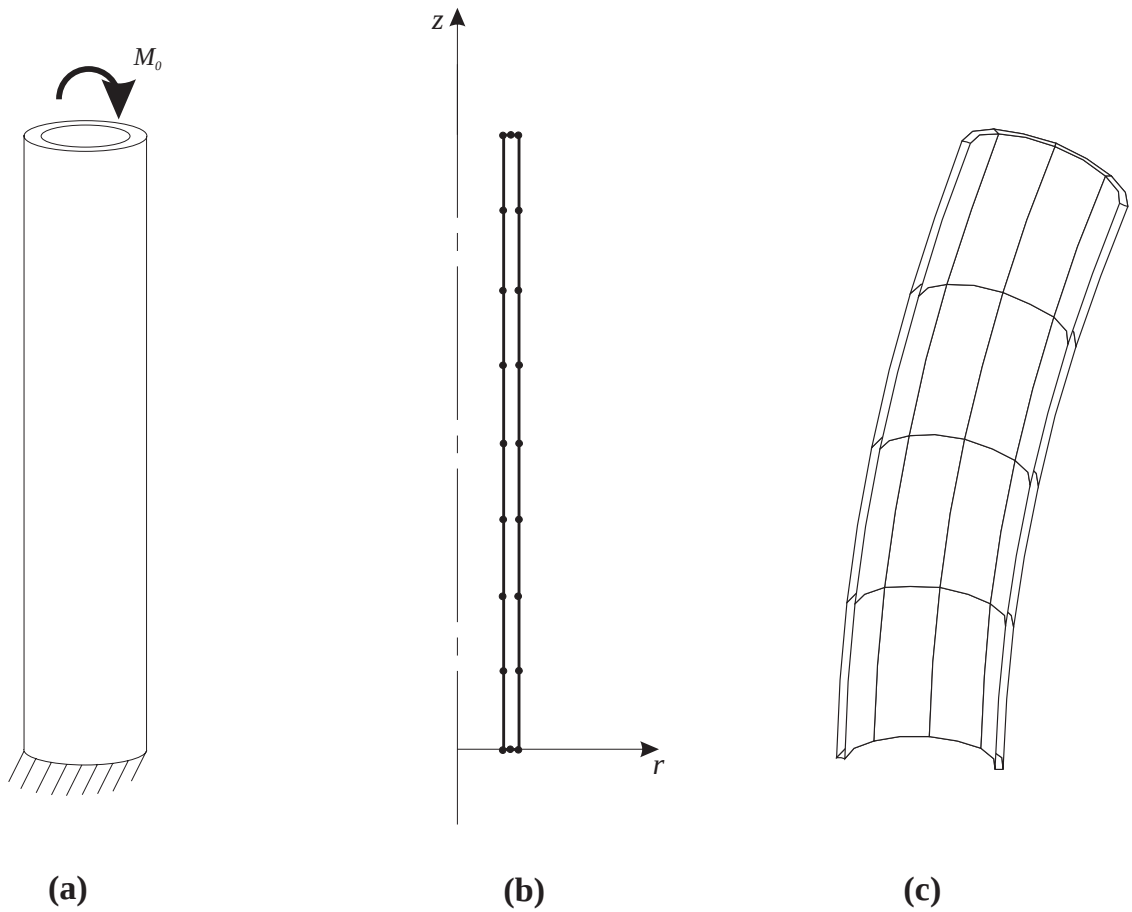


Figure for CONTour-plot Card (AXISymmetry GENE Problem):
(a) Cylinder With End-Moment (b) Axisymmetric BE Model
(c) Post-Processed Deformation

DECIbel-output PRESREF INTREF POWREF ENEREF

Status - OPTIONAL

Full Keyword - **`DECIbel-output`**

Function - Define the reference values for decibel output of pressure, intensity, power, and energy density results in an acoustic analysis.

Input Variables -

PREREF (Real) - REQUIRED
The reference value for pressure (P_{ref}).
INTREF (Real) - REQUIRED
The reference value for intensity (I_{ref}).
POWREF (Real) - REQUIRED
The reference value for power (W_{ref}).
ENEREF (Real) - REQUIRED
The reference value for energy density (E_{ref}).

Additional Information -

The sound pressure level (SPL), the intensity level (IL), the sound power level (SWL), and the sound energy density level (SEDL) are defined as:

$$
\begin{aligned}
SPL &= 20 \cdot log_{10}\left(\frac{Pressure}{P_{ref}}\right) dB \\
IL &= 10 \cdot log_{10}\left(\frac{Intensity}{I_{ref}}\right) dB \\
SWL &= 10 \cdot log_{10}\left(\frac{Power}{W_{ref}}\right) dB \\
SEDL &= 10 \cdot log_{10}\left(\frac{EnergyDensity}{E_{ref}}\right) dB
\end{aligned}
$$

If the DECIbel-output card is not input then the following default reference values (PRESREF, INTREF, POWREF, ENEREF) are assumed:

$$
\begin{aligned}
P_{ref} &= 2x10^{-5}\frac{N}{m^2} \\
I_{ref} &= 1x10^{-12}\frac{W}{m^2} \\
W_{ref} &= 1x10^{-12}W \\
E_{ref} &= 2.9x10^{-15}\frac{Ws}{m^3}
\end{aligned}
$$

These default values are approximately the intensity (given in SI units) of a 1000 Hz pure tone in air, which is barely audible to humans. If this card is input, then all four values must be input.

Examples of Use -

1. Specify the typical reference values for air.

```
**CASE
  TITLe Pulsating Sphere
  ACOUstic PERIODIC DIRECT
  FREQuency 10.0
  NEUTral-file
  DECIbel-output 2.0e-5 1.0e-12 1.0e-12 2.9e-15
  ECHO-input
```

VECTor-intensity OFF

Status - OPTIONAL (For ACOUstic analysis only)

Full Keyword - **`VECTor-intensity`**

Function - Restricts the output of the vectors of velocity and the vectors of sound intensity in the NEUTRAL.OUT file.

Input Variables -

OFF (Alphanumeric) - REQUIRED

Additional Information -

If this card is not input, then the vectors of velocity and sound intensity for sampling points are output to the NEUTRAL.OUT file and available for translation using the POSTBEST translator. However, having these quantities present in the NEUTRAL.OUT file will prevent the NEUTRAL.OUT file from being imported into BESTVIEW! Therefore, if the NEUTRAL.OUT file is to be imported into BESTVIEW, this card must be included.

Examples of Use -

1. Analysis of a car interior.

```
**CASE
  TITLe Car Interior
  ACOUstic PERIODIC DIRECT
  FREQuency 20.0
  NEUTral-file
  VECTor-intensity OFF
```

EIGEnvalue OUTPUT MODES NVECTORS

Status - OPTIONAL

Full Keyword - **`EIGEnvalue`**

Function - Specify the number of eigenvalues and eigenvectors to be calculated and output.

Input Variables -

OUTPUT (Alphanumeric) - REQUIRED MODES (Integer) - REQUIRED The number of eigenvalues to be output NVECTORS (Integer) - OPTIONAL The number of eigenvectors to be output

Additional Information -

The maximum number of eigenvalues and eigenvectors (MODES, NVECTORS) that can be calculated is 50.

If this card is not input, the default value of 20 eigenvalues and 20 eigenvectors are reported.

If MODES is less than NVECTORS, then MODES is reset to NVECTORS.

If NVECTOR is not input, then the value of NVECTOR is assumed to be equal to MODES.

Examples of Use -

1. Request the output of the lowest 30 eigenvalues and the lowest 20 eigenvectors.

```
**CASE
  TITLe Free Vibration of a Shell
  FREE-vibration
  EIGEnvalue OUTPUT 30 20
```

2. Request the output five eigenvalues and two eigenvectors above 2000 hertz.

```
**CASE
  TITLe Free Vibration of a Crankshaft
  FREE-vibration
  EIGEnvalue-option-2 SHIFT 2000.0
  EIGEnvalue OUTPUT 5  2
```

5.1.11 MISCELLANEOUS CONTROL OPTIONS

CHECk-data

Status - OPTIONAL

Full Keyword - **CHECk-data**

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

Input Variables - NONE

Additional Information -

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

Examples of Use -

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

```
**CASE
  TITLe  TURBINE BLACE A7311 - THERMOELASTIC
  CTHErmal STEADY
  RESTart WRITE
  CHECk-data  $  DATA CHECKING ONLY
```

2. Only output **GPBEST** model geometry into a **GPBEST** neutral file for post-processing. A graphics program such as BESTVIEW can be used to display the model before proceeding with the rest of the analysis.

```
**CASE
  TITLe  SIMPLE ELASTIC MODEL
  ELAStic
  NEUTral-file
  CHECk-data
```

LOAD TYPE

Status - OPTIONAL

Full Keyword - **LOAD**

Function - Identifies a problem having complex loading (boundary conditions and body forces) with both real and imaginary parts.

Input Variables -

TYPE (Alphanumeric) - REQUIRED

Allowable value is COMPLEX

COMPLEX - For complex-valued loading in periodic elastodynamic and acoustic analysis.

Additional Information -

If this card is input, all boundary conditions have to be input with real and imaginary parts.

If this card is not input then only the real part of the complex loading is input. i.e. the imaginary part of all boundary conditions and body forces are assumed to be zero.

Boundary conditions and body forces of all quantities input, may be specified as complex values of either the maximum (peak) values or the effective (root-mean-square) values of those quantities. All results reported by GPBEST will then be output consistent with the type of values that were input. The user, however, must be consistent and specify all input using the same type of values (maximum or effective values).

If the maximum value and phase angle or the effective value and phase angle are known, then they must be converted to the complex value (real and imaginary parts) according to the following relations. Given the form:

$$P(t) = \hat{P} cos\left(\omega t + \phi_p\right),$$

the real part P_r and imaginary part P_i are determined by

$$P_r = \hat{P} cos\phi_p$$
$$P_i = -\hat{P} sin\phi_p$$

where $\hat{P}$ is either the maximum or effective value and ϕ_p is the phase angle.

Examples of Use -

1. Acoustic periodic analysis of car interior.

```
**CASE
  TITLe ANALYSIS OF CAR INTERIOR NUMBER 3725
  ACOUstic PERIODIC
  FREQuency-output HERTZ   100.0   150.0   200.0
  NEUTral
  LOAD COMPLEX
```

BCFIle FILENAM

Status - OPTIONAL

Full Keyword - **`BCFIle`**

Function - Specify a file which contains frequency-dependent, nodal based displacement or velocity boundary condition information for a periodic ACOUstic analysis. The displacement/velocity information can be obtained from a finite element analysis or from measurements of structural dynamics after a discrete Fourier transformation.

Input Variables -

FILENAM (Alphanumeric) - REQUIRED

Name of file containing the boundary condition information.

Additional Information -

If the full pathname of the file is not given, GPBEST will assume the file resides in the execution directory. The filename/pathname must not exceed 240 characters.

This file allows a maximum of 128 characters per line. All other GPBEST input data protocol should be observed when creating the BCFIle. Comments are allowed and are identified with the symbols $ or #.

If the ECHO-input card is input under **CASE input, then the contents of the BCFIle are output to the GPBEST result file immediately after the main GPBEST input data file is printed.

Only one BCFIle can be defined in a GPBEST data set.

Format of BCFIle -

There are two basic formats that can be used for the BCFIle. The major difference between these two formats are in the way the displacement/velocity information is related to the GPBEST input data.

In the first case the nodes specified in the BCFIle refer to the node IDs specified in the **GMR input of the GPBEST input data file. In the second case, a point list is input in the BCFIle and the displacement/velocities which are input refer to these points. The point list consists of the node IDs and the coordinates obtained from the FEM analysis (or other outside sources) and are independent of nodes in the GPBEST input data set. GPBEST then automatically relates the FEM node (and hence the nodal displacement/velocity values) to the nodes in the GPBEST data set based on geometric proximity. The advantage to this approach is that the FEM nodes do not have to directly line up with the GPBEST nodes allowing more flexibility in modeling and allowing the FEM discretization to be more refined than the BEM discretization which is often desirable.

The boundary condition file can also be appended to the end of the GPBEST data set with the keyword **BCFILE preceding the file contents.

Case Control

FORMAT 1

```
DISPLACEMENT (or VELOCITY)
FREQUENCY FVALUE1
  BENODE1     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
  BENODE2     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
     ⋮         ⋮      ⋮        ⋮      ⋮        ⋮      ⋮

FREQUENCY FVALUE2
  BENODE1     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
  BENODE2     UX-R   UX-I     UX-R   UY-I     UZ-R   UZ-I
     ⋮         ⋮      ⋮        ⋮      ⋮        ⋮      ⋮
```

Repeat the displacement/velocity information for each frequency listed on the FREQuency-output card and/or the FREQuency SWEEP card.

FORMAT 2

```
NEIGHBORHOOD VALUE
POINT
  FENODE1     COOR-X   COOR-Y   COOR-Z
  FENODE2     COOR-x   COOR-y   COOR-Z
     ⋮          ⋮        ⋮        ⋮

DISPLACEMENT (or VELOCITY)
FREQUENCY FVALUE1
  FENODE1     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
  FENODE2     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
     ⋮         ⋮      ⋮        ⋮      ⋮        ⋮      ⋮

FREQUENCY FVALUE2
  FENODE1     UX-R   UX-I     UY-R   UY-I     UZ-R   UZ-I
  FENODE2     UX-R   UX-I     UX-R   UY-I     UZ-R   UZ-I
     ⋮         ⋮      ⋮        ⋮      ⋮        ⋮      ⋮
```

Repeat the displacement/velocity information for each frequency listed on the FREQuency-output card and/or the FREQuency SWEEP card.

The value on the neighborhood line represents the geometric distance in which a FEM node is considered close enough to a BEM node to contribute in the determination of the displacement or velocity at the BEM node.

Examples of Use -

1. 1-way coupled DBEM with multiple frequency sweep:

```
**CASE
  TITLe  Test 1
```

```
SACOustic   PERI   DIRECT   ONEWAY
FREQuency   SWEEP  10.0   20.0   2.0
BCFIle test1.nd
ECHO-input
```

INCLude-file FILE

Status - OPTIONAL

Full Keyword - **`INCLude-file`**

Function - Used to include additional lines of data, which are stored in another file, in the current **GPBEST** data file. The additional lines are inserted at the point where the INCLude-file statement is located. This allows the user to place frequently used data (e.g., material properties) in a single file and include it in any **GPBEST** data file using the INCLude-file keyword.

Input Variables -

FILE (Alphanumeric) - REQUIRED

Name of the file to be included.

Additional Information -

This card can appear anywhere in the GPBEST data file except under the **DCE input section.

The user may include as many files as desired.

When **GPBEST** is executed, it will start to read the **GPBEST** data file (which was specified upon execution) in successive order until it encounters an INCLude-file statement. At that point, the program will begin to read the file specified on the INCLude-file card and continues until the end-of-file is reached. At that point, the program switches back and begins to read the main **GPBEST** data file starting with the next card following the INCLude-file statement. Reading continues until either the program encounters another INCLude-file statement, at which time that INCLude-file file is read, or until all the cards in the main data file have been read.

If the full pathname of the file is not given, **GPBEST** will assume the file resides in the execution directory.

The filename/pathname must not exceed 240 characters.

Recursive INCLude-file is not allowed. A file that is included using this keyword cannot contain INCLude-file statements.

If the ECHO-input card is input under **CASE input, then all files that are included in the data set with the INCLude-file card will be printed as part of the data echo.

The INCLude-file card can appear anywhere in the **GPBEST** data file, not just in **CASE.

Examples of Use -

1. Include material information using an INCLude-file statement.

```
**CASE
  TITLe FAN BLADE
  ELAStic
  TIMEs-output   1.0   2.0
  NEUTral-file
  ECHO-input

  INCLude-file /disk1/matlib/steel.5

**GMR
  ID-Gmr BLADE
  MATErial STEEL
  TREFerence 72.0
  TINTegration 72.0
  POINts
    1         1.0     0.0     0.0
    . . .


The file /disk1/matlib/steel.5 contains the following cards:

**MATERIAL
  ID-Material STEEL
  TEMPeratures       50.0       300.0      500.0       $ (F)
  EMODulus-temp-dep 30.3E+6  29.0E+6  27.56E+6  $ (psi)
  ALPHa-temp-dep     6.0E-6   6.7E-6   7.2E-6   $ (1/F)
  POISson       0.3
```

COORdinate-system ATYPE

Status - OPTIONAL

Full Keyword - **`COORdinate-system`**

Function - Specifies the type of coordinate system that was used in the generation of the **GPBEST** input data that is currently being read.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are the following:

CARTESIAN - for a cartesian coordinate system X,Y (2-D) and X,Y,Z (3-D)

CYLINDRICAL - for a cylindrical coordinate system R,θ (2-d) and R,θ,Z (3-D)

RZ - for an axisymmetric coordinate system in which the first component refers to the radial (R) direction and the second component refers to the axial (Z) direction.

ZR - for an axisymmetric coordinate system in which the first component refers to the axial (Z) direction and the second component refers to the radial (R) direction.

Additional Information -

If this card is not input, then a cartesian coordinate system is assumed in 2-D and 3-D analysis and an axisymmetric RZ system is assumed in axisymmetric analysis.

If this card is input, then all data (geometry, boundary conditions, body forces, etc.) must refer to the type of coordinate system selected.

The order of the components for each system is indicated above and must be followed for input of coordinates as well as all other input, i.e., the global directions that are referred to in the boundary condition and body force input will assume the component order indicated above.

The value of the θ component should be input in degrees when defining the coordinates of a nodal point.

This card is not available in strucural acoustic analysis.

In analyses containing anisotropic material properties in which the cylindrical coordinate system is specified, the material must be directionally independent in the R-θ plane.

When the input data is read, **GPBEST** automatically converts the input data from the coordinate system that was specified to the cartesian coordinate system (or the axisymmetric RZ system). All calculations in **GPBEST** are carried out in the cartesian coordinate system (or in the axisymmetric RZ system).

Examples of Use -

1. Specify a cylindrical coordinate system.

```
**CASE
  TITLe  ANALYSIS OF A CYLINDRICAL WATER TANK
  ELAStic
  COORdinate-system CYLINDRICAL
  NEUTral-file
```

EIGEnvalue-option-1 ATYPE

Status - OPTIONAL

Full Keyword - **`EIGEnvalue-option-1`**

Function - Identifies that sampling points will be employed in the generation of the mass matrix in an acoustic eigenfrequency analysis or in a free-vibration analysis.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable value is SAMPLING

SAMPLING - identifies that all sampling points present in the data file (or created through the use of the CONTour-plot, CONTour-gmr or SAMPling-surfaces cards) are used in the formation of the mass matrix.

Additional Information -

If this card is not input, then sampling points are NOT included in the mass matrix.

Sampling points added to the interior of a body can increase the accuracy of the eigenvalue solution in an efficient manner.

In acoustic eigenfrequency analysis with all rigid boundaries (i.e., all zero velocity boundary conditions) or in free-vibration analysis with all unconstrained elements (i.e., free-free boundary conditions), sampling points cannot be used in the formation of the mass matrix; and therefore, this card is ignored. However, results are still reported at sampling points.

Sampling points must not coincide with boundary nodes (or other sampling points). If any sampling point coincides with another node, then a fatal error will occur.

Examples of Use -

1. Request that sampling points be included in the generation of the mass matrix. included.

```
**CASE
  TITLe  FREE VIBRATION OF A CANTILEVER BEAM
  FREE-vibration
  EIGEnvalue-option-1 SAMP
  TOLErance-convergence 0.005
  MAXImum-iterations 30
```

EIGEnvalue-option-2 SHIFT VALUE

Status - OPTIONAL

Full Keyword - **`EIGEnvalue-option-2`**

Function - Specify a shift in the frequency range for the eigenvalue search. Only eigenvalues above this value are extracted and reported.

Input Variables -

SHIFT (Alphanumeric) - REQUIRED

VALUE (Real) - REQUIRED

The shift frequency specified in Hertz.

Additional Information -

If this card is not input, then a default of zero hertz is assumed, and eigenvalues beginning at zero hertz are reported.

Note that the units are specified in Hertz, not rad/s.

Examples of Use -

1. Find the lowest 25 eigenvalues and mode shapes above 500 Hertz in a FREE-vibration analysis.

```
**CASE
  TITLe  Turbine Blade - Mode Shapes
  FREE-vibration
  EIGEnvalue-option-2 SHIFT 500.0
  EIGEnvalue OUTPUT   25   25
```

2. Request output of five eigenvalues and two eigenvectors above 2000 Hz.

```
**CASE
  TITLe  Free Vibration of a Crankshaft
  FREE-vibration
  EIGEnvalue-option-2 SHIFT 2000.0
  EIGEnvalue OUTPUT   5   2
```

5.1.12 NONLINEAR ALGORITHM CONTROL OPTIONS

ITERative

Status - OPTIONAL

Full Keyword - **`ITERative`**

Function - Selects the iterative algorithm for plasticity.

Input Variables - NONE

Additional Information -

The iterative algorithm is generally not recommended for nonlinear analysis. Instead, the Newton-Raphson algorithm is recommended. (See the NEWTon-raphson card on the next page.)

Examples of Use -

1. Plastic analysis of a notch plate.

```
**CASE
  TITLe  PLASTIC ANALYSIS OF A NOTCH PLATE
  PLANe STRAIN
  TIMEs-output   1.0   2.0   3.0
  ITERative
  MAXImum-iterations  20
```

NEWTon-raphson ITYPE NSKIP

Status - OPTIONAL

Full Keyword - **NEWTon-raphson**

Function - Selects the Newton-Raphson algorithm.

Input Variables -

ITYPE (Alphanumeric) - OPTIONAL

Allowable value is SKIP.

SKIP - Selects the Modified Newton-Raphson algorithm

A full Newton-Raphson algorithm is assumed if ITYPE = SKIP is not input.

NSKIP (Integer) - OPTIONAL (default is NSKIP = 1) Form a new Newton-Raphson matrix every ‘‘NSKIP’’ iterations.

Additional Information -

The Newton-Raphson algorithm is recommended for all nonlinear analysis.

If NEWTon-raphson or ITERative is not specified, then NEWTon-raphson is the default nonlinear solution algorithm for nonlinear problems.

In some cases, the use of the Modified Newton-Raphson algorithm can reduce analysis cost for nonlinear problems, however convergence is slower than for the full Newton-Raphson approach.

Setting NSKIP = 1 is equivalent to a full Newton-Raphson approach.

Examples of Use -

1. Select a modified Newton-Raphson algorithm for a transient thermoplastic analysis.

```
**CASE
  TITLe  CONTAINER - INELASTIC ANALYSIS
  CTHErmal TRANS PLASTIC
  TIMEs STEP  4  1.0
  NEWTon-raphson SKIP 4
  MAXImum-iterations  20
  RESTart WRITE $ THIS IS USEFUL FOR NONLINEAR PROBLEMS
```

5.2 MATERIAL PROPERTY DEFINITION

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

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

SECTION	KEYWORD	PURPOSE	PAGE
5.2.1	Material Property Input Card		5.2.7
	**MATERIAL	beginning of a material property input set	5.2.7
5.2.2	Material Identification		5.2.8
	ID-Material	identifier of a material type	5.2.8
5.2.3	Mass Parameter		5.2.9
	DENSity	material mass density	5.2.9
5.2.4	Isotropic Elastic Parameters		5.2.10
	EMODulus	Young's modulus	5.2.10
	ALPHa	Coefficient of thermal expansion	5.2.11
	POISson	Poisson's ratio	5.2.12
	SPEEd	speed of sound	5.2.13
5.2.5	Isotropic Thermal Parameters		5.2.14
	CONDuctivity	conductivity of material	5.2.14
	SPECific	specific heat	5.2.15
5.2.6	Isotropic Temperature-dependent Thermoelastic Parameters		5.2.16
	TEMPeratures	temperature values at which elastic material properties will be defined	5.2.16
	EMODulus-temp-dep	Young's modulus	5.2.18
	ALPHa-temp-dep	Coefficients of thermal expansion	5.2.19

SECTION	KEYWORD	PURPOSE	PAGE
5.2.7	Anisotropic Elastic Parameters		5.2.20
	ANISotropy	identifier for anisotropic material	5.2.20
	STIFfnesses	material stiffnesses	5.2.22
	COMPliances	material compliances	5.2.25
	TECHnical	technical constants for an material orthotropic	5.2.28
	ALPHa-axes	co-efficients of thermal expansion	5.2.30
	CONDuctivity-axes	conductivity tensor	5.2.31
	ORIEntation	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	5.2.32
5.2.8	Additional Isotropic Poroelastic Parameters		5.2.34
	FDENsity	fluid mass density	5.2.34
	PERMeability	permeability of the material	5.2.35
	SBULk	effective bulk modulus of solid grains	5.2.36
	UPOIsson	undrained Poisson's ratio	5.2.37
5.2.9	Isotropic Viscous Parameters		5.2.38
	DAMPing	viscous damping coefficient in elastodynamics	5.2.38
5.2.10	Elastoplastic and Viscoplastic Parameters		5.2.39
	INELastic	signals the beginning of an elastic material model input	5.2.39
5.2.10.1	Von Mises Model		5.2.40
	VON-mises	Von Mises material model	5.2.40
	YIELd	proportional limits of linear elastic behavior	5.2.41
	CURVe	stress-strain curve for Von Mises model with isotropic hardening	5.2.42
	(NONE)	define a single point on the stress-strain curve	5.2.43

SECTION	KEYWORD	PURPOSE	PAGE
5.2.10.2	Concurrent Thermoplastic Model	(Additional Parameters for Von Mises)	5.2.45
	TMELt	temperature for zero yield stress	5.2.45
	TPEAk	temperature for peak yield stress	5.2.46
5.2.10.3	Modified Cam-clay Soil Model		5.2.48
	MODIfied	select modified cam-clay model	5.2.48
5.2.10.4	Isotropic Plasticity Soil Model		5.2.49
	ISOIl	select isotropic plasticity soil model	5.2.49
		(Additional Soil Parameters:)	
	KAPPa	soil parameter - slope of rebound curve in e - ln p space (K)	5.2.50
	LAMBda	soil parameter - slope of normally consolidated line (NCL) in e - ln p space	5.2.51
	VOID-ratio	soil parameter - current void ratio	5.2.52
	PHI-soil	soil parameter - angle of internal friction	5.2.53
	CURRent-stress	current residual stress components to define existing stress state of the soil mass	5.2.54
	PAST-maximum-stress	past maximum consolidation stress for the soil mass	5.2.55
	COHEsion	cohesion of soil or rock	5.2.56

Note: Refer to the following table for a list of required material properties corresponding to a particular type of analysis.

Material Property Definition

A list of material properties required for different types of analysis are defined below:

REQUIRED MATERIAL PROPERTIES

TYPE OF ANALYSIS	MATERIAL PROPERTIES
1. Elastic Stress Analysis (Isotropic) and Distortion Analysis	EMODulus, POISson (TEMP: optional) (ALPHa-temp-dep: if thermal body force is present) (DENSity: if centrifugal body force is present) (DENSity: if inertial body force is present)
2. Elastic Stress Analysis (Anisotropic)	ANISotropy, STIFfnesses (or COMPliances , or TECHnical),ORIEntation (ALPHa-temp-dep: if thermal body force is present) (DENSity: if centrifugal body force is present) (DENSity: if inertial body force is present)
3. Heat Transfer Analysis (Isotropic)	
3a. Steady-state (Potential Flow)	CONDuctivity (DENSity, SPECific: if convective kernels are used)
3b. Transient (Diffusion)	CONDuctivity, DENSity, SPECific
4. Heat Transfer Analysis (Anisotropic)	
4a. Steady-state	ANISotropy, CONDuctivity, ORIEntation
4b. Transient	ANISotropy, CONDuctivity, ORIEntation, DENSity, SPECific
5. Concurrent Thermoelastic Analysis (Isotropic)	
5a. Steady-state	TEMPeratures, EMODulus, POISson, ALPHa-temp-dep, CONDuctivity (DENSity: if centrifugal body force is present) (DENSity: if inertial body force is present)
5b. Transient (Quasistatic)	TEMPeratures, EMODulus, POISson, ALPHa-temp-dep, CONDuctivity, DENSity, SPECific

TYPE OF ANALYSIS	MATERIAL PROPERTIES
6. Concurrent Thermoelastic Analysis (Anisotropic)	
6a. Steady-state	ANISotropy (TRAN) STIFfnesses (or COMPliances or TECHnical) ALPHa-temp-dep, CONDuctivity, ORIEntation (DENSity: if centrifugal body force is present) (DENSity: if inertial body force is present)
7. Poroelastic or Consolidation Analysis (Quasistatic)	EMODulus, POISson, PERMeability, FDENsity (UPOIsson, SBULk: optional)
8. Acoustic Analysis	SPEEd DENSity
9. Periodic Dynamic Analysis	EMODulus, POISson, DENSity, DAMPing (DAMPing: optional)
10. Plasticity Analysis	
10a. Von Mises Model	EMODulus, POISson, YIELd, CURVe (TEMPeratures: optional) (ALPHa-temp-dep: if thermal body force is present) (DENSity: if centrifugal body force is present) (DENSity: if inertial body force is present)
11. Nonlinear Consolidation	
11a. Modified Cam-clay Model	EMODulus, POISson, PERMeability, FDENsity, KAPPa, LAMBda, VOID-ratio, PHI-soil, CURRent-stress, PAST-maximum-stress (TEMPeratures: optional) (UPOIsson, SBULk: optional)
11b. Isotropic Plasticity Soil Model	EMODulus, POISson, PERMeability, FDENsity, KAPPa, LAMBda, VOID-ratio, PHI-soil, CURRent-stress, PAST-maximum-stress (TEMPeratures: optional) (COHEsion: optional) (UPOIsson, SBULk: optional)

Material Property Definition

12.	Current Thermoplastic Analysis	
	12a. Steady-state	TEMPeratures, EMODulus, POISson, ALPHa-temp-dep, CONDuctivity, TMELt, TPEAk (Von Mises Model: YIELd, CURVe)
	12b. Transient (Quasistatic)	TEMPeratures, EMODulus, POISson, ALPHa-temp-dep, CONDuctivity, DENSity, SPECific, TMELt, TPEAk (Von Mises Model: YIELd, CURVe)
13.	Structural Acoustics	SPEEd, DENSity (of fluid) EMODulus, POISson (of structure)

5.2.1 MATERIAL PROPERTY INPUT CARD

**MATERIAL

Status - REQUIRED

Full Keyword - **MATERIAL**

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.

```
**MATERIAL
  ID-Material STEEL
    EMODulus   30.3+6   $ (psi)
    POISson   0.30

  ID-Material ALUM3003
    CONDuctivity   25.0         $ lbf/(s F)
    DENSity   0.000253     $ (lbf s^2)/(in^4)
    SPECific   7.8E5        $ (in^2)/(s^2 F)

  ID-Material ALUM
    EMODulus     10.E6
    ALPHa        13.E-6
    POISson       0.33
```

5.2.2 MATERIAL IDENTIFICATION

ID-Material NAME

Status - REQUIRED

Full Keyword - **`ID-Material`**

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-Material ALUM3003
    CONDuctivity   25.0         $ lbf/(s F)
    DENSity   0.000253     $ (lbf s^2)/(in^4)
    SPECific   7.8E5        $ (in^2)/(s^2 F)
```

5.2.3 MASS PARAMETERS

DENSity DEN1

Status - (see required material property table)

Full Keyword - **DENSity**

Function - Defines the material mass density.

Input Variables -

DEN1 (Real) - REQUIRED

Additional Information - Average density of some common materials:

Material	Mass Density (kg/m^3)
Air	1.12
Water	1000
Steel	7850
Aluminum Alloy	2700
Brass/Bronze	8500
Cast Iron	7200
Reinforced Concrete	2400
Copper	8900
Glass	2600

Although mass density is commonly given in terms of mass per unit volume, these units must be converted to force-compatible units. See the examples below.

Examples of Use -

1. Define material properties for an elastic analysis with centrifugal body force.

```
**MATERIAL
  ID-Material  STEEL
    EMODulus   30.E+6       $ (psi)
    POISson   0.30
    DENSity   7.324E-4     $ (lbf s^2)/(in^4)
```

2. Define material properties for the previous problem using metric units.

```
**MATERIAL
  ID-Material  STEEL
    EMODulus   2.07E+5      $ N/(mm^2)
    POISson   0.30
    DENSity   7.854E-9     $ (N s^2)/(mm^4)
```

5.2.4 ISOTROPIC ELASTIC PARAMETERS

EMODulus EM1

Status - (see required material property table)

Full Keyword - **`EMODulus`**

Function - Defines values of Young's modulus

Input Variables -

EM1 (Real) - REQUIRED

Additional Information - Average Young's modulus value of some common materials:

Material	Young's Modulus Value (GPa)
Steel	210
Aluminum Alloy	75
Brass/Bronze	100
Cast Iron	100
Reinforced Concrete	25
Copper	120
Glass	45

Examples of Use -

1. Specify elastic material properties for aluminum.

```
**MATERIAL
  ID-Material  MAT1
    EMODulus     7.5E4      $ N/(mm^2)
    POISson     0.36
    DENSity     2.7E-9     $ (N s^2)/(mm^4)
```

2. Specify the above material properties in English units.

```
**MATERIAL
  ID-Material  MAT1
    EMODulus     1.0E7      $ (psi)
    POISson     0.36
    DENSity     2.53E-4    $ (lbf s^2)/(in^4)
```

ALPHa AL1

Status - REQUIRED (for thermal stress analysis)

Full Keyword - **ALPHa**

Function - Defines the values of the coefficient of thermal expansion.

Input Variables -

AL1 (Real) - REQUIRED

Additional Information -

NONE

Examples of Use -

1. Define a temperature independent thermoelastic material model.

```
**MATERIAL
  ID-Material ALUM
    EMODulus     10.E6
    ALPHa        13.E-6
    POISson       0.33
```

POISson POI

Status - (see required material property table)

Full Keyword - **POISson**

Function - Defines the (temperature independent) value of Poisson's ratio.

Input Variables -

POI (Real) - REQUIRED

Allowable values - $-1.0 < POI \leq 0.5$

Additional Information - Average Poisson's ratio of some common materials:

Material	Poisson's Ratio
Steel	0.3
Aluminum Alloy	0.33
Brass/Bronze	0.34
Cast Iron	0.25
Reinforced Concrete	0.2
Copper	0.33
Glass	0.22
Rubber	0.5

Poisson's ratio is a dimensionless quantity.

A Poisson's ratio of 0.5 causes no particular difficulty in **GPBEST** since this material parameter is included in the elastic Green's function.

Examples of Use -

1. Specify room temperature elastic properties of carbon steel.

```
**MATERIAL
  ID-Material  STEEL
    EMODulus  30.E6     $ (psi)
    POISson  0.30
```

SPEEd SPD

Status - (see required material property table)

Full Keyword - **SPEEd**

Function - Defines the value of speed of sound.

Input Variables -

SPD (Real) - REQUIRED

Additional Information - Speed of sound through some common fluids:

Fluid	Speed of Sound (m/s)
Air	342
Water	1500

Examples of Use -

1. Specify speed of sound

```
**MATERIAL
  ID-Material  AIR
  SPEEd  3.42E5       $ (mm/s)
```

5.2.5 ISOTROPIC THERMAL PARAMETERS

CONDuctivity CD1

Status - REQUIRED (for concurrent thermoelasticity or heat conduction)

Full Keyword - **CONDuctivity**

Function - Defines the isotropic conductivity.

Input Variables -

CD1 (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify thermal properties of aluminum for steady-state heat conduction.

```
**MATERIAL
  ID-Material  ALUM
    CONDuctivity  204.0     $ N/(s C)
```

2. Specify thermoelastic properties for a quasistatic analysis.

```
**MATERIAL
  ID-Material  M200
    TEMPeratures  500.0       $ F
    EMODulus  1.0E+7      $ (psi)
    POISson  0.24
    ALPHa-temp-dep  1.E-5       $ 1/F
    CONDuctivity  25.0        $ lbf/(s F)
    DENSity  2.5E-4      $ (lbf s^2)/(in^4)
    SPECific  7.7E5       $ (in^2)/(s^2 F)
```

SPECific SP1

Status - REQUIRED (for transient concurrent thermoelasticity or heat conduction)

Full Keyword - **SPECific**

Function - Defines the specific heat.

Input Variables -

SP1 (Real) - REQUIRED

Additional Information -

The user must be careful in selecting appropriate units for specific heat. The CONDuctivity divided by the product of DENSity times SPECific equals the diffusivity. The diffusivity must have units of (length**2)/time.

Examples of Use -

1. Material model for transient heat conduction.

```
**MATERIAL
  ID-Material  STEEL
    CONDuctivity   5.8       $ (in lbf)/(s in F)
    DENSity   7.329E-4  $ (lbf s^2)/(in^4)
    SPECific   4.07E5    $ (in^2)/(s^2 F)
```

2. Redo the above example with Metric units.

```
**MATERIAL
  ID-Material  STEEL
    CONDuctivity   54.       $ (N mm)/(s mm C)
    DENSity   7.85E-9   $ (N s^2)/(mm^4)
    SPECific   4.65E8    $ (mm^2)/(s^2 C)
```

5.2.6 TEMPERATURE-DEPENDENT ISOTROPIC THERMOELASTIC PARAMETERS

TEMPeratures 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 TEMPeratures.

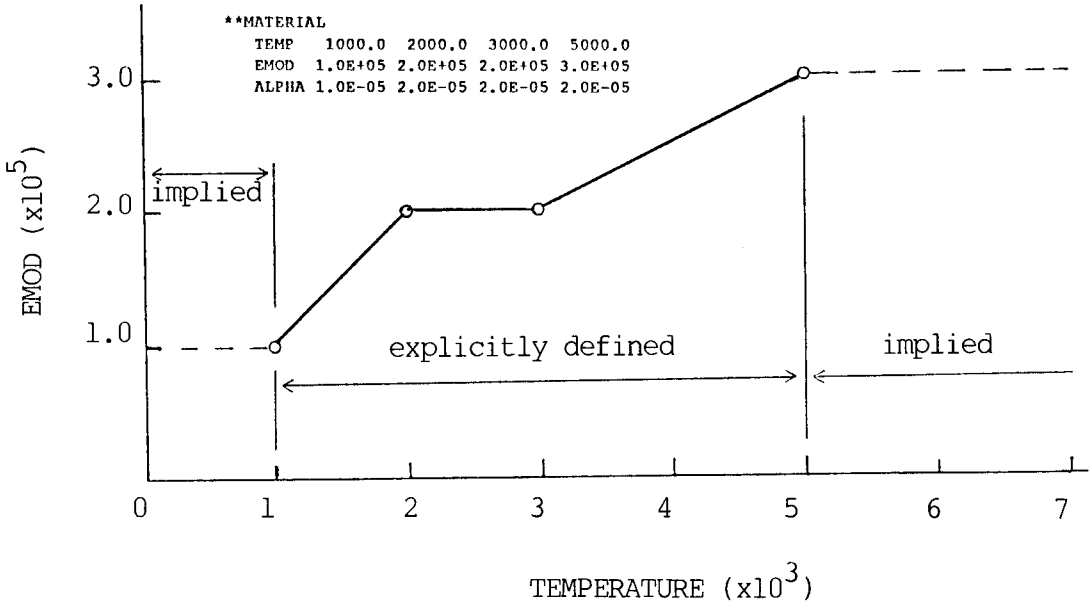
A maximum of 21 temperature values may be specified for the material.

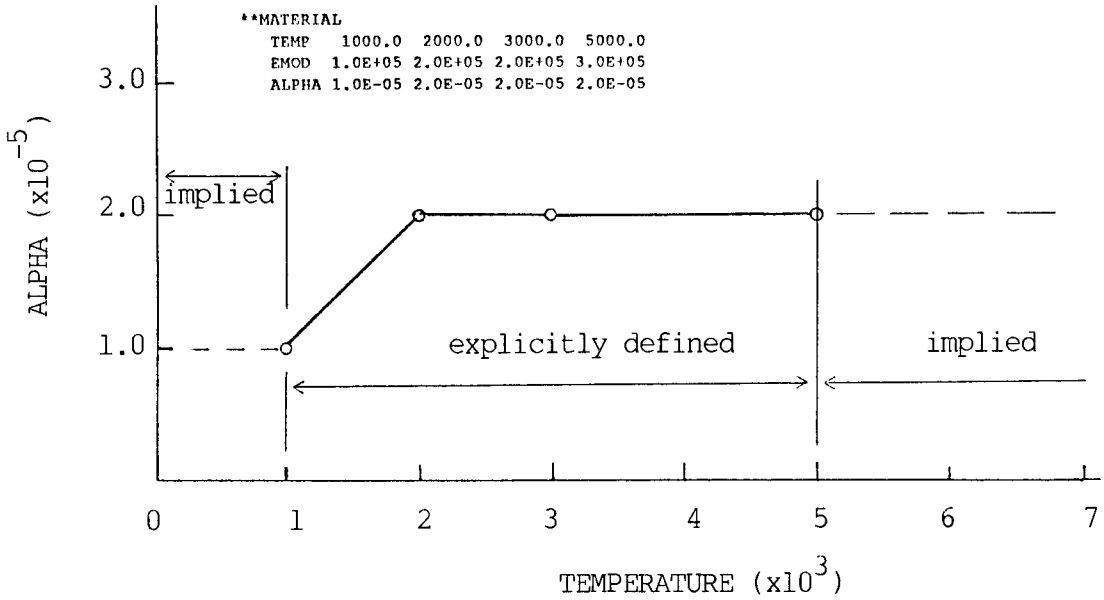
It is important to note that for anisotropy, no more than one value of temperature, viz, TEM1 can be given, i.e., the elastic anisotropic material properties are essentially kept as temperature independent in this version of the code.

Examples of Use -

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

```
**MATERIAL
  ID-Material  MAT0IT
    TEMPeratures  50.0  300.0  500.0                $ (F)
    EMODulus-temp-dep  30.3E+6  29E+6  27.5E+6  $ (psi)
    POISson  0.3
```





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

Definition for temperature dependent elastic modulus and thermal coefficient of expansion

EMODulus-temp-dep EM1 EM2 ... EMN

Status - (see required material property table)

Full Keyword - **EMODulus-temp-dep**

Function - Defines values of Young's modulus at the temperature values specified on the TEMPeratures 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-Material  MAT1
    TEMPeratures     0.0     200.0     300.0     350.0 $ (F)
    TEMPeratures   400.0     450.0     500.0
    EMODulus-temp-dep 1.E7  0.95E7 0.9E7   0.83E7 $ (psi)
    EMODulus-temp-dep 0.7E7 0.6E7  0.35E7
    POISson     0.36
    DENSity     2.5E-4   $ (lbf s^2)/(in^4)
```

ALPHa-temp-dep AL1 AL2 ... ALN

Status - REQUIRED (for thermal stress analysis)

Full Keyword - **`ALPHa-temp-dep`**

Function - Defines the values of the coefficient of thermal expansion at the temperature values specified on TEMPeratures cards. (Note the difference with the format of input variables when this card appears in the ANISotropy section)

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 ALPHa-temp-dep.

Examples of Use -

1. Define a thermoelastic material model.

```
**MATERIAL
  ID-Material ALUM
    TEMPeratures     0.0     200.0     300.0      $ (F)
    EMODulus-temp-dep     10.E6   9.8E6     8.E6    $ (psi)
    ALPHa-temp-dep     13.E-6  12.E-6    10.5E-6    $ (1/F)
    POISson     0.33
```

5.2.7 ANISOTROPIC ELASTIC PARAMETERS

ANISotropy 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 for ELAStic analysis

Allowable values are GENE, ORTH, TRAN and CUBI

GENE - General anisotropy. In 3D, general anisotropy requires 21 independent constants. In 2D, the general anisotropy is a plane of elastic symmetry about the x-y plane. This requires 10 independent constants in 2D problems.

ORTH - Orthotropy or Orthogonal anisotropy (three planes of elastic symmetry). In 3D, orthotropy requires 9 independent constants. In 2D, orthotropy requires 7 independent constants.

TRAN - Transverse isotropy or Cross isotropy or Plane of isotropy. (Directionally solidified material.) In 3D, this requires 5 independent constants. In 2D, this requires 4 independent constants.

CUBI - Cubic crystal anisotropy. This requires 3 independent constants in both 2D and 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 selected.

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 ORIEntation card) with respect to the global coordinate system which is used to define the geometry of the body.

If geometric and loading symmetry (the SYMMetry 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 SYMMetry card should not be used with general anisotropy.

When data is input using the cylindrical COORdinate-system system, the material must be directionally independent in the R-θ plane.

Anisotropy is not available for CTHErmal or CONSolidation analysis.

Examples of Use -

1. Define option for 2D anisotropic elastic analysis together with the type of anisotropy.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  10.0
  ANISotropy  ORTHO
  TECHnical 1.31E+5 0.13E+5 0.13E+5 0.038 0.038 0.492 0.064E+5
```

STIFfnesses S11 ... SNN

Status - OPTIONAL (if ANISotropy is input, either STIFfnesses, COMPliances or TECHnical 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 (3D)	S11, S12, S13, S14, S15, S16, S22, S23, S24, S25, S26 S33, S34, S35, S36, S44, S45, S46, S55, S56, S66
GENE (2D)	S11, S12, S13, S14, S22, S23, S24, S33, S34, S44
ORTH (3D)	S11, S12, S13, S22, S23, S33, S44, S55, S66
ORTH (2D)	S11, S12, S13, S22, S23, S33, S44
TRAN (3D)	S11, S13, S33, S44, S55
TRAN (2D)	S11, S13, S33, S44
CUBI (3D)	S11, S12, S44
CUBI (2D)	S11, S12, S44

Additional Information -

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

GENE - General Anisotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{13} & S_{14} & S_{15} & S_{16} \\ & S_{22} & S_{23} & S_{24} & S_{25} & S_{26} \\ & & S_{33} & S_{34} & S_{35} & S_{36} \\ & & & S_{44} & S_{45} & S_{46} \\ & Symm. & & & S_{55} & S_{56} \\ & & & & & S_{66} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

In 2D the general anisotropy is a plane of elastic symmetry about the x-y plane. In this case $S_{15} = S_{16} = S_{25} = S_{26} = S_{35} = S_{36} = S_{45} = S_{46} = 0$.

ORTH -Orthotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{13} & 0 & 0 & 0 \\ & S_{22} & S_{23} & 0 & 0 & 0 \\ & & S_{33} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{55} & 0 \\ & & & & & S_{66} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

TRAN - Transverse Isotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & (S_{11}-2S_{44}) & S_{13} & 0 & 0 & 0 \\ & S_{11} & S_{13} & 0 & 0 & 0 \\ & & S_{33} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{55} & 0 \\ & & & & & S_{55} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

CUBI - Cubic Crystal Anisotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{12} & 0 & 0 & 0 \\ & S_{11} & S_{12} & 0 & 0 & 0 \\ & & S_{11} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{44} & 0 \\ & & & & & S_{44} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

IMPORTANT NOTES:

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

  For convenience, the stiffness matrix may be defined with respect to an arbitrary material axis and then the actual orientation of the material axes is defined with respect to the global(geometric) axes using the ORIEntation card.

- In 2D, the deformation is restricted to the x-y plane, so the most general anisotropy is a plane of elastic symmetry about the x-y plane. The stress-strain relationships for 2D can therefore be derived from the above constitutive laws by deleting the fifth and sixth rows and columns. **2D anisotropy must be input using the convention that the x-y plane is the plane of elastic symmetry**.

- 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 ORIEntation card.

- **The coefficients in the above matrices represent the real material stiffness for both 2D and 3D analyses and should not be confused with the modified (or reduced) coefficients that are sometimes used to represent stiffnesses in plane stress analysis**.

- The material stiffnesses appearing on the STIFfnesses card may be split and put onto two or more cards with each new card beginning with the keyword STIFfnesses. 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.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy TRAN
  STIFfnesses 2.1601E+7 0.9691E+6 2.0832E+7 0.65E+6 1.0279E+6
```

2. Input stiffness coefficients relevant to a 2D general anisotropy analysis (a plane of elastic symmetry) relative to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy GENE
  STIFfnesses  1.1601E+7   0.9691E+6   0.9691E+6   0.  2.0832E+6
  STIFfnesses  1.0430E+6   0.          2.0832E+6   0.  0.65E+6
```

3. Input stiffness coefficients relevant to a 2D analysis for an orthotropic medium relative to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy ORTH
  STIFfnesses  2.1601E+7   0.9692E+6   0.9691E+6   2.0832E+6
  STIFfnesses  1.0430E+6   2.0832E+6   0.65E+6
```

COMPliances C11 ... CNN

Status - OPTIONAL (if ANISotropy is input, either STIFfnesses, COMPliances or TECHnical 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 (3D)	C11, C12, C13, C14, C15, C16, C22, C23, C24, C25, C26 C33, C34, C35, C36, C44, C45, C46, C55, C56, C66
GENE (2D)	C11, C12, C13, C14, C22, C23, C24, C33, C34, C44
ORTH (3D)	C11, C12, C13, C22, C23, C33, C44, C55, C66
ORTH (2D)	C11, C12, C13, C22, C23, C33, C44
TRAN (3D)	C11, C13, C33, C44, C55
TRAN (2D)	C11, C13, C33, C44
CUBI (3D)	C11, C12, C44
CUBI (2D)	S11, S12, S44

Additional Information -

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

GENE - General Anisotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ & & C_{33} & C_{34} & C_{35} & C_{36} \\ & & & C_{44} & C_{45} & C_{46} \\ & Symm. & & & C_{55} & C_{56} \\ & & & & & C_{66} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

In 2D the general anisotropy is a plane of elastic symmetry about the x-y plane. In this case $C_{15} = C_{16} = C_{25} = C_{26} = C_{35} = C_{36} = C_{45} = C_{46} = 0.$

ORTH -Orthotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ & C_{22} & C_{23} & 0 & 0 & 0 \\ & & C_{33} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symm. & & & C_{55} & 0 \\ & & & & & C_{66} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

TRAN - Transverse Isotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & (C_{11} - C_{44}/2) & C_{13} & 0 & 0 & 0 \\ & C_{11} & C_{13} & 0 & 0 & 0 \\ & & C_{33} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symm. & & & C_{55} & 0 \\ & & & & & C_{55} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

CUBI - Cubic Crystal Anisotropy

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

IMPORTANT NOTES:

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

  For convenience, the compliance matrix may be defined with respect to an arbitrary material axis and then the actual orientation of the material axes is defined with respect to the global (geometric) axes using the ORIEntation card.

- In 2D, the deformation is restricted to the x-y plane, so the most general anisotropy is a plane of elastic symmetry about the x-y plane. The stress-strain relationship for 2D can therefore be derived from the above constitutive laws by deleting the fifth and sixth, rows and columns. **2D anisotropy must be input using the convention that the x-y plane is the plane of elastic symmetry**.

- 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 ORIEntation card.

- **The coefficients in the above matrices represent the real material compliances for both 2D and 3D analyses and should not be confused with the modified (or reduced) coefficients that are sometimes used to represent compliances in plane strain analysis**.

- The material compliances appearing on the COMPliances card may be split and put onto two or more cards with each new card beginning with the keyword COMPliances. 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.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy TRAN
  COMPliances 4.7619E-7 -1.4762E-8 6.4516E-7 1.5385E-8 7.2750E-8
```

2. Input compliance coefficients relevant to a 2D general anisotropy analysis (a plane of elastic symmetry) relative to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy GENE
  COMPliances  4.7619E-7 -1.4762E-8 -1.4762E-8 0. 6.4516E-7
  COMPliances -3.1613E-7  0.         6.4516E-7 0. 1.5385E-7
```

3. Input compliance coefficients relevant to a 2D analysis for an orthotropic medium relative to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy ORTH
  COMPliances  4.7619E-7   -1.4762E-8   -1.4762E-8   6.4516E-7
  COMPliances -3.1613E-7    6.4516E-7    1.5385E-7
```

TECHnical E1 E2 E3 ν12 ν13 ν23 G12 G13 G23

Status - OPTIONAL (if ANISotropy is input, either STIFfnesses, COMPliances, or TECHnical 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 (for 3D analysis)

E1,E2,E3,ν12,ν13,ν23,G12 (Real) - REQUIRED (for 2D analysis)

Additional Information -

The TECHnical 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:

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

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**.

  For convenience, the technical constants may be defined with respect to an arbitrary material axis then the actual orientation of the material axes is defined with respect to the global (geometric) axes using the ORIEntation card.

- In 2D, the deformation is restricted to the x-y plane, so the most general anisotropy is a plane of elastic symmetry about the x-y plane. The stress-strain relationships for 2D can therefore be derived from the above constitutive laws by deleting the fifth and sixth rows and columns. **2D anisotropy must be input using the convention that the x-y plane is the plane of elastic symmetry.**

- **The coefficients in the above matrices represent the real material parameters for both 2D and 3D analyses, and should not be confused with the modified (or reduced) parameters that are sometimes used to represent the material parameters in plane stress analysis.**

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

Examples of Use -

1. Input technical constants (also called ‘engineering constants’) relevant to a 2D analysis for an orthotropic medium relative to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy  ORTH
  TECHnical  21.0E+6   1.55E+6   1.55E+6
  TECHnical  0.31      0.31      0.49      0.65E+6
```

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

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy  ORTH
  TECHnical  21.0E+6   1.55E+6   1.55E+6
  TECHnical  0.31      0.31      0.49
  TECHnical  0.65E+6   0.5E+6    0.5E+6
```

ALPHa-axes $\alpha 11\ \alpha 22\ \alpha 33\ \alpha 12\ \alpha 13\ \alpha 23$

Status - OPTIONAL

Full Keyword - **`ALPHa-axes`**

Function - Inputs coefficients of thermal expansion relative to the material axes for an **anisotropic** material.

Input Variables -

$\alpha 11, \alpha 22, \alpha 33, \alpha 12$ (Real) - REQUIRED (for 2-D analysis)

$\alpha 11, \alpha 22, \alpha 33, \alpha 12, \alpha 13, \alpha 23$ (Real) - REQUIRED (for 3-D analysis)

Additional Information -

This card is required in a material definition if there are thermal body forces present in any region in which the material set is being used. The relationship between the alpha components and strains is defined below in terms of Engineering strains.

$$\begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} \alpha_{11} \\ \alpha_{22} \\ \alpha_{33} \\ \alpha_{12} \\ \alpha_{13} \\ \alpha_{23} \end{pmatrix} * Temperature$$

The alpha components must be input relative to the same material axis for which the STIF, COMPliances, or TECHnical card is input. The actual orientation of the material axis with respect to the global (geometric) axes are defined using the ORIEntation card.

The coefficients in the above matrices represent the real thermal coefficients of expansion for both 2-D and 3-D analysis and should not be confused with the modified (or reduced) coefficients that are sometimes used in plane stress or plane strain analysis.

Examples of Use -

1. Give coefficients of thermal expansion for a 2D orthotropic medium relevant to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy  ORTHO
  TECHnical  100.0001   100.0   100.0   0.25   0.25   0.25   40.0
  ALPHa-axes  1.0   1.0   1.0   0.0
```

CONDuctivity-axes K11 K22 K33

Status - OPTIONAL

Full Keyword - **`CONDuctivity-axes`**

Function - Defines the conductivity tensor relative to the material axes for **anisotropic** material.

Input Variables -

K11, K22 (Real) - REQUIRED (for 2-D analysis)

K11, K22, K33 (Real) - REQUIRED (for 3-D analysis)

Additional Information -

This card is required in a material definition of a HEAT or three-dimensional cross-anisotropic CTHErmal analysis. The relationship between the flux and temperature is

$$\begin{pmatrix} f_1 \\ f_2 \\ f_3 \end{pmatrix} = - \begin{pmatrix} K_{11} & 0 & 0 \\ 0 & K_{22} & 0 \\ 0 & 0 & K_{33} \end{pmatrix} \begin{pmatrix} \theta_{,1} \\ \theta_{,2} \\ \theta_{,3} \end{pmatrix}$$

The conductivity tensor must be input relative to the same material axis for which the STIF, COMPliances, or TECHnical card is input. The actual orientation of the material axis with respect to the global (geometric) axes are defined using the ORIEntation card.

Examples of Use -

1. Give the conductivity tensor for a 2D medium relevant to the principal material axes.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy GENE
  CONDuctivity-axes  1.0  2.0
```

ORIEntation 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). In 2D this represents a rotation about the z-axis. In 3D, a full three-dimensional rotation is possible.

Input Variables -

A1 (Real) - REQUIRED (in 2D analysis)

A1,A2,A3,B1,B2,B3 (Real) - REQUIRED (in 3D analysis)

where in 2D

A1 represents the angle **(in degrees)** that the material axes is rotated with respect to the global (geometric) coordinate system. Counterclockwise is positive (right-hand rule).

where in 3D

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.

If geometric and loading symmetry (the SYMMetry 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 SYMMetry card should not be used with general anisotropy.

In 3D, the x-axis and the y-axis defined on the ORIEntation card must be at right angles. The third (z) axis will be calculated by GPBEST .

Examples of Use -

1. For 2D orthotropy, provide the angle (in degrees) of inclination of the material principal axis with respect to the global coordinate x-axis. This angle is to be measured in a counter clockwise direction from the x-axis.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy  ORTHO
  TECHnical 1.31E+5 0.13E+5 0.13E+5 0.038 0.038 0.492 0.064E+5
  ORIEntation  45.0
```

2. For 3D transverse isotropy, provide the directional cosines for material axes at an oblique rotation.

```
**MATERIAL
  ID-Material  MAT1
  DENSity  1.0
  ANISotropy  TRAN
  COMPliances 4.76E-7 -1.47E-8 6.46E-7  1.538E-7 7.27E-7
  ORIEntation 0.7071   0.7071  0.7071  -0.7071   0.7071  0.7071
```

5.2.8 ADDITIONAL ISOTROPIC POROELASTIC PARAMETERS

FDENsity FD1

Status - OPTIONAL (for poroelastic or consolidation analysis)

Full Keyword - **FDENsity**

Function - Defines the fluid mass density.

Input Variables -

FD1 (Real) - REQUIRED (Default is 1.0)

Additional Information - NONE

Examples of Use -

1. Define a saturated poroelastic material.

```
**MATERIAL
  ID-Material  SOIL
    EMODulus   10.        $ N/(mm^2)
    POISson   0.1        $
    FDENsity   0.01       $ (N s^2)/(mm^4)
    PERMeability   0.1        $ mm/s
```

PERMeability PM1

Status - REQUIRED (for poroelastic or consolidation analysis)

Full Keyword - **PERMeability**

Function - Defines the isotropic permeability of the material.

Input Variables -

PM1 (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify a material model for a consolidation analysis.

```
**MATERIAL
  ID-Material  CONSOL
    EMODulus   1.92E+01
    POISson   0.3
    PERMeability   5.80E-01
    FDENsity   64.0
    UPOIsson   0.5
```

SBULk SB1

Status - OPTIONAL (for poroelastic or consolidation analysis)

Full Keyword - **SBULk**

Function - Defines the effective bulk modulus of the isotropic solid.

Input Variables -

SB1 (Real) - REQUIRED (Default is 1.0)

Additional Information - NONE

Examples of Use -

1. Define a poroelastic material model for fractured granite layer.

```
**MATERIAL
  EMODulus  75.0E+6
  POISson  0.25
  PERMeability  2.04E-5
  FDENsity  1.0E+4
  SBULk  70.3E+6
  UPOIsson  0.30
```

UPOIsson UP1

Status - OPTIONAL (for poroelastic or consolidation analysis)

Full Keyword - **UPOIsson**

Function - Defines the undrained Poisson ratio.

Input Variables -

UP1 (Real) - REQUIRED (Default is 0.5)

Additional Information - NONE

Examples of Use -

1. Define a partially saturated soil.

```
**MATERIAL
  ID-Material  PSSOIL
    EMODulus  2394.0
    POISson  0.25
    PERMeability  1.22E-7
    FDENsity  8.3
    UPOIsson  0.36
```

5.2.9 ADDITIONAL VISCOUS PARAMETERS

DAMPing DAMPR

Status - REQUIRED (for steady-state forced vibration 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-Material MAT1
    EMODulus   2.6666
    POISson   0.3333
    DENSity 1.0
    DAMPing 0.05
```

5.2.10 ADDITIONAL ELASTO-PLASTIC PARAMETERS

INELastic

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 INELastic card is used during a material property definition, the program will then expect one of three models to be chosen. These models are specified using the VON-mises, MODIfied, or ISOIl keywords. Only one of these three cards may be used in a single material property input data set. Input for the three nonlinear models is described below.

NOTE: In present version of the program, all non-linear materials used in an analysis must be based on the same model. However, the material properties may differ.

Examples of Use -

1. Request an inelastic analysis using the Von Mises Material Model.

```
**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus 7000.
  POISson 0.2
  INELastic
    VON-mises
```

5.2.10.1	VON MISES MODEL

VON-mises

Status - OPTIONAL

Full Keyword - **VON-mises**

Function - Selects the use of the inelastic Von Mises Model in which the stress-strain curve is defined as a set of equivalent stress-equivalent plastic strain pairs, with linear interpolation between points.

Input Variables - NONE

Additional Information -

The input for this model is extremely easy to generate if a monotonic stress-strain curve is available for the material. The model is intended primarily for the analysis of monotonic loading situations.

Examples of Use -

1. Request the use of the Von Mises Model for an inelastic analysis.

```
**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus 7000.0
  POISson 0.2
  INELastic
    VON-mises
```

YIELd Y1

Status - REQUIRED (if VON-mises is input)

Full Keyword - **YIELd**

Function - Defines the value of initial yielding strength σ^o.

Input Variables -

Y1 (Real) - REQUIRED

Initial yielding strength σ^o

Examples of Use -

1. Specify the yield stress value at which the material ceases to behave in an elastic manner.

```
**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus 7000.0
  POISson 0.2
  INELastic
    VON-mises
    YIELd 24.3
```

CURVe

Status - REQUIRED (with VON-mises)

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 CURVe is input)

Full Keyword - NO KEYWORD REQUIRED

Function - To define a single point on the material stress strain curve.

Input Variables -

SIG1 (Real) - REQUIRED

Stress value

EP1 (Real) - REQUIRED

Equivalent plastic strain value

Additional Information -

This card type is input once for each point used to describe the stress-strain curve. The stress values must be monotonically increasing. The point (yield stress,0) is inserted automatically by the program and must not be input by the user.

In order to define a perfectly plastic material the user should input a single point at which the stress is the yield stress and the equivalent plastic strain has any positive value.

A maximum of nineteen points (excluding the initial point) can be input by the user.

Linear interpolation is used between points.

Examples of Use -

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

```
**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus 70000.0
  POISson 0.2
  INELastic
    VON-mises
    YIELd 850.0
    CURVe
      1425.0   0.005
      1775.0   0.012
      2000.0   0.020
```

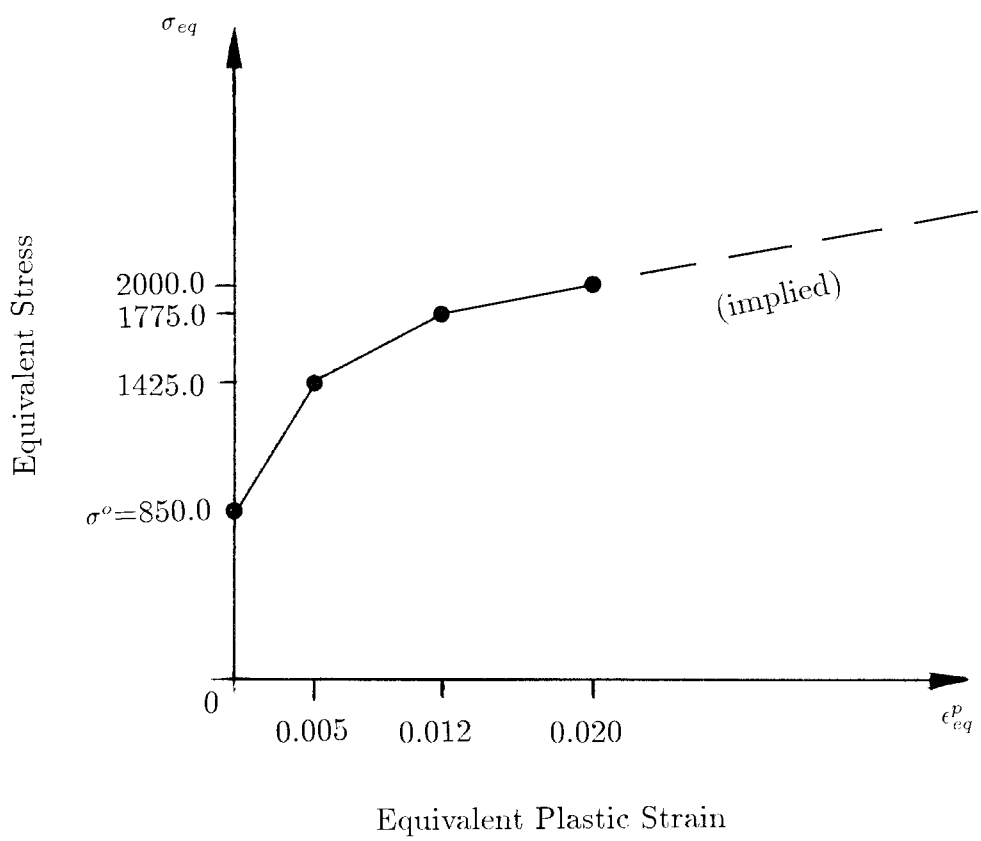


Figure for Von Mises Material Stress-Strain Curve, Defined Above

5.2.10.2 Concurrent Thermoplastic Model

TMELt TM1

Status - OPTIONAL (for concurrent thermoplastic analysis)

Full Keyword - **TMELt**

Function - Defines the temperature for zero yield stress.

Input Variables -

TM1 (Real) - REQUIRED (Default is 100000.0)

Additional Information - NONE

Examples of Use -

1. Specify the temperature value at which the yield stress reduces to zero.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 0.0           $ F
    EMODulus 10.0E+06      $ psi
    POISson 0.33
    DENSity 2.5E-4        $ (lbf s^2)/(in^4)
    ALPHa-temp-dep 13.0E-06      $ 1/F
    SPECific 7.7E5         $ (in^2)/(s^2 F)
    CONDuctivity 25.0          $ lbf/(s F)
  INELastic
    VON-mises
    YIELd 1000.0       $ psi
    CURVe
      2000. 5.00E-03   $ psi   in/in
    TMELt 1700.        $ F
    TPEAk 500.0        $ F
```

TPEAk TP1

Status - OPTIONAL (for concurrent thermoplastic analysis)

Full Keyword - **TPEAk**

Function - Defines the temperature up to which the yielding stress remains unchanged.

Input Variables -

TP1 (Real) - REQUIRED (Default is 0.0)

Additional Information - NONE

Examples of Use -

1. Specify the temperature value for peak yield stress.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 0.0           $ F
    EMODulus 10.0E+06      $ psi
    POISson 0.33
    DENSity 2.5E-4        $ (lbf s^2)/(in^4)
    ALPHa-temp-dep 13.0E-06      $ 1/F
    SPECific 7.7E5         $ (in^2)/(s^2 F)
    CONDuctivity 25.0          $ lbf/(s F)
  INELastic
    VON-mises
    YIELd 1000.0       $ psi
    CURVe
      2000. 5.00E-03   $ psi   in/in
    TMELt 1700.        $ F
    TPEAk 500.0        $ F
```

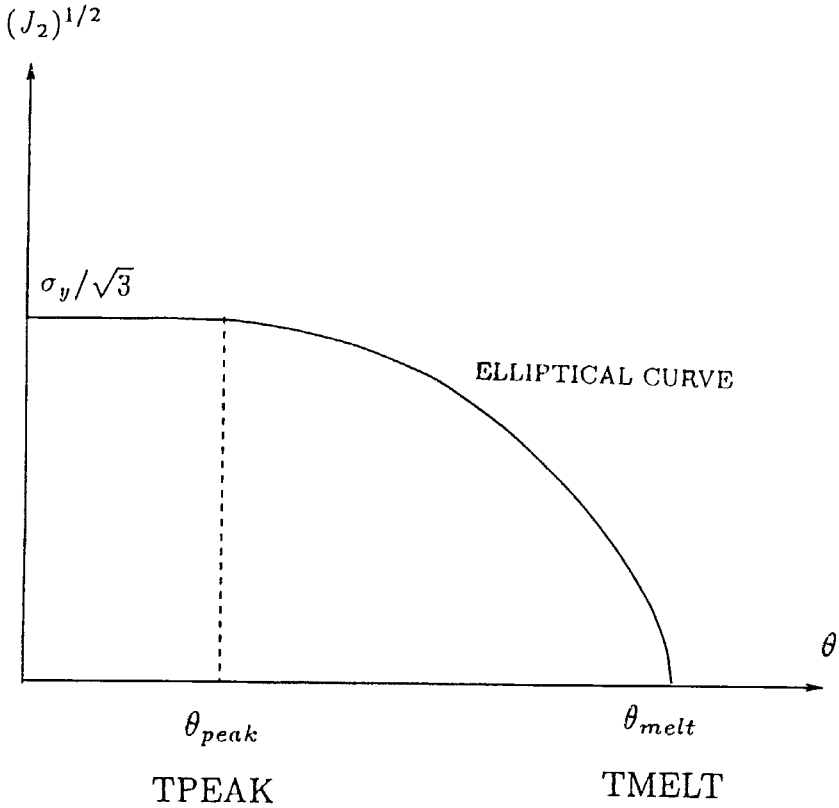


Figure for Thermally-sensitive Von Mises Yield Surface

5.2.10.3	MODIFIED CAM-CLAY MODEL

MODIfied

Status - OPTIONAL

Full Keyword - **MODIfied**

Function - Selects the use of Modified Cam-clay soil plasticity model

Input Variables - NONE

Additional Information -

This model is based upon the original Modified CAMCLAY model. It should be used for the analysis of clays under isotropic consolidation subjected to axisymmetric, plane strain or general 3-D loading conditions.

If this card is used, then the following material property cards must be specified:
KAPPa, LAMBda, VOID-ratio, PHI-soil,CURRent-stress, PAST-maximum-stress.

Examples of Use -

1. Selects the use of the Modified Cam-clay soil plasticity model for an inelastic analysis.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    MODIfied CAM-CLAY MODEL
```

5.2.10.4 ISOTROPIC PLASTICITY SOIL MODEL

ISOIl

Status - OPTIONAL

Full Keyword - **`ISOIl`**

Function - Selects the use of the isotropic plasticity soil model

Input Variables - NONE

Additional Information -

This model is based on the Mohr-Coulomb failure criteria. It is also to be used for the analysis of clays under isotropic consolidation. The cohesion is included in this model.

If this card is used, then the following material property cards must be specified for each GMR:

KAPPa, LAMBda, VOID-ratio, PHI-soil, CURRent-stress, PAST-maximum-stress, and COHEsion is optional.

Examples of Use -

1. Selects the use of isotropic plasticity soil model.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    ISOIl
```

KAPPa KAP

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **KAPPa**

Function - Soil parameter defining the slope of the swelling or rebound curve from the e-lnp data.

Input Variables -

KAP (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Define the soil parameter for an inelastic analysis.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    MODIfied CAM-CLAY MODEL
$   MATERIAL PROPERTIES FOR MODIFIED CAM-CLAY MODEL:
    KAPPa 0.05
```

LAMBda LAM

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **LAMBda**

Function - Soil parameter defining the slope of the normally consolidated line (NCL) in e-lnp space for each GMR.

Input Variables -

LAM (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify the value of the soil property Lambda from the e-lnp curve.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    MODIfied CAM-CLAY MODEL
$   MATERIAL PROPERTIES FOR MODIFIED CAM-CLAY MODEL:
    KAPPa 0.05
    LAMBda 0.14
```

VOID-ratio VOI

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **VOID-ratio**

Function - Defines the current void ratio of the soil

Input Variables -

VOI (Real) - REQUIRED

Additional Information -

Depends upon the current state of stress and must be input as a GMR-based quantity. Note that only one average value must be input for each GMR.

Examples of Use -

1. Specify the value of the current void ratio of the soil sample.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    MODIfied CAM-CLAY
$   MATERIAL PROPERTIES FOR MODIFIED CAM-CLAY MODEL:
    KAPPa 0.05
    LAMBda 0.14
    VOID-ratio 1.0096
```

PHI-soil PHI

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **PHI-soil**

Function - Defines the angle of internal friction of the soil

Input Variables -

PHI (Real) - REQUIRED

Additional Information -

Note that only one PHI-soil value is allowed for each GMR.

Examples of Use -

1. Specify the value of the angle of internal friction of the soil sample.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    MODIfied CAM-CLAY
$   MATERIAL PROPERTIES FOR MODIFIED CAM-CLAY MODEL:
    KAPPa 0.05
    LAMBda 0.14
    VOID-ratio 1.0096
    PHI-soil 29.5
```

CURRent-stress S11 S22 S33 S12 S13 S23 (for 3-D) or, S11 S22 S12 S33 (for 2-D)

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **`CURRent-stress`**

Function - Defines the components of the existing state of stress in the soil.

Input Variables -

S11, S22, S12, S33 (Real) - REQUIRED (for Axisymmetric and Plane strain problems; for axisymmetric analysis 1, 2, and 3 stand for r, z and θ-direction respectively.)

S11, S22, S33, S12, S13, S23 (Real) - REQUIRED (for 3-D problems)

Additional Information -

This input defines the average state of stress inside the GMR (region) with the material definition. If a different state of stress is required to be defined, then a second material (and hence GMR) must be defined with this different stress state. Usually the stress components σ_{12}, σ_{13} and σ_{23} are zero.

Note that only one set of values is allowed for each GMR.

Examples of Use -

1. Defines the values of the different components of the state of stress that exists in the soil sample before the analysis. (axisymmetric case)

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    ISOIl
$   MATERIAL PROPERTIES FOR ISOIL MODEL:
    KAPPa 0.05
    LAMBda 0.14
    VOID-ratio 1.0096
    PHI-soil 29.5
    COHEsion 5.0
    CURRent-stress   -27.0   -27.0   0.0   -27.0
$   NOTE: THIS IS AN AXISYMMETRIC ANALYSIS
```

PAST-maximum-stress S11 S22 S33 S12 S13 S23 (for 3-D) or, S11 S22 S12 S33 (for 2-D)

Status - REQUIRED (if MODIfied or ISOIl is input)

Full Keyword - **`PAST-maximum-stress`**

Function - Defines the components of the past maximum consolidation stress of the GMR.

Input Variables -

S11, S22, S12, S33 (Real) - REQUIRED (for Axisymmetric and Plane strain problems; for axisymmetric analysis, 1, 2, and 3 stand for r, z, and θ-direction respectively)

S11, S22, S33, S12, S13, S23 (Real) - REQUIRED (for 3-D problems)

Additional Information -

As in the case with CURRent-stress, this input is also constant within a particular GMR for which the material is defined. Every point inside a GMR has the same past maximum stress state and a new material must be defined for a different stress state. Usually the stress components σ_{12}, σ_{13}, σ_{23} are zero.

Examples of Use -

1. Defines the values of the different components of the past maximum consolidation stress state for the soil sample under analysis.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    ISOIl
$   MATERIAL PROPERTIES FOR ISOIL MODEL:
    KAPPa 0.05
    LAMBda 0.14
    VOID-ratio 1.0096
    PHI-soil 29.5
    COHEsion 5.0
    CURRent-stress   -27.0   -27.0   0.0   -27.0
    PAST-maximum-stress -27.0   -27.0   0.0   -27.0
$   NOTE: THIS IS AN AXISYMMETRIC ANALYSIS
```

COHEsion COH

Status - OPTIONAL (Only used if ISOIl is input)

Full Keyword - **COHEsion**

Function - Defines the cohesion of the soil.

Input Variables -

COH (Real) - OPTIONAL (Default is 0.0)

Additional Information -

The cohesion is usually a small positive value.

This parameter does not apply to MODIfied (Modified Cam-Clay Model).

Examples of Use -

1. Specify the value of the cohesion of the soil sample.

```
**MATERIAL
    ID-Material MAT1
    TEMPeratures 70.0
    EMODulus 1446.912
    POISson 0.3
  INELastic
    ISOIl
$   MATERIAL PROPERTIES FOR ISOIL MODEL:
    KAPPa 0.05
    LAMBda 0.14
    VOID-ratio 1.0096
    PHI-soil 29.5
    COHEsion 5.0
```

5.3 DEFINITION OF GEOMETRY

SECTION	KEYWORD	PURPOSE	PAGE
5.3.1	Geometry Input Card		5.3.15
	**GMR	start of geometric modeling region input	5.3.15
5.3.2	Region Identification		5.3.17
	ID-Gmr	region ID	5.3.17
	AXISymmetry-gmr	axisymmetric region	5.3.18
	PLANe-gmr	plane (2-D) region	5.3.19
	PLANe-gmr STRE	plane stress region	5.3.19
	PLANe-gmr STRA	plane strain region	5.3.19
	THICkness	thickness of a 2D plane or plate/ shell region	5.3.20
	MATErial	material property(set) for region	5.3.24
	RESTart-gmr	restart option for region	5.3.25
	SAVE-gmr	save reduced system matrix and right hand side for future analyses	5.3.27
	TINTegration	temperature used to determine material properties for integration	5.3.28
	TREFerence	reference (initial) temperature of region	5.3.29
	EXTErior	region is an infinite body	5.3.30
	HALF	region is a half-space	5.3.32
	CONTour-gmr	GMR control of contour plot information	5.3.34
5.3.3	Nodal Point Definition		5.3.36
	POINts	nodal points for boundary and volume discretization	5.3.36
	(NONE)	(coordinates)	5.3.37
5.3.4	Surface Element Definition		5.3.39
	SURFace	beginning of surface discretization	5.3.39
	TYPE-surface LINE	linear surface variation of field quantities	5.3.41
	TYPE-surface BUBBLE	5-noded surface variation of field quantities	5.3.41
	TYPE-surface QUAD	quadratic surface variation of field quantities	5.3.41
	TYPE-surface QUAR	quartic surface variation of field quantities	5.3.41

SECTION	KEYWORD	PURPOSE	PAGE
	GEOMetry LINE	linear geometry specified	5.3.46
	GEOMetry QUAD	quadratic geometry specified	5.3.46
	ELEMents-surface	element list	5.3.48
	(NONE)	(element connectivity)	5.3.49
	TRANslate	surface generation by translation	5.3.58
	REFErenece	axis of rotation for surface generation	5.3.59
	DIREction	direction of axis of rotation	5.3.60
	ROTAtion	angle of rotation for surface generation	5.3.61
	CRACk	defines the nodes at the crack front	5.3.62
5.3.5	Normal Definition		5.3.63
	NORMal	defines outer normal of surface	5.3.63
	NORMal SIGN	defines sign of normal	5.3.67
5.3.6	Enclosing Element Definition		5.3.69
	ENCLosing-elements	enclosing element list	5.3.69
	(NONE)	(enclosing element connectivity)	5.3.70
5.3.7	Discontinuity Element Definition		5.3.73
	DISCcontinuity-elements	discontinuity element list	5.3.73
	(NONE)	(discontinuity element connectivity)	5.3.76
5.3.8	Line Element Definition		5.3.77
	LINE-element-definition	start of line definition	5.3.77
	POINts-line	nodal points for line discretization	5.3.78
	(NONE)	(coordinates)	5.3.79
	TYPE-line LINE	linear variation of field quantities for 3-noded lines	5.3.80
	TYPE-line QUAD	quadratic variation of field quantities for 3-noded lines	5.3.80
	HOLE	start of hole element connectivity	5.3.81
	RUNNer	start of runner element connectivity	5.3.83
	CLINe	start of cooling line element connectivity	5.3.85
	TREFerence-line	reference temperature of runner/ cooling line	5.3.87
	CONVection-line	surface convection coefficient of runner/cooling line	5.3.89

SECTION	KEYWORD	PURPOSE	PAGE
	RESIstance-line	surface resistance coefficient of runner/cooling line	5.3.89
	ELEMents-line	line element list	5.3.90
	(NONE)	(element connectivity, radius, etc.)	5.3.91
5.3.9	Sampling Points and Sampling Surface Definition		5.3.95
	SAMPling-points POINTS	start of definition of sampling	5.3.95
	(NONE)	points (coordinates)	5.3.96
	SAMPling-surfaces SURF	definition of sampling surface	5.3.97
	ELEMents-sampling-surf	sampling surface element list	5.3.99
	(NONE)	(sampling surface element connectivity)	5.3.100
5.3.10	History Point Definition		5.3.101
	HISTory-points	list of points for which history of specified results is recorded at every time step or frequency	5.3.101
5.3.11	Volume Cell Definition (Plasticity)		5.3.102
	VOLUme	Initiates definition of interior volume	5.3.102
	TYPE-volume LINE	linear variation of initial stress through cell	5.3.103
	TYPE-volume QUAD	quadratic variation of initial stress through cell	5.3.103
	CELLs	cell list	5.3.104
	(NONE)	cell connectivity	5.3.105
	FULL	indicates GMR is completely filled with volume cells	5.3.109
5.3.12	Property Definition for FE GMR		5.3.110
	PROPerty	defining property attributes of materials to be used in the finite element definition	5.3.110
	MASS	identify the mass type	5.3.117
	OVDAmping	define the overall damping value of the structural system	5.3.118
	ECCEntricity	nodal eccentricity for plate/shell stiffeners used in the FE structural definition of a coupled structural-acoustics problem	5.3.119

Modeling Geometry and GMRs

The geometry of a problem for **GPBEST** analysis is typically broken down into several geometric modeling regions (GMRs). The GMRs are further broken down into surfaces, elements and nodes.

The simplest **GPBEST** input data file would contain one GMR which would consist of one surface containing several boundary elements and nodes. Every **GPBEST** input data file must contain at least one GMR. However, if a problem consists of more than one GMR, interfaces must then be specified between the GMRs to define how they are connected. The subject of interface connections is treated in Section 5.4 of this manual. A problem with a single GMR would not need any interfaces.

There are several reasons for breaking up a problem into separate GMRs:

1) Modeling considerations
2) Efficiency considerations
3) Problems with multiple materials
4) 2-D problems with regions of different thickness
5) Interface conditions
6) Finite Element regions

1. Modeling Considerations

As problems become quite large and complex, it is desirable to break up the large problem into various GMR regions where each GMR represents a logical section of the component. Even if the entire model could be analyzed with one GMR, it may be more desirable to model individual sections at a time.

This is similar to the concept of ‘superelements’, or substructuring, in finite element analysis. Superelements permit a team of engineers to work on different parts of the model at the same time and then piece them together for a global analysis when all sections are completed. GMRs perform a similar role in **GPBEST** .

Figure 5.3.1 illustrates a problem which has been broken up into four GMRs. A benefit of doing this is that the highly stressed pin GMR can be modified independently of the rest of the model, either through geometry change or material change.

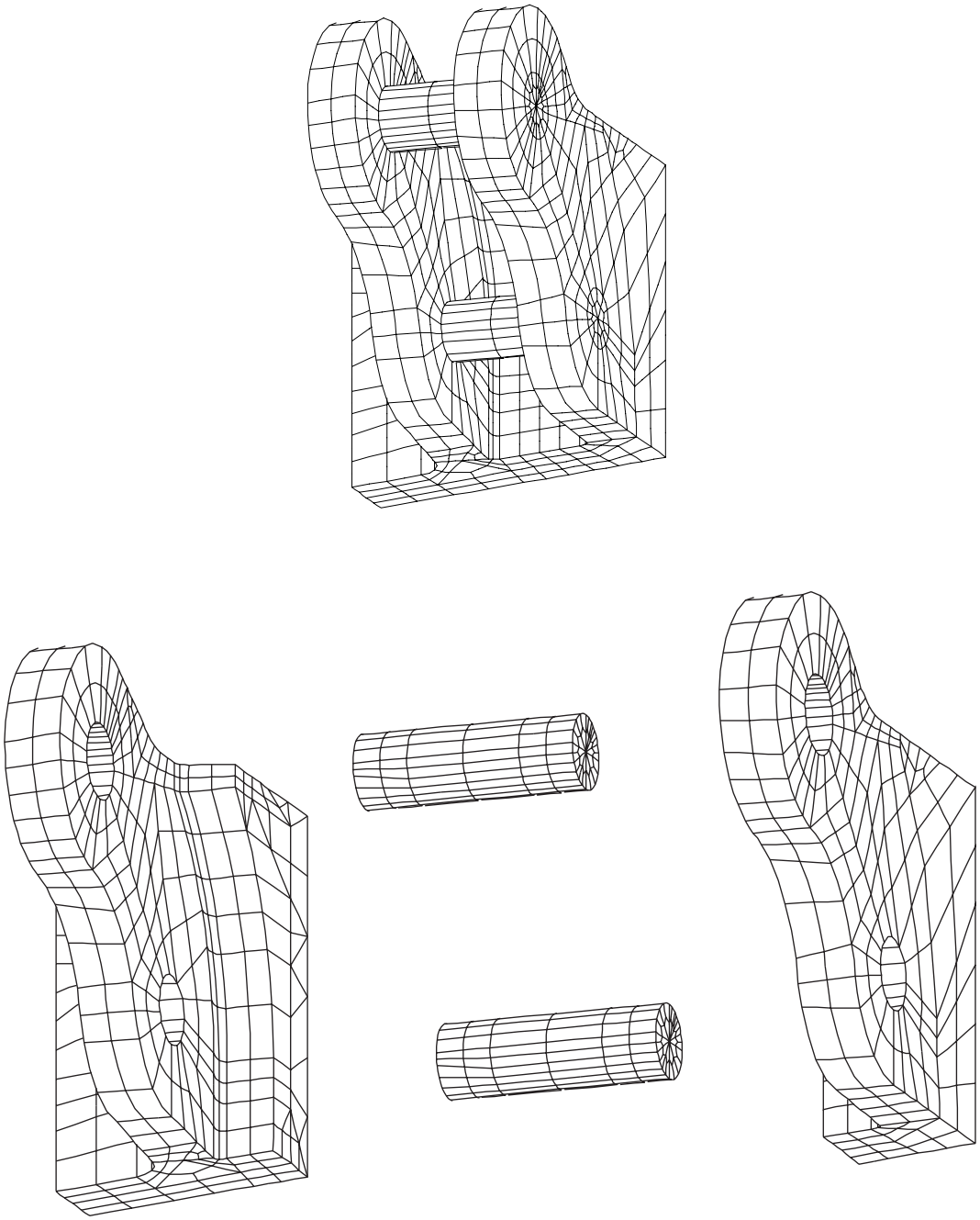


Figure 5.3.1 Four GMR, 3-Dimensional Model

Definition of Geometry

Figure 5.3.2 shows a 2-D gap-and-contact analysis model consisting of six GMRs.

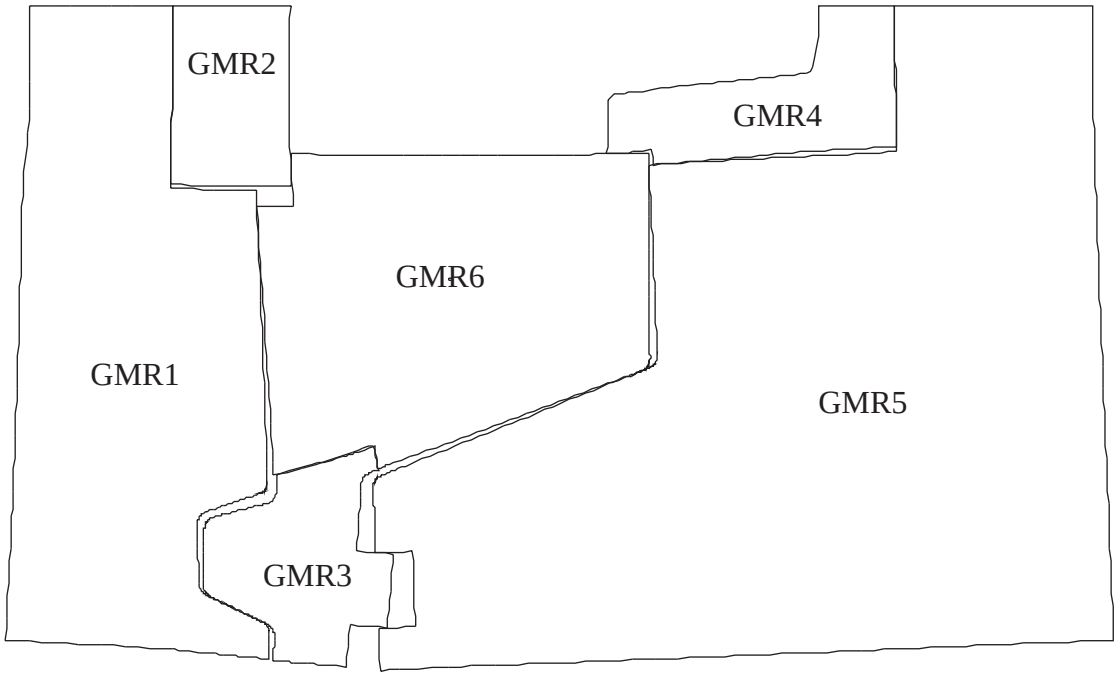


Figure 5.3.2 Six-GMR Model Using Gaps and Contact

2. **Efficiency Considerations**

Another reason for using GMRs involves the practical matter of efficiency. A large problem which is broken down into several GMRs runs more efficiently than the one-GMR model. The overall analysis time and disk space requirements are reduced by using multiple GMRs. In particular, the **GPBEST** integration step is considerably faster through the use of multiple GMRs.

Of even greater impact is the fact that **GPBEST-DCE** (Distributed Computing Environment) can send individual GMRs to be processed on networked computer workstations. In this way, each GMR can be processed simultaneously, thus reducing the overall analysis time. Sections 5.8 and 8 discuss **GPBEST-DCE** in greater detail.

3. **Problems With Multiple Materials**

If a problem consists of multiple materials, then each material would logically go in its own GMR. For linear analysis, each GMR must contain only one material. For instance, if an aluminum bracket contained a steel reinforcement, the aluminum portion and the steel portion would have separate GMRs.

4. **2-D Problems With Regions of Different Thickness**

Two-dimensional problems, such as shown above in Figure 5.3.2 can have regions of different thickness (in the direction perpendicular to the page). Each GMR must have only one thickness, so multiple thicknesses must be handled with multiple GMRs.

Another way of using GMRs is in MIXEd region problems. This is activated by specifying the MIXEd card in **CASE. In MIXEd problems, the dimensionality of the problem is declared region by region, or GMR by GMR. For instance, one GMR could be an axisymmetric region, another one could be a plane stress region, while yet another GMR could be a plane strain region. Figure 5.3.3 illustrates an axisymmetric model touching a plane stress region. In this case, there are actually ten plane stress regions distributed around the axisymmetric region, similar to gear teeth. Force compatability is enforced across the interface in MIXEd problems.

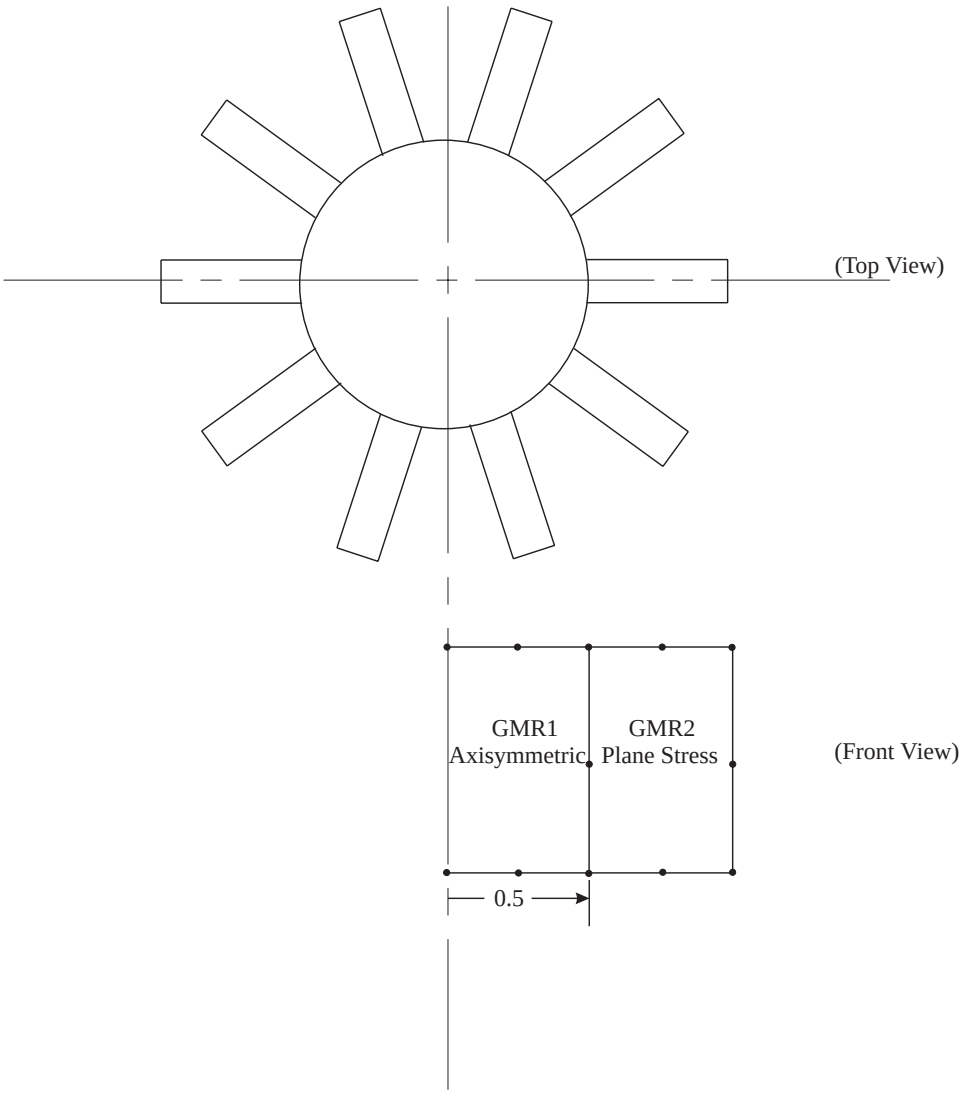


Figure 5.3.3 Example of MIXEd Problem

Note that the MIXEd feature is only available for two-dimensional problems and is not available for three-dimensional problems or generalized axisymmetric problems. This highlights an important way multiple GMRs can be combined to solve complex problems.

5. Interface Conditions

Definition of Geometry

Sometimes none of the above conditions necessitate the use of multiple GMRs. However, at times it is preferable to specify a special condition on the edge of a region, or between two or more regions. This involves the use of interface conditions. A wide variety of interface conditions can be specified in **GPBEST** : bonded and sliding contact, interference fits, spring or convection conditions and cyclic symmetry contact.

Again, Figure 5.3.2 shown above, illustrates the use of special gap and contact interfaces which allows regions to press into each other and separate from each other, depending on the particular load and geometry.

Interfaces are covered in detail in Section 5.4 of this manual.

Other GMR Considerations

Each GMR contains several important pieces of data. The corresponding **GPBEST** input cards are shown in parentheses:

1) A unique GMR name (e.g., ID-Gmr GMR1)
2) A material name, referring to a ID-Material in **MATERIAL input (e.g., MATErial STEEL)
3) Point (or node) locations (POINts-line or POINts)
4) Surface information, containing ELEMents-surface, etc. (e.g., SURFace SURF1)
5) Internal or external sampling points (SAMPling-points)

Points

Points are merely nodal locations in space. They are used to define where boundary elements are located, as well as specifying nodal boundary conditions and locations of concentrated forces.

Surfaces

Each GMR is made up of one or more surfaces. The term ‘surface’ in **GPBEST** is not the same as ‘surface’ used in CAD programs. CAD programs commonly use the term ‘surface’ (or ‘patch’) to refer to a continuous curve in space without any kinks. Often, these are used to generate a mesh. In **GPBEST** , the term ‘surface’ refers to an entire boundary.

Figure 5.3.4a shows a 2-D single GMR with one surface defined. Although the outer boundary constitutes one surface in **GPBEST** , it is made up of several straight-line segments with sharp corners connecting them. The GMR in Figure 5.3.4b has two surfaces: one for the outer boundary, and one for the internal hole. If there were more internal holes, each would require its own surface definition.

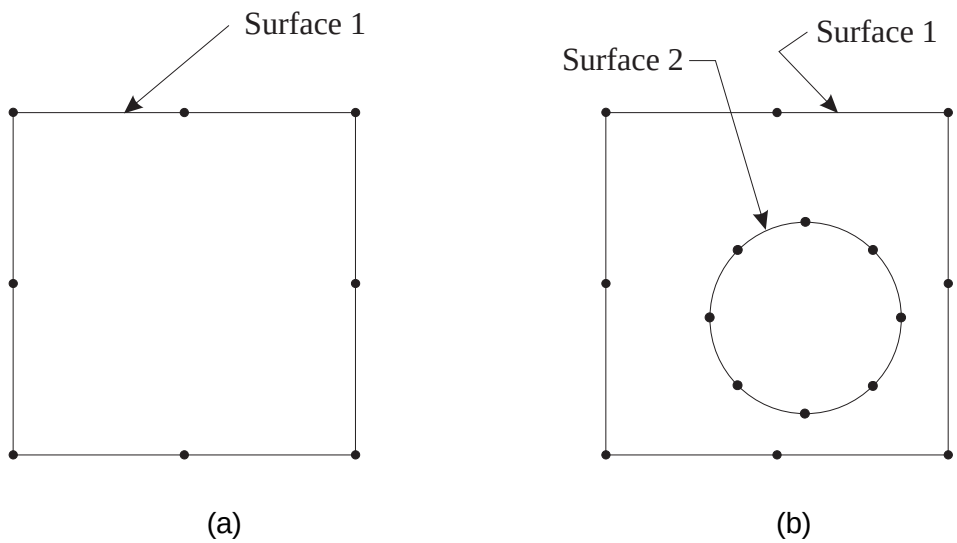


Figure 5.3.4 (a) One Surface, (b) Two Surfaces

A surface is composed of boundary elements. The elements are located in space through the connectivity with the points, which were described above. For 2-D and axisymmetric problems, the boundary elements are straight or curved lines. For 3-D problems, the boundary elements are six-, eight-, or nine-noded parametric elements. Also specified in the SURFace definition is the variation of field quantities over the elements through the TYPE-surface card.

Sampling Points

Sampling points are associated with a GMR, but are not associated with any elements in the GMR. They are merely points in space used to determine results. Since the primary results determined by **GPBEST** are on the boundary, sampling points provide the user with a way to determine results inside or outside a body. A number of sampling points can be dispersed inside a GMR to determine the internal behavior. Sampling points are also convenient in that they allow the user to retrieve results at an exact location in space, for instance at the location (x,y,z)=(4.3,6.7,3.0). Since they are not attached to any elements, their placement in space is quite simple.

For instance, in the 2-D model of Figure 5.3.5, there is a sharp stress concentration in the vicinity of the notch. It may be desirable to plot the stress at some internal points to determine the rate of stress near the notch. This can be done by specifying a few sampling points near the notch, as shown.

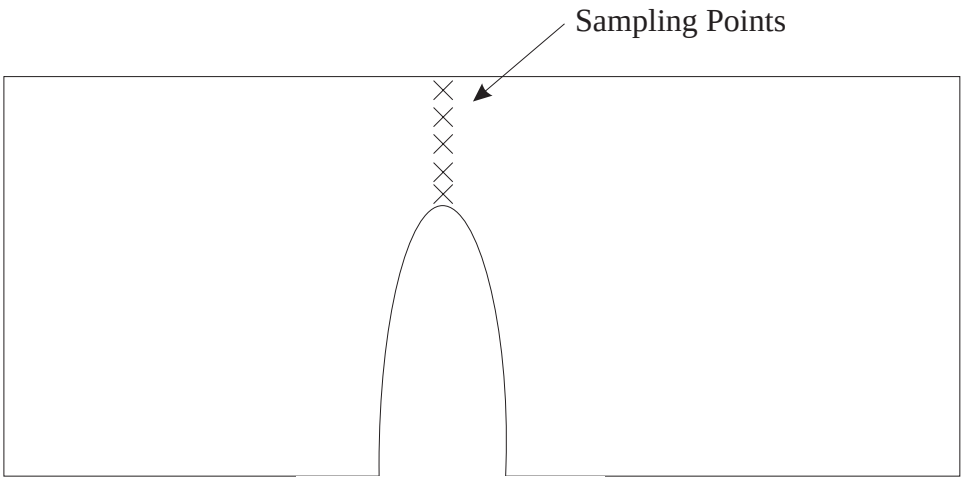


Figure 5.3.5 Sampling Points Placed Near a Stress Concentration

Sampling points do not increase the cpu time to obtain the boundary solution. They merely add to the post-processing step, when boundary values are used to calculate results at the internal (sampling) points.

It should be noted that if the user is using BESTVIEW, PATRAN, or I-DEAS for post-processing, internal results for 2-D problems can be visualized without having to specify a large number of sampling points. This is due to the fact that **GPBEST** uses the CONTour-plot card in **CASE to automatically generate interior results for post-processing. Thus, the user is saved the painful step of having to specify a large number of interior sampling points just to see results in the interior.

Another useful feature is the sampling surface. Figure 5.3.6 illustrates a sampling surface consisting of four elements inside a cube. The sampling surface shown here is used to post-process results in the interior along the diagonal plane. Another use of the sampling surface is in acoustics analysis, where you want to monitor results on the exterior of a body. Several sampling surfaces can be used to monitor results in various locations.

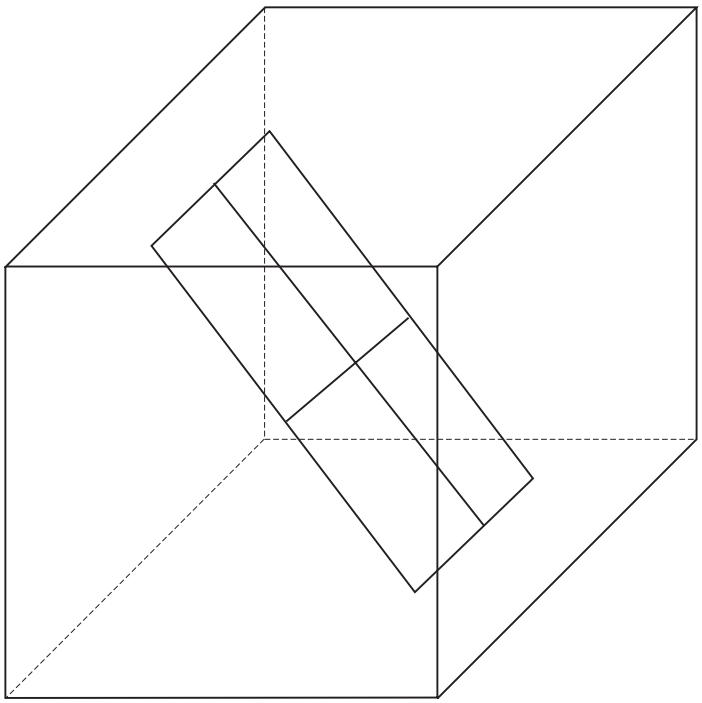


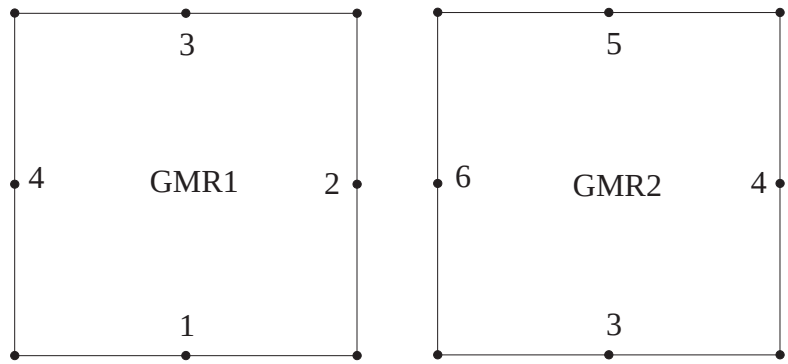
Figure 5.3.6 Sampling Surface Used to Monitor Internal Results

Other Geometry Considerations

There are limits on the current version of **GPBEST** regarding the number of GMRs and the number of elements in a single GMR, etc. These limits are discussed in Section 5.0. Also, the PRINtout-control LIMITS feature in **CASE can be used to have **GPBEST** print the current limits to the output file (see Section 5.1).

Node and Element Numbering

An important point should be made regarding node and element numbering. Each GMR must have unique node and element numbers within each GMR. However, **GPBEST** allows identical node and element numbers in different GMRs. Since each GMR is processed independently, each GMR is unaware of nodes and elements in other GMRs. Figure 5.3.7 illustrates two GMRs having non-unique element numbers.



(Element Numbers Shown)

Figure 5.3.7 Two GMRs with Non-Unique Element Numbers

Although **GPBEST** allows non-unique node and element numbering, most commercial CAD and Finite Element modeling programs do not. If programs such as PATRAN or I-DEAS are being used to *model* problems, these will not allow the user to specify non-unique node and element numbers.

BESTVIEW on the other hand, being tightly integrated with **GPBEST** , globally allows non-unique entity numbering. It should also be mentioned that PREBEST and POSTBEST permit the use of non-unique entity numbers as well.

The only situation that could be troublesome is if a **GPBEST** data file is obtained from another source or is edited manually. In particular, some of the example problems provided with **GPBEST** distribution contain multiple GMRs with non-unique node and element numbers. **GPBEST** will run correctly with these, and BESTVIEW will display these models and results without difficulty. However, if the model is read into a commercial package such as PATRAN or I-DEAS, the program will detect the non-unique numbers and renumber the offending entities. In this case, the one-to-one correspondence with the original data file numbering is lost and results cannot be displayed. It is recommended that **GPBEST** users develop the habit of using globally-unique node and element numbers.

6. Finite Element Regions

In order to solve a structural-acoustics problem, a special GMR must be declared with an ID associated with the qualifier FINI to define a finite element (FE) region describing the structure. Each FE region (or GMR) should be associated with the following sections:

a) MATErial card which should always have a qualifier ‘‘STRUCTURE.’’
b) MASS card with a relevant qualifier.
c) THICkness card describing a constant thickness of a plate/shell type structure at every node.
d) POINts_surface card followed by the point ID, its spatial location and an optional thickness value (if it is different from the value prescribed with the THICkness card).

e) ECCEntricity card, in the case of frames (stiffeners) placed at an eccentricity with respect to the plate/shell mid-plane, followed by a list of nodes and eccentricity values.

f) PROPerty card followed by property attributes and respective values.

g) ELEMents-surface card followed by the element ID and the nodal connectivity list. The elements would have the same attributes as those declared by the last PROPerty card coming before the ELEMents-surface list, i.e., multiple element properties may be defined by PROPerty section, ELEMents section, PROPerty section, ELEMents-surface section and so on.

Depending on the formulation being used, the following options are available for a one-way coupled structural-acoustic analysis:

a) DIRECT BEM formulation:

The structure: One FE GMR is to be defined describing the structure being solved under the imposed mechanical loading. The available element types include the 6-noded and 8-noded shells and the 3-noded frames. The frame element, if acting as a stiffener, could have an eccentricity value associated.

The fluid: One or more BEM GMRs may define the acoustic domain. Note that each such GMR must be a closed region, as usual. For the fluid domain the entire **GPBEST** element family is available.

b) INDIRECT BEM formulation:

The structure: One FE GMR as described in Section (a) above (DIRECT BEM formulation).

The fluid: Only one BEM GMR should be used to define the acoustic domain. In contrast to the ‘‘closed’’ GMR of a DIRECT BEM, the fluid GMR here could be ‘‘open.’’ Only 6 and 8-noded elements are permitted

Both the structural GMR and the acoustic GMR may have the same mesh density or unequal mesh density. This allows the FE structural GMR to contain a finer mesh compared to the BEM acoustic GMR.

Figure 5.3.8 illustrates a structural acoustics problem. It is a cantilevered shell, where the curved shell and stiffeners are modeled with finite elements, whereas the surrounding fluid region is modeled with boundary elements.

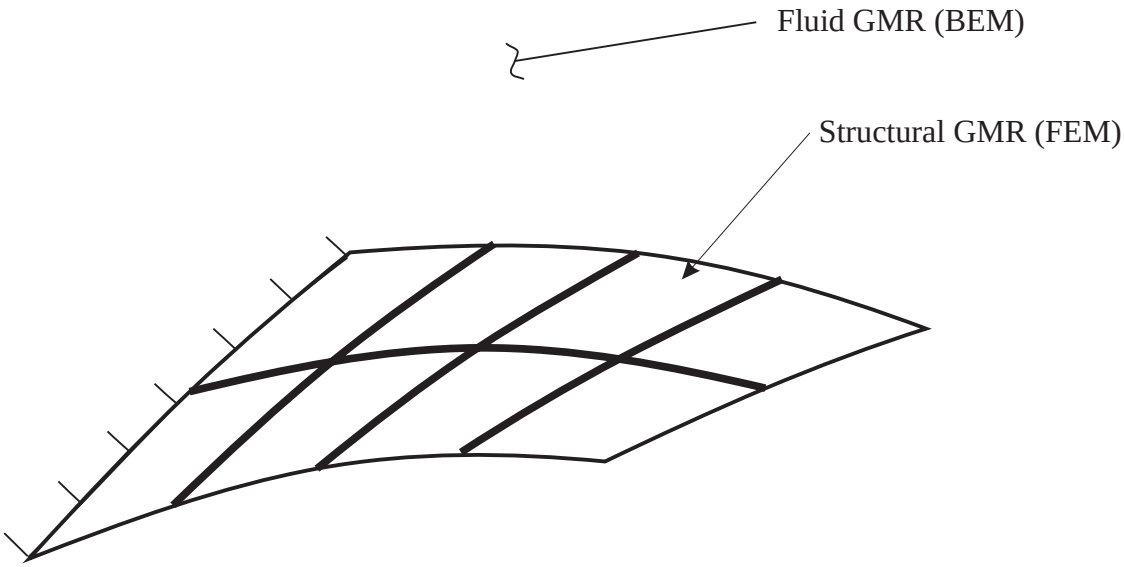


Figure 5.3.8 Direct BEM Formulation for Structural Acoustics Problem

For a fully coupled structural acoustic system, the following options are available:

a) DIRECT BEM formulation:

The structure: One FE GMR with imposed boundary constraints and loading. The available element types include the 6-noded and 8-noded shells and the 3-noded frames. The frame element, if acting as a stiffener, could have an eccentricity value associated.

The fluid: One or more BEM GMRs may define the acoustic domain. Note that each such GMR must be a closed region, in the absense of any global symmetry.

b) INDIRECT BEM formulation:

The structure: One FE GMR as described in Section (a) above (DIRECT BEM formulation).

The fluid: Only one BEM GMR should be used to define the acoustic domain. The mid-surface of the FE shell GMR could be used to define the acoustic GMR.

The important distinction here (with respect to the one-way coupled analysis) is that the structural GMR and the acoustic GMR must have the same mesh density. There should be at least one **INTERFACE section in the **GPBEST** data file of type ‘‘SANDwich.’’ All structural nodes connected to shells are considered ‘‘wet,’’ i.e., every shell element must be present in any of the SANDwich interface sections.

5.3.1 GEOMETRY INPUT CARD

**GMR

Status - REQUIRED

Full Keyword - **GMR**

Function - This card signals the beginning of the definition of a geometric modeling 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 two-dimensional or three-dimensional space.** However, under the following two circumstances, the region may be open :

1 - In axisymmetric and 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. Likewise, when the axis of symmetry passes through the interior of an axisymmetric body, no elements should lie along the axis of symmetry.

2 - In GMRs with boundaries extending to infinity, a GMR may have open boundaries. However, this must be indicated through the use of the EXTErior or HALF cards or by enclosing the open boundary with Enclosing elements (see the ENCLosing-elements card). Note : One of the above three devices MUST be used in an infinite region.

A GMR may have multiple internal boundaries in addition to a single external boundary.

A finite element GMR (defining a plate/shell/frame type structure) or a boundary element GMR (used to define the acoustic region) for an Indirect boundary element formulation need not be a closed region.

Examples of Use -

```
**GMR
  ID-Gmr  REG1
  MATErial  STEEL
  TREFerence  70.0
  TINTegration  70.0
  POINts_surface
    1  10.0  0.0  2.0
    2  10.0  1.0  2.0
            .
            .
```

5.3.2 REGION IDENTIFICATION

ID-Gmr NAME [FINITE]

Status - REQUIRED

Full Keyword - **ID-Gmr**

Function - This card provides the identifier for the GMR.

Input Variables -

NAME (Alphanumeric) - REQUIRED

The alpha qualifier FINITE is necessary when a finite element GMR is to be defined for a structural-acoustics problem. The absence of this qualifier would imply that this is a Boundary Element (BE) GMR.

Additional Information -

The NAME must be eight or less alphanumeric characters. Blank characters occurring 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 **GPBEST** output file.

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

Examples of Use -

1.
```
**GMR
  ID-Gmr REG1
  MATErial STEEL
```

2.
```
**GMR
  ID-Gmr REG2 FINITE
  MATErial STRUCTURE
```

AXISymmetry-gmr

Status - OPTIONAL

Full Keyword - **`AXISymmetry-gmr`**

Function - Identifies an axisymmetry region in a MIXEd problem (in **CASE). The default is the problem type as specified in **CASE input.

Input Variables - NONE

Additional Information -

This card overrides the dimensionality specified in case input for this region only.

Region mixtures are only allowed between axisymmetric, plane stress, and plane strain regions. Generalized Axisymmetric regions are also not allowed in MIXEd problems. Therefore, this card must not appear in GMR input for AXISymmetry-gmr GENERAL case input.

In an axisymmetric region, the axis of symmetry should not be modeled, and therefore, boundary elements should not lie along on the axis of symmetry.

The use of an axisymmetric region automatically invokes the condition of zero radial displacement at the origin, therefore, displacement in the radial direction does not have to be set to zero at the origin or at any other point for the purpose of preventing (arbitrary) ‘rigid body’ motion (in the radial direction). The displacement in the axial direction, however, must still be fixed at some point in order to prevent rigid body translation in this direction.

Examples of Use -

1. Request an axisymmetric region for GMR REG1.

```
**GMR
  ID-Gmr REG1
  MATErial MAT1
  TREFerence  0.0
  AXISymmetry-gmr
```

PLANe-gmr ETYPE

Status - OPTIONAL

Full Keyword - **PLANe-gmr**

Function - Identifies a two-dimensional region in a MIXEd case input. Default is the problem type as specified in case input.

Input Variables -

ETYPE (Alphanumeric) - OPTIONAL

Allowable values are STRE or STRA.

STRESS - specifies a plane stress region.

STRAIN - specifies a plane strain region.

Additional Information -

This card overrides the dimensionality specified in case input for this region only for MIXEd dimensioned problems in case input.

Region mixtures can only occur between axisymmetric, plane stress and plane strain regions. Generalized Axisymmetric regions are not allowed in MIXEd problems. Therefore, this card must not appear in GMR input for AXISymmetry-gmr GENERAL case input.

The thickness of this region is assumed to be 1.0 unless a value is specified on the THICkness card.

For heat conduction (or potential) problems, only the first keyword "PLAN" needs to be specified.

If ETYPE is not specified, STRAIN is assumed.

Examples of Use -

1. Request a plane strain region for GMR REG1.

```
**GMR
  ID-Gmr REG1
  MATErial MAT1
  TREFerence 0.0
  PLANe-gmr STRAIN
```

2. GMR HOT is used for 2-D heat transfer.

```
**GMR
  ID-Gmr HOT
  MATErial 4
  TREFerence 0.0
  PLANe-gmr
```

THICkness VAL IVAL

Status - OPTIONAL

Full Keyword - **`THICkness`**

Function - Defines the the thickness of a two-dimensional plane or a plate/shell FE region in a structural-acoustic problem.

Input Variables -

VAL (Real) - REQUIRED (Default is 1.0)

VAL is the thickness of the plane region (in the Z direction)

IVAL (Integer) - OPTIONAL (Default is 1)

IVAL is the number of circumferential repetitions of a planar region about an axisymmetric region in an analysis with mixed plane/axisymmetric regions. (IVAL multiplies the thickness of a planar region)

Additional Information -

The thickness must be greater than zero.

Force compatibility is maintained across interfaces in MIXEd region problems.

Two regions sharing a common interface may have different thicknesses.

For two regions of different thicknesses sharing a common (bonded) interface, the tractions integrated in the z directions must be set equal and opposite to the integrated tractions on the other region. i.e.

$$a^I t_i^I = -a^{II} t_i^{II}$$

where a^I and a^{II} are the thicknesses of region I and region II, respectively.

The THICkness card should only be used in a 2-D planar region or an FE GMR while solving a structural-acoustics problem.

A uniform thickness of a plate/shell structure (FE region) may be defined for solving structural-acoustics problems. Nodal thickness may be defined later, which would override this value at specific nodes.

If a planar region is connected to an axisymmetric region, the THICkness card is used only in the planar region, not in the axisymmetric region. In the axisymmetric region, the thickness is automatically set to 2π times the radius of the axisymmetric region at the interface node.

If IVAL is specified, then the thickness ‘a’ is defined as a = VAL * IVAL

IVAL must be an integer value greater than zero.

Examples of Use -

1. Define an axisymmetric region next to a plane stress region. (See the figure.) There are ten plane stress regions around the circumference of the axisymmetric region (similar to having ten gear teeth.) The thickness of each plane stress region is .314. In terms of the above terminology,

$$a^I = 2\pi r = 3.14159$$

$$a^{II} = .314159 \times 10 = 3.14159$$

Therefore, in this case the tractions at the interface will be equal and opposite.

```
**CASE
  TITLe - ELAS044 - MIXED AXISYMMETRIC AND PLANE STRESS PROBLEM
  TIMEs-output  1.0
  MIXEd                          $ MIXED REGIONS
  . . .

**GMR
  ID-Gmr GMR1
  MATErial MAT1
  AXISymmetry-gmr                        $ (AXISYMMETRIC REGION)
  TINTegration 70.0
  POINts
    101    0.0  0.0
    . . .

**GMR
  ID-Gmr GMR2
  MATErial MAT1
  TINTegration 70.0
  PLANe-gmr STRESS
  THICkness  .314159265359  10   $ (10 PLANE STRESS REGIONS;
  POINts                  $  THICKNESS OF EACH IS .314)
    201   0.5   0.0
    . . .
```

Definition of Geometry

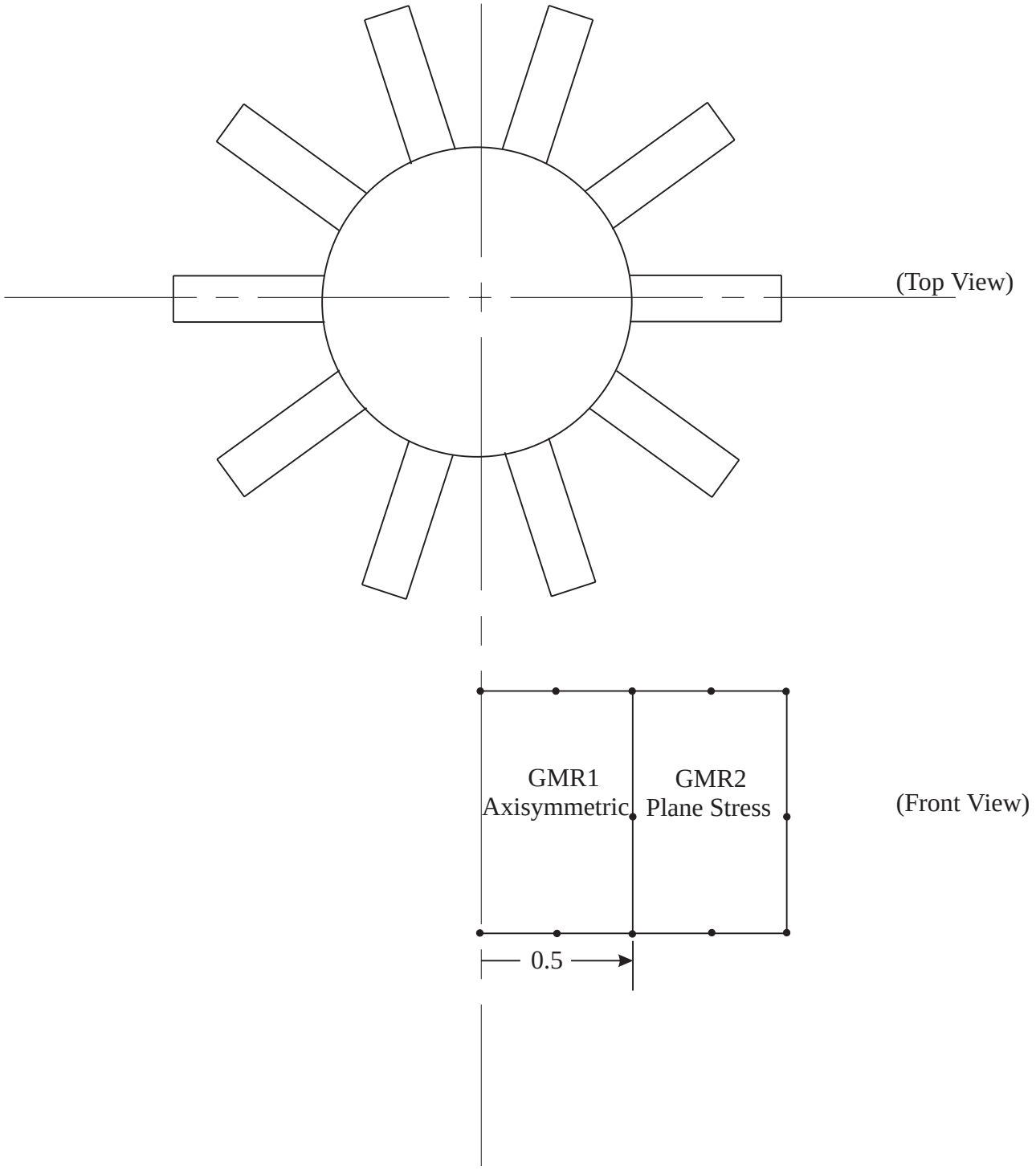


Figure for THICkness Card: Mixed-Region Problem

2. Define a structural GMR for analyzing a structural-acoustics problem.

```
**CASE
  TITLe  CANTILEVER PLATE: STRUCTURAL ACOUSTICS PROBLEM
  SACOustic PERIODIC DIRECT ONEWAY
  FREQuency STEP   10.0   50.0   2.0
  LOAD COMPLEX
  ECHO-input
  NEUTral-file
  ..........
  ..........
**GMR
  ID-Gmr  g01  FINI
  MATErial STRUCTURE
  MASS CONSISTENT
  THICkness 0.267E-02
  POIN
  1   0.0   0.0   0.0
  ..........
  ..........
```

MATErial NAME

Status - REQUIRED

Full Keyword - **MATErial**

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-Material card in **MATERIAL input).

While solving a structural-acoustics problem, the GMR describing the structure should have ‘‘MATErial STRUCTURE’’ explicitly defined. The structure is solved through a Finite Element method using material property attributes defined through a ‘‘PROPerty’’ card.

Examples of Use -

```
**GMR
  ID-Gmr GMR1
  MATErial STEEL
```

RESTart-gmr RTYPE

Status - OPTIONAL

Full Keyword - **`RESTart-gmr`**

Function - Defines the restart level for the current GMR.

Input Variables -

RTYPE (Alphanumeric) - REQUIRED

Allowable values are WRIT, READ, HOLE and SOLV.

WRITE - Performs integration for this GMR, and saves the resulting coefficients for later reuse.

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

HOLE - Only surface integration coefficients for the current GMR are reused from the restart files. All hole related coefficients are recomputed and stored on the restart files.

SOLVER - Uses the reduced system matrix for the current GMR from a previous run. Integration, assembly and elimination are skipped for the current GMR.

Additional Information -

This option is only available for elastostatics and steady-state versions of heat conduction, concurrent thermoelasticity, thermal distortion, and consolidation. Transient analyses cannot employ GMR-based restarts.

A RESTart GMR card must appear in the **CASE section. See the description of the RESTart card in Section 5.1 for more information.

Geometry and material properties for the current GMR must be the same for both the RESTart-gmr WRITE and RESTart-gmr READ data sets. However, only minimal checking is done by **GPBEST**. This is the user's responsibility.

It is also the user's responsibility to be certain that the material properties and surface geometry for the current GMR remain unchanged for the RESTart-gmr HOLE option. Only hole element geometry may change.

RESTart-gmr SOLVER is only available for steady-state versions of heat conduction, thermal distortion, and concurrent thermoelasticity.

The RESTart-gmr SOLVER option can only be used if a SAVE-gmr SOLVER card has been included in the previous run, along with RESTart-gmr WRITE, READ or HOLE. The only items that may be changed for a RESTart-gmr SOLVER analysis are the values specified for VARIable boundary condition sets and ambient temperatures for RELAtion bcsets. Geometry, material properties, interface definitions, and VALUe bcsets must be identical to those used for the previous run.

Examples of Use -

1. The positioning of cooling lines in an injection mold has been changed since the previous restart analysis. Consequently, use the RESTart-gmr HOLE option to minimize processing time for this reanalysis.

```
**GMR
  ID-Gmr MOLD1
  MATErial STEEL
  RESTart-gmr HOLE
```

SAVE-gmr STYPE

Status - OPTIONAL

Full Keyword - **SAVE-gmr**

Function - Requests that certain matrices and problem information for the current GMR be saved on disk for subsequent analyses.

Input Variables -

STYPE (Alphanumeric) - REQUIRED

Allowable value is SOLV

SOLVER - Saves reduced system matrix and right hand side matrices for the current GMR for subsequent analyses. Material, geometry, and interface information is also retained.

Additional Information -

This option is only available for the steady-state versions of heat conduction, concurrent thermoelastic and consolidation analyses.

The SAVE-gmr SOLVER card must be used in conjunction with RESTart-gmr WRITE, READ or HOLE.

Once a SAVE-gmr SOLVER analysis is completed, the decomposed system matrix and right hand side matrices remain on disk. These can be accessed on the following run by including a RESTart-gmr SOLVER card. Only the values specified in VARIable boundary condition sets and ambient temperatures defined in RELAtion bcsets may differ between the two analyses. Geometry, material properties, interface definitions and VALUe bcsets must remain unchanged. Essentially, only a new right hand side vector may be formed.

Examples of Use -

1. Save the integration and solver files for GMR1 generated during the heat conduction analysis of a piston.

```
**GMR
  ID-Gmr  GMR1
  MATErial  STEEL
  RESTart-gmr READ
  SAVE-gmr SOLVER
```

TINTegration TEMP

Status - OPTIONAL

Full Keyword - **`TINTegration`**

Function - Defines the 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 **MATERIAL, the material properties used in the integration of the GMR will be calculated based on the temperature specified on this card using linear interpolation.

For problems in which the temperature changes in time and/or space, it is recommended that the reference temperature be chosen as the (time/volume weighted) average temperature over the GMR.

If this card is not input then the reference temperature is used (see TREFerence card).

Examples of Use -

1. Specify the integration temperature at which the material properties are evaluated.

```
**GMR
  ID-Gmr REG1
  MATErial MAT1
  TINTegration 70.0
```

2. GMR CENTER is initially at 600 deg F and being heated to 1000 deg F in a transient heat transfer problem, the average value of the temperature range is used to evaluate the material properties (800 deg F).

```
**GMR
  ID-Gmr CENTER
  MATErial PLASTIC
  TREFerence   600.
  TINTegration 800.
```

3. This problem is equivalent to example 2, except the initial body datum temperature is set to zero by default (TREFerence card is not used).

```
**GMR
  ID-Gmr CENTER
  MATErial PLASTIC
  TINTegration 800.
```

TREFerence TEMP

Status - OPTIONAL (used in temperature dependent problems)

Full Keyword - **TREFerence**

Function - This card defines the reference (or initial) temperature (i.e. the datum temperature of 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 reference temperature is assumed as zero.

Examples of Use -

1. Specify the initial temperature of the region REG1 to be 70 deg C with material properties evaluated at 100 deg C.

```
**GMR
  ID-Gmr REG1
  MATErial  MAT1
  TREFerence  70.0
  TINTegration 100.0
```

EXTErior

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.

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 or temperature at infinity in axisymmetric and three-dimensional media. A similar decay does not occur in the two-dimensional case and consequently, considerable caution must be exercised.

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 EXTErior 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).

This card is not valid for a FE GMR.

An alternative method to account for infinite boundaries is to model the infinite boundary with enclosing elements (see ENCLosing-elements 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 than using the EXTErior card.

Examples of Use -

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

```
**GMR
  ID-Gmr  GMR1
  MATErial MAT1
  TREFerence 70.0
  EXTErior
  POINts
        1      0.0      212.00
        2     41.36     207.93
        .       .          .
        .       .          .
        .       .          .
```

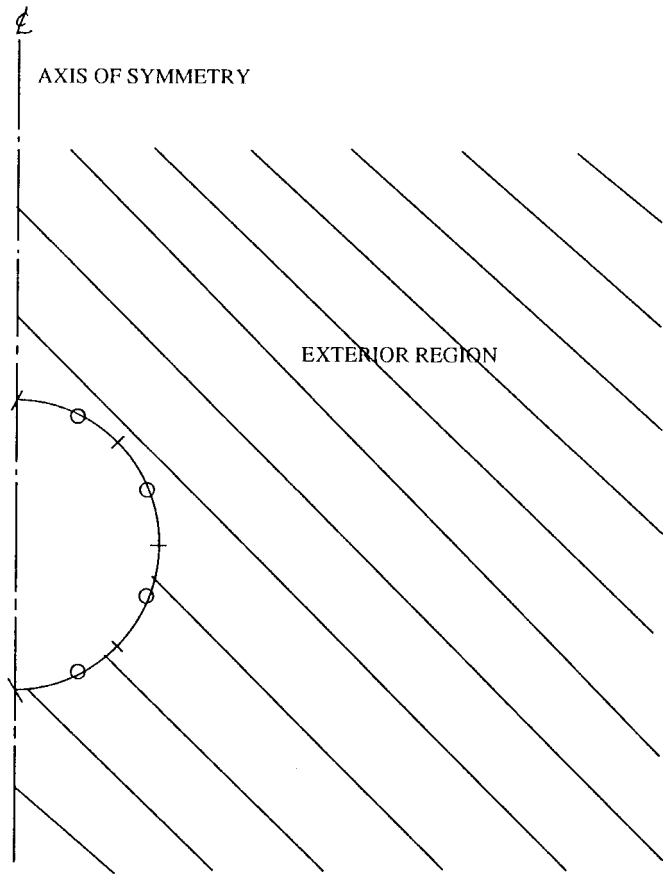


FIGURE FOR EXTE CARD

HALF

Status - OPTIONAL

Full Keyword - **HALF** (Not Available in Axisymmetry)

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 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 and temperature at infinity for axisymmetric and three-dimensional media. A similar decay does not occur in the two-dimensional case and consequently, considerable caution must be exercised.

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.

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.

This card is not valid for a FE GMR.

An alternative method to account for the semi-infinite boundary is to model the boundary with enclosing elements (see ENCLosing-elements 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-Gmr  GMR1
  MATErial MAT1
  TREFerence 70
  HALF
  POINts
        1      0.00      0.00      0.00
        2      0.70      0.70      0.00
        .        .         .         .
        .        .         .         .
```

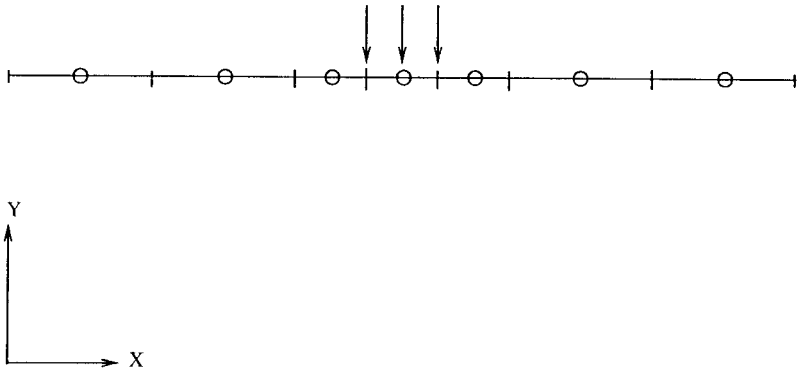


FIGURE FOR HALF CARD

CONTour-gmr TYPE

Status - OPTIONAL (2D and axisymmetric analyses only)

Full Keyword - **`CONTour-gmr`**

Function - Requests the generation of additional output for this GMR in the neutral file that can be used to create contour plots of results over a two-dimensional or axisymmetric region.

Input Variables -

TYPE (Alphanumeric) - OPTIONAL

Allowable values are QUAD and LINE

QUAD - The results over the 2D contour cells assume a quadratic variation.

LINE - The results over the 2D contour cells assume a linear variation. (Values at the mid-nodes of the cells are linearly interpolated from the values at the corner nodes.)

OFF - Results for 2D contours will not be calculated in the GMR (overrides CONTour-plot definition in **CASE)

Additional Information -

This card is not available in generalized axisymmetry or three-dimensional analyses.

In 3-D analysis, the CONTour-plot card can be used under the **CASE input for generating additional output in the neutral file which can be used to display contour plots on the surface of a hole, runner or cooling line. See the CONTour-plot card under **CASE for more information.

If the CONTour-plot card is input, a neutral file will automatically be created. The NEUTral-file card is therefore not required, however, if the NEUTral-file card is included, the options included on the NEUTral-file card (if any) have precedence over the CONTour-plot card.

If TYPE is not input, then in most cases **GPBEST** will assume QUAD. If **GPBEST** determines an excess number of nodes are generated relative to the number of boundary nodes, then LINE will be assumed, and the values of the variables of the mid-nodes of the cell are linearly interpolated from the values at the corner nodes. If a fine contour is required the user should input QUAD. If fast execution is important the user should input LINE.

The CONTour-gmr card definition under **GMR, overrides the CONTour-plot definition (if included) under **CASE for this GMR only.

Examples of Use -

```
**CASE
  TITLe  MOLD COMPONENT
  PLANe STRESS
  CONTour-plot
```

5.3.3 NODAL POINT DEFINITION

POINts

Status - REQUIRED (for defining the GMR)

Full Keyword - **POINts**

Function - This card initiates the definition of nodal points for the boundary element, the finite element and volume point of the GMR.

Input Variables - NONE

Additional Information -

Under certain conditions, **GPBEST** 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; i.e., when a GMR is obtained by translation and/or rotation of previously defined surfaces.

Nodal points used for hole element discretization **CANNOT** be defined at this point. Instead, the nodal points for holes must be defined under their respective sections.

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

While solving a structural-acoustics problem, a uniform thickness value for all points within a structural GMR is defined by the THICkness card. Thickness values defined for individual nodal points would override the uniform thickness value at specific nodes.

(NONE) NNODE X Y Z [THK]

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. For 2-D and axisymmetry problems, only two coordinates x and y (or r and z) need to be input.

THK (Real) - OPTIONAL (for a finite element GMR only). This value overrides the value input on the THICkness card at this node.

Additional Information -

In axisymmetry, the first coordinate represents the radial direction (referred to as the r-axis in this manual) and the second coordinate represents the axial direction (referred to as the z-axis). If ''COORdinate-system ZR'' appears in **CASE input, then the order is reversed. See the COORdinate-system input card in Section 5.1 for more information.

If the **GPBEST** data is input using the cylindrical coordinate system (see the COORdinate-system input card under **CASE input in Section 5.1), the order that the components are input is R,θ in 2-D, and R,θ,Z in 3-D. The R,θ,Z values should be input in place of the X,Y,Z values on this card where the θ component should be expressed in degrees.

This card is input once for each point.

User node numbering must be unique.

All node numbers must be less than or equal to 99999; for generalized axisymmetry, they must be less or equal to 9999.

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

Nodal points used for hole elements **CANNOT** be defined here. Instead, the nodal points for holes must be defined under their respective sections.

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 point coordinates in GMR1 for a 2-D analysis.

```
**GMR
  ID-Gmr  GMR1
  MATErial MAT1
  TREFerence 70.0
  POINts
        1     0.0     0.0
        2     0.5     0.0
        3     1.0     0.0
        4     1.0     0.5
        5     1.0     1.0
        6     0.5     1.0
        7     0.0     1.0
        8     0.0     0.5
```

2. Define a set of nodal coordinates in GMR1 for a 3-D analysis.

```
**GMR
  ID-Gmr  GMR1
  MATErial MAT1
  TREFerence 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
        .     .        . .
        .     .        . .
        .     .        . .
```

3. Define a set of nodal coordinates for a finite element GMR with the thickness at points 1, 2, 3, 4 as 0.05, 0.1, 0.1, 0.15, respectively.

```
**GMR
  ID-Gmr  GMR1  FINITE
  MATErial STRUCTURE
  THICkness   0.10
  POINts
        $     X      Y      Z     THK
        1    0.00   0.0    0.0    0.05
        2    0.50   0.0    0.0
        3    1.0    0.0    0.0
        4    1.5    0.0    0.0    0.15
```

5.3.4 SURFACE ELEMENT DEFINITION

SURFace 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 (Available only in 3-D, but not for acoustics or structural acoustics problems)

The name of a previously defined surface which will be used to create the current surface (defined as NAME on SURFace 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 (in 3-D) or line noded elements (2-D) may be defined using explicitly defined nodes. In this case REFNAME is not input, and the card following the SURFace card must be a TYPE-surface 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 TRANslate card and/or a set of cards consisting of a REFErence, DIREction and ROTAtion 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-surface variation of the new surface is automatically assumed to be the same as the reference surface (this cannot be changed).

REFNAME can only be used in three dimensional analysis. Boundary conditions and/or interface conditions may be applied to the automatically generated surfaces, however, it is the user responsibility to insure that the elements and nodes generated by the program are referenced by the correct numbers. The RESTart-gmr WRITE option may be useful in this respect. In any case, boundary conditions that are applied to the entire surface can be input without complications.

The current verions of **GPBEST** 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-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
 1003  5  6  7  107  207  206  205  105
 1004  7  8  1  105  201  208  207  107
NORMal 101 +
```

2. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFace  SURF1
TYPE-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFace SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFErence   1.0   0.0   0.0
 DIREction   0.0   0.0   1.0
 ROTAtion   180.0
NORMal 1001 +
```

TYPE-surface ATYPE BTYPE

Status - REQUIRED (if REFNAME not input)

Full Keyword - **`TYPE-surface`**

Function - This card defines the variation of field quantities over the elements of the current surface.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are LINE, QUAD and QUAR

LINEAR - linear shape function
BUBBLE - linear shape function with center node
QUADRATIC - quadratic shape function
QUARTIC - quartic shape function **(Not available in larger versions of GPBEST** .)

BTYPE (Alphanumeric) - OPTIONAL (for QUAD and QUAR in 3-D)

Allowable values are LAGR and SERE. The default is SERE

SERENDIPITY - Serendipity type shape function
LAGRANGIAN - Lagrangian type shape function

Additional Information -

See figure on subsequent pages.

A TYPE-surface card must be defined for each surface.

All of the elements of a single surface must have the same type of variation. Different surfaces of the same GMR may have different variation. When three-dimensional surfaces of different variation types meet, the element sides along the junction are constrained to have the variation of the element with the lower order. (see figure on subsequent pages).

A surface may consist of a single element. By contrast a single surface may define the entire boundary of a GMR.

In general, the use of QUARTIC elements is not effective. Better results can be obtained with QUADRATIC elements. An exception to this behavior is for problems which include CENTrifugal body forces where improved results can be obtained. A second exception is for the bending of long, thin beams where QUARTIC elements work well, since the displacement behavior along the span is very well represented by the quartic shape function.

QUADRATIC LAGRANGIAN elements are used as default for the exterior acoustic analysis.

QUARTIC SERENDIPITY elements are not recommended for general use.

Problems involving structural acoustics should use only QUADRATIC variation.

Examples of Use -

1. Specify that the field quantities vary quadratically over the elements of the surface SURF1.

```
SURFace  SURFl
TYPE-surface  QUAD SERE
ELEMents-surface
    101   1   2   3   103   203   202   201   101
    102   3   4   5   105   205   204   203   103
    .     .   .   .   .     .     .     .     .
    .     .   .   .   .     .     .     .     .
    .     .   .   .   .     .     .     .     .
    .     .   .   .   .     .     .     .     .
```

2D BOUNDARY ELEMENT FUNCTIONAL VARIATION

LINE

QUAD

QUAR

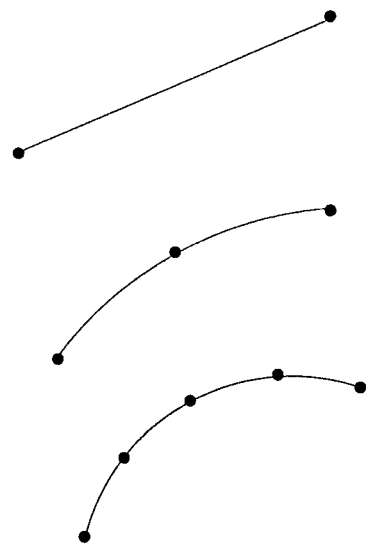


Figure for **GMR: TYPE card

(See element connectivity card for geometrical input)

3D BOUNDARY ELEMENT FUNCTIONAL VARIATION

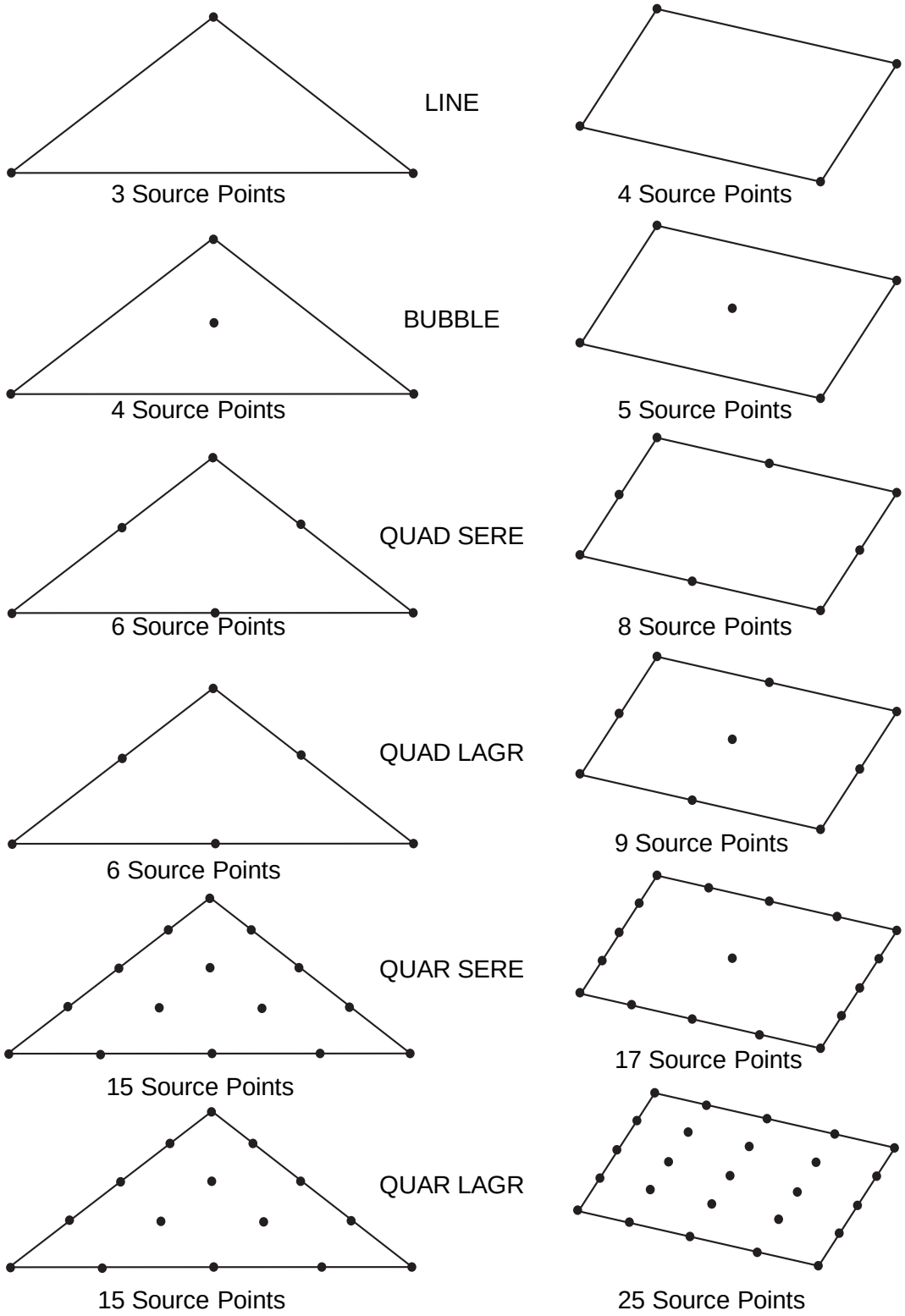


Figure for **GMR: TYPE card `(See element connectivity for geometrical input)`

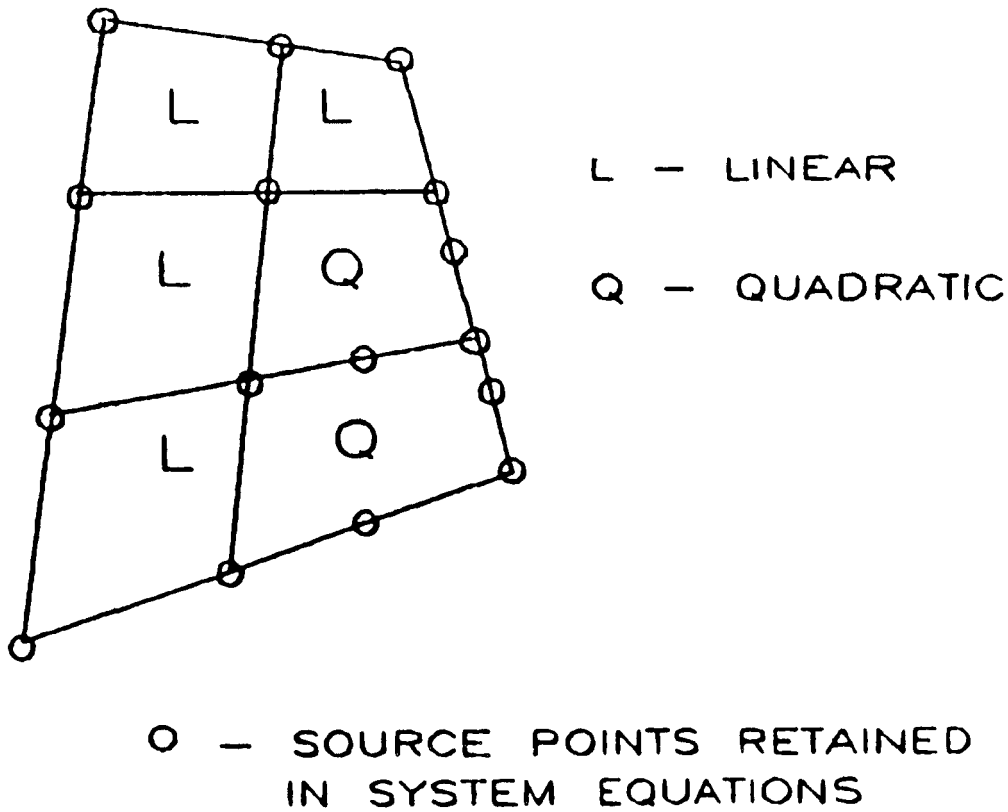


Figure for **GMR: TYPE card
Effect of mixed variation

GEOMetry ATYPE

Status - OPTIONAL (if REFNAME not input)

Full Keyword - **`GEOMetry`**

Function - This card defines the variation of geometry over the elements of the current surface.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are LINE and QUAD

LINEAR - linear shape function

QUADRATIC - quadratic shape function

Additional Information -

QUAD is the default for ATYPE.

In 3-D, when ATYPE is specified as LINE, **all of the elements on this surface** must be defined in terms of either 3 or 4 nodes. Geometry is internally expanded from 3 (or 4) noded elements to 6 (or 8) noded elements and the printout corresponds to the type of variation specified in the TYPE-surface card. Three and four noded elements can only be used if a GEOMetry LINE card is present.

In 2-D, when ATYPE for a surface is specified as LINE, **all of the elements on this surface** must be defined in terms of 2 nodes. Geometry is internally expanded from 2 to 3 noded elements. The printout corresponds to the type of variation specified in the TYPE-surface card. Two noded elements can only be used if a GEOMetry LINE card is present.

This card is not valid for problems of structural acoustics and periodic acoustics. The geometry is always considered to be the default (QUAD) type.

Examples of Use -

1. Specify the input geometry (points and connectivity) as linear over the elements of the surface SURF1 for a unit cube.

```
POINts
 1       0.0      0.0    0.0
 2       1.0      0.0    0.0
 3       1.0      1.0    0.0
 4       0.0      1.0    0.0
11       0.0      0.0    1.0
12       1.0      0.0    1.0
13       1.0      1.0    1.0
14       0.0      1.0    1.0
SURFace SURF1
  TYPE-surface QUAD
  GEOMetry LINE
  ELEMents-surface
 1    1   2   3   4
 2   11  12  13  14
 3    3   4  14  13
 4    1   2  12  11
 5    2   3  13  12
 6    4   1  11  14
```

ELEMents-surface

Status - REQUIRED (if REFNAME not input)

Full Keyword - **`ELEMents-surface`**

Function - Signals the beginning of the connectivity definition for surface elements of the current surface.

Input Variables - NONE

Additional Information -

For perodic acoustic analysis using an indirect boundary element formulation, only 6-noded and 8-noded shell type elements are available. For structural-acoustics problems, 6-noded and 8-noded shell elements and 3-noded frame elements are available.

(NONE) IFLAG NEL NODE1 ... NODEN NODEREF

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

Full Keyword - NO KEYWORD REQUIRED

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

Input Variables -

IFLAG (Alphanumeric) - OPTIONAL

IFLAG is omitted except in the case of a crack element. IFLAG = D identifies the first row of elements inside the crack front and IFLAG = T identifies the first row of elements outside the crack front.

NEL (Integer) - REQUIRED

User element number.

NODE1 ... NODEN (Integer) - REQUIRED

User node numbers of the three, four, six, eight or nine nodes (for 3-D), or two or three nodes (for 2-D or Axisymmetry) for defining the geometry of the element. Every surface element must have two, three, four, six, eight or nine nodes regardless of whether TYPE = LINE, QUAD or QUAR. (The shape functions for geometry are always quadratic).

NODEREF (Integer) - OPTIONAL

User-defined reference node for orienting a 3-noded frame element in structural-acoustics problems.

Additional Information -

This card is input once for each element.

The input card need not specify whether an element with two, three, four, six, eight or nine nodes is being defined. The starting point in 3-D may be any of the corner nodes. However, the direction of input around the element is arbitrary, as long as input is consecutive. For 2-D, the input must be consecutive starting with an end node. Adjacent elements must be defined in the same direction. The direction is defined with the NORMal card.

User element numbers must be unique and less than or equal to 99999, except for generalized axisymmetry, where the limit is 9999.

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

Crack elements must be 8-noded quadratic (serendipity) quadrilaterals for 3-D analyses. For 2-D and axisymmetry, crack elements must contain three nodes and employ a quadratic functional variation.

Definition of Geometry

The elements should be such that the mid-nodes for the sides emanating from the crack front must be at geometric quarter points. When these elements are specified, the CRACk card must be included.

Examples of Use -

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

```
SURFace  SIDE
TYPE-surface  QUAD
ELEMents-surface
   101   1   2   3   103   203   202   201   101
   102   3   4   5   105   205   204   203   103
   103   5   6   7   107   207   206   205   105
   104   7   8   1   105   201   208   207   107
NORMal 101 +
```

2. Crack Elements

```
SURFace  SIDE
TYPE-surface  QUAD
ELEMents-surface
   D   101   1   2   3     7    11    10     9     6
   D   102   3   4   5     8    13    12    11     7
   T   103   9  10  11    15    19    18    17    14
   T   104  11  12  13    16    21    20    19    15
```

For a finite element GMR, elements are associated with the current property attributes defined through a PROPerty card. Hence, for proper syntax, after the POINts card, there should be a PROPerty card and then an ELEMents-surface card. In case multiple properties are needed within the same FE region, the PROPerty and ELEMents-surface cards could be used more than once. The following example clarifies the point:

1. FE region using multiple materials.

```
**GMR
  ID-Gmr  GMR1  FINITE
  MATErial    STRUCTURE
  THICkness   0.2E-02
  POINts
    1   0.0   0.0   0.0
    ⋮
  PROPerty
  EMODULUS   1.0E+6
  POISSON   0.30
```

```
DENSITY   0.15
ELEMents-surface
  1    1   2   3   4   5   6   7   8
  2    5   6   7  14  13  11   7   1
  ⋮
 100  56  57  58  70  71  72  78  79
 PROPerty
 EMODULUS    1.0E+05
 POISSON    0.2
 DENSITY 0.10
 ELEMents-surface
   101   78   79   80   92   97   83  82  81
    ⋮
   200  100  105  102  172  175  174
```

In the above example, elements 1 to 100 are associated with the first PROPerty set and elements 101 to 200 are associated with the second PROPerty set.

Note that all the frame-type elements, whether they are used as stiffeners or pure frames, are positioned after all the 6-noded and 8-noded elements. This is currently required. The PROPerty attributes for frame-type elements contain additional quantities compared to those required for shell-type elements, viz., AREA, IXPR IYPR, etc. However, if one PROPerty set is used to define all shell-type and frame-type elements, the frame-type attributes would be automatically considered redundant for the shell-type elements. So if the Young's modulus, Poissons ratio and density are the same for shells and frames, only one PROPerty set is required.

Also note that 3-noded frame elements are only available for the FE structural definition. Each frame element may be oriented correctly with the help of a reference node, which should be declared in the POINts list of the same **GMR section.

Consider the following input structure:

```
**GMR
  ID-Gmr  GMR1  FINI
  MATErial  STRUCTURE
  MASS LUMPED
  OVDAmping 0.02
  POINts
    1    3.0   0.0   0.0
    2    2.12  2.12  0.375
    3    0.0   3.0   0.750
    4   -2.12  2.12  1.125
    5   -3.0   0.0   1.50
   100   0.0   0.0   100.0
  PROPerty
  EMODULUS   0.2E+07
  POISSON   0.3
  DENSITY   0.8E-02
  AREA   0.03
  IXPR   2.25E-04
  IYPR   2.50E-05
  ELEMents-surface
    1    1   2   3   100
```

Definition of Geometry

```
2    3   4   5   100
```

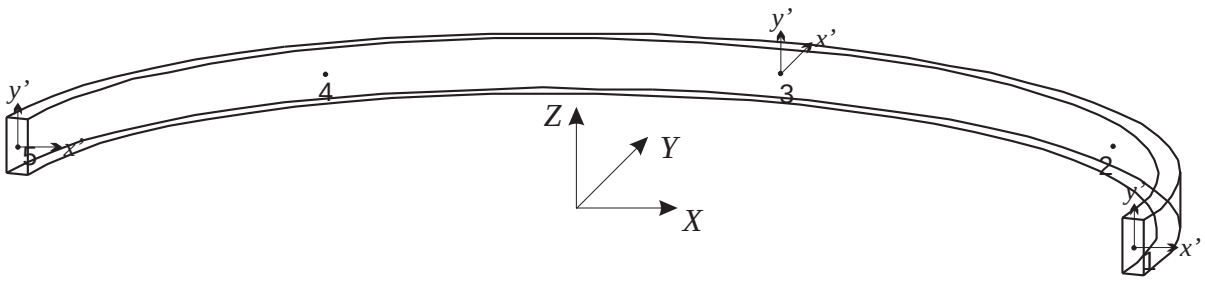


Figure for the FE ELEM Card: Frame Elements

The manner in which the $x' - y'$ axes are determined is as follows:

1. The nodal connectivity defines the local axis, z'.
2. If there is a reference node, a reference vector is formed as one going from the global origin to the reference node. This vector is normalized. Let it be denoted by r.

   If the reference node is not defined, r is assumed to be (0,0,1).

3. If z' is parallel to global z, then x' and y' are taken as the global x and y directions, irrespective of the reference node. Otherwise, $x' = r \times z'$ and $y' = z' \times x'$.

For the above input, reversing the values of IXPR and IYPR, viz.,

```
PROPerty
EMODULUS   0.2E+07
POISSON   0.3
DENSITY   0.8E-02
AREA   0.03
IXPR   2.50E-05
IYPR   2.25E-04
```

would give the following orientation:

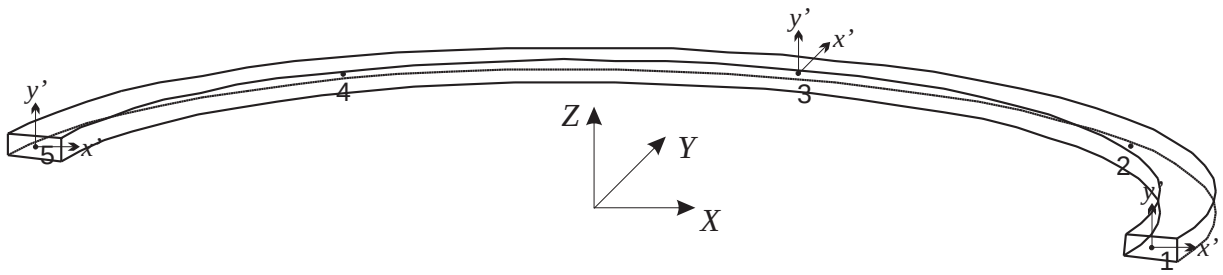
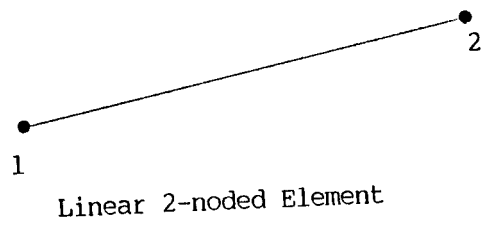


Figure for the FE ELEM Card: Frame Elements

2-D BOUNDARY ELEMENT FAMILY FOR GEOMETRICAL INPUT



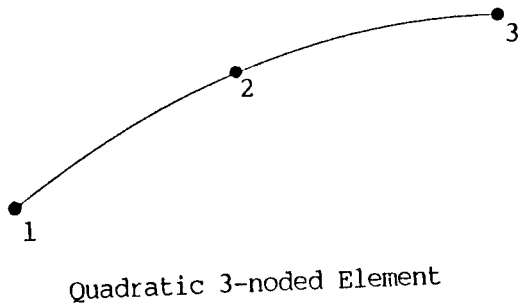
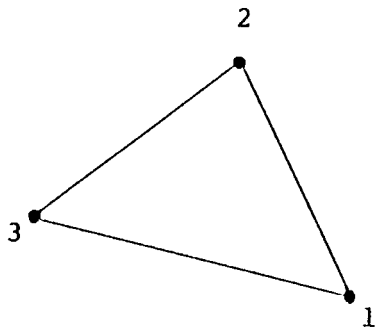
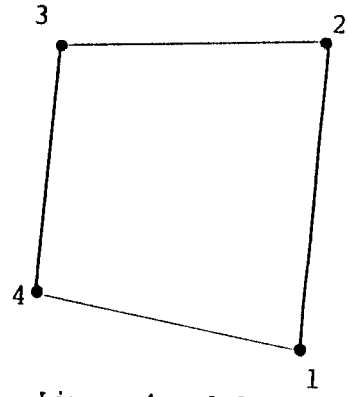


Figure for **GMR: Element connectivity card
(see TYPE-surface card for functional variation)

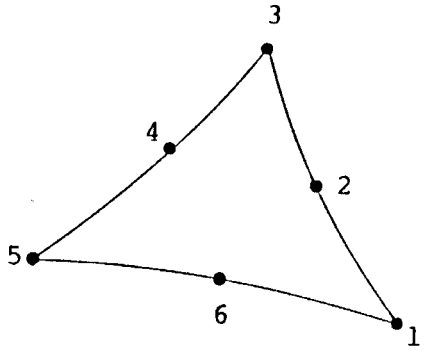
3-D BOUNDARY ELEMENT FAMILY FOR GEOMETRICAL INPUT



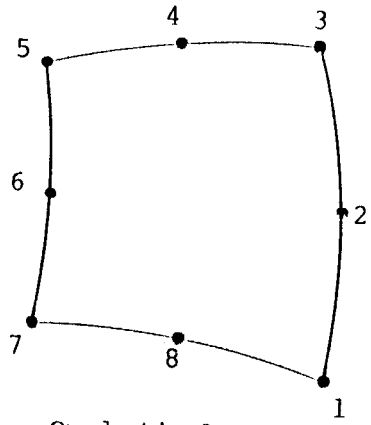
Linear 3-noded Element



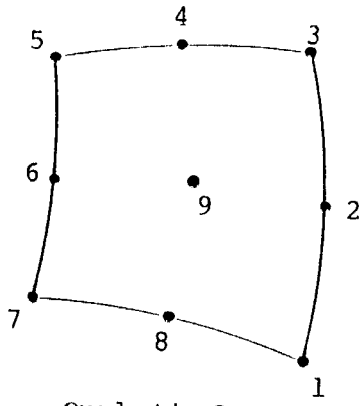
Linear 4-noded Element



Quadratic 6-noded Element



Quadratic 8-noded Element



Quadratic 9-noded Element

Figure for **GMR: Element connectivity card
(see TYPE-surface card for functional variation)

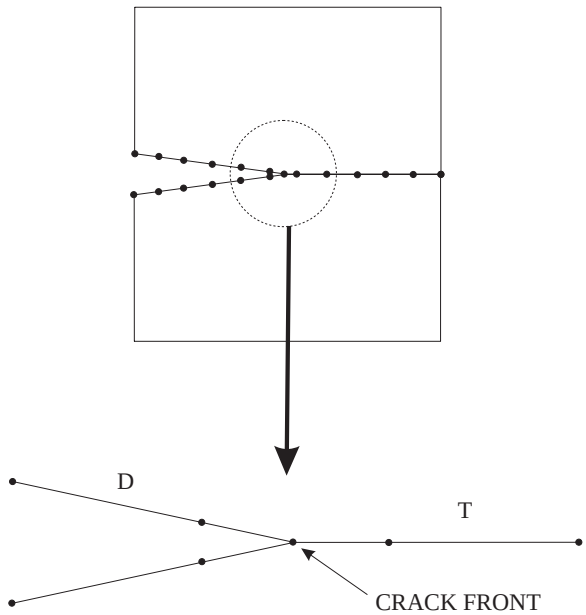


Figure for **GMR: 2-D Crack Element

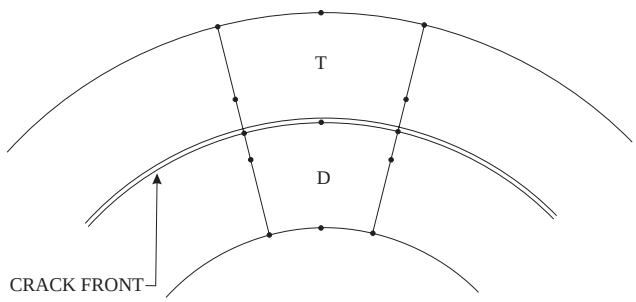
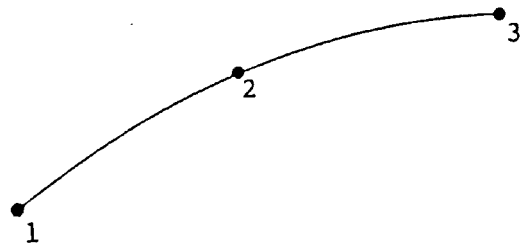
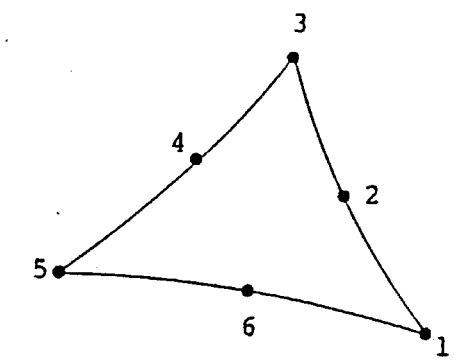


Figure for **GMR: 3-D Crack Element

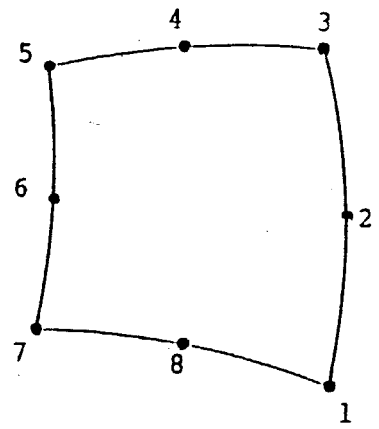
3-D FINITE ELEMENT FAMILY FOR GEOMETRICAL INPUT



Quadratic 3-noded Frame Element



Quadratic 6-noded Shell Element



Quadratic 8-noded Shell Element

Figure for **GMR: Finite Element Connectivity Card

TRANslate 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 -

If cylindrical coordinates are used for input, then only a translation in the Z-direction is allowed. (DELX, DELY will be assumed to be zero).

Available for 3-D only and generally a useful command for manual data preparation.

This card is not valid for structural-acoustics and periodic acoustics.

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-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFace SURNEW SURF1
 TRANslate   1.0  -1.0   0.0
 REFErence   1.0   0.0   0.0
 DIREction   0.0   0.0   1.0
 ROTAtion   180.0
NORMal 1001 +
```

REFErence 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.

GPBEST also accepts data defined using the cylindrical coordinate system. See the ‘‘COORdinate-system’’ system card under **CASE input.

Additional Information -

Available for 3-D only.

This card is not valid for structural-acoustics and periodic acoustics.

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-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFace SURNEW SURF1
 TRANslate   1.0  -1.0   0.0
 REFErence   1.0   0.0   0.0
 DIREction   0.0   0.0   1.0
 ROTAtion   180.0
NORMal 1001 +
```

DIREction 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 -

Available for 3-D only.

This card is not valid for structural-acoustics and periodic acoustics.

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-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFace SURNEW SURF1
 TRANslate   1.0  -1.0   0.0
 REFErence   1.0   0.0   0.0
 DIREction   0.0   0.0   1.0
 ROTAtion   180.0
NORMal 1001 +
```

ROTAtion 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 -

Available in 3-D only.

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).

This card is not valid for structural-acoustics and periodic acoustics.

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-surface  QUAD
ELEMents-surface
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFace SURNEW SURF1
 TRANslate   1.0  -1.0   0.0
 REFErence   1.0   0.0   0.0
 DIREction   0.0   0.0   1.0
 ROTAtion   180.0
NORMal 1001 +
```

CRACk NODE1 NODE2 . . . NODEN

Status - OPTIONAL

Full Keyword - **CRACk**

Function - Defines the nodes at the crack front

Input Variables -

NODE1 NODE2 . . . NODEN (Integer) - REQUIRED

User node numbers of the crack front.

Additional Information -

This card is required if crack elements are specified.

Only one crack front is allowed in a problem.

Examples of Use -

1. Define nodes at the crack front.

```
SURFace SIDE
TYPE-surface QUAD
ELEMents-surface
  D  101  1   2   3   7   11  10  9   6
  D  102  3   4   5   8   13  12  11  7
  T  103  9   10  11  15  19  18  17  14
  T  104  11  12  13  16  21  20  19  15
NORMal  101+
CRACk   9   10  11  12  13
```

5.3.5 NORMAL DEFINITION

NORMal NEL1 F1 NEL2 F2 ... NELN FN

Status - REQUIRED - (for 3-D analysis)

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 (for 3-D only).

F1 (Alphanumeric) - REQUIRED

Flag relating the outer normal direction on the element NEL1 to the input node ordering. Allowable values for F1 are + and – .

If the input ordering of the nodes for element NEL1 (or NEL2 or NELN) is counter-clockwise when viewed from outside the element, then F1 = +. Otherwise F1 = – (right hand rule).

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 -

With the outer normal defined for one element on each disjoint boundary, the outer normals of all other elements in the (3-D) GMR are automatically determined by the program.

A ‘disjoint’ boundary should not be confused with a ‘surface’. A surface is merely a user convenience, permitting the collection of any number of elements without regard to connectivity. On the other hand, each disjoint boundary is the collection of contiguous elements. For example, the user could employ a single surface to model a thick-walled hollow sphere, but there will always be two disjoint boundaries. (see Figures on subsequent pages)

The number of (element number, flag) pairs to be input for a given GMR is exactly equal to the number of disjoint boundaries which make up that GMR.

All (element number, flag) pairs must be input on the same normal card. Only one NORMal card per region is allowed.

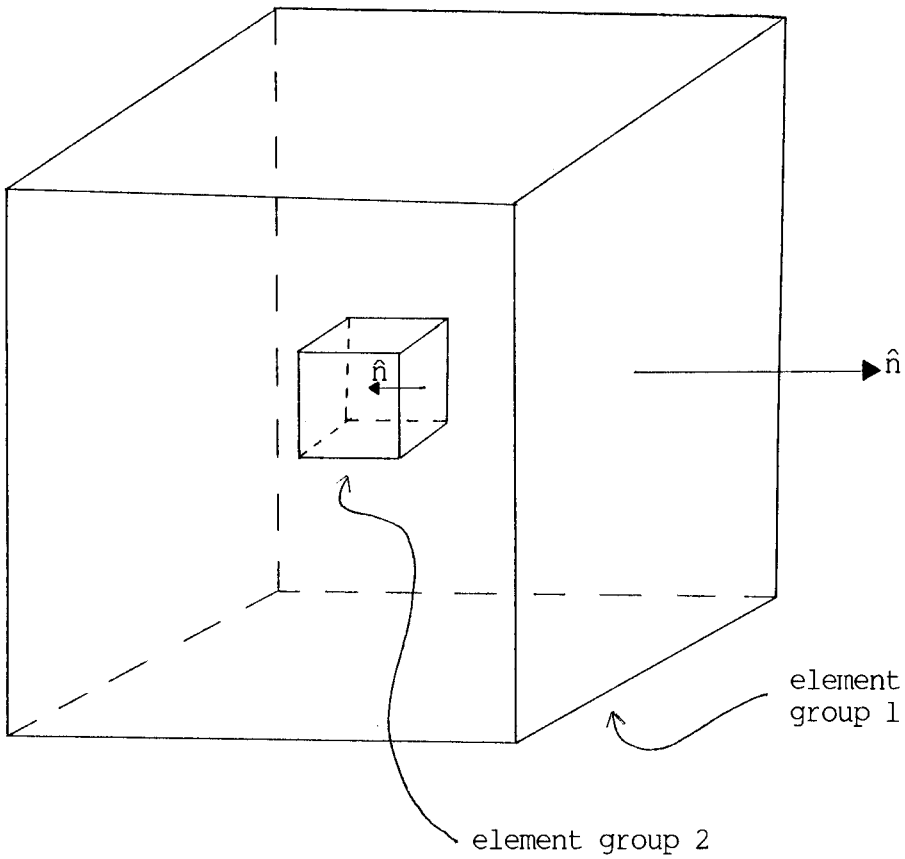
This card does not apply to enclosing elements. All of the connectivity for enclosing elements must be defined such that their normals are positive, i.e., the NORMal card does not apply to enclosing elements.

A NORMal card is not necessary for a finite element GMR. It is also redundant in case of an indirect boundary element acoustic GMR.

Examples of Use -

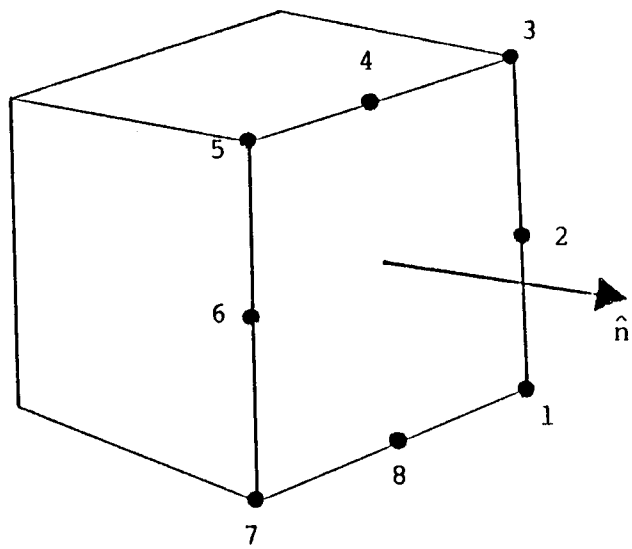
1. Define the direction of outward normal to the element 101 as being positive in relation to input node ordering.

```
SURFace  SIDE
  TYPE-surface  QUAD
  ELEMents-surface
      101     1     2     3   103   203   202   201   101
      102     3     4     5   105   205   204   203   103
      103     5     6     7   107   207   206   205   105
      104     7     8     1   105   201   208   207   107
      501   501   502   503   603   703   702   701   601
      502   503   504   505   605   705   704   703   603
  ENCLosing-elements
      501   107  1001   1002 1003  1004  1005  1006   104
  NORMal 101 +   501 -
```

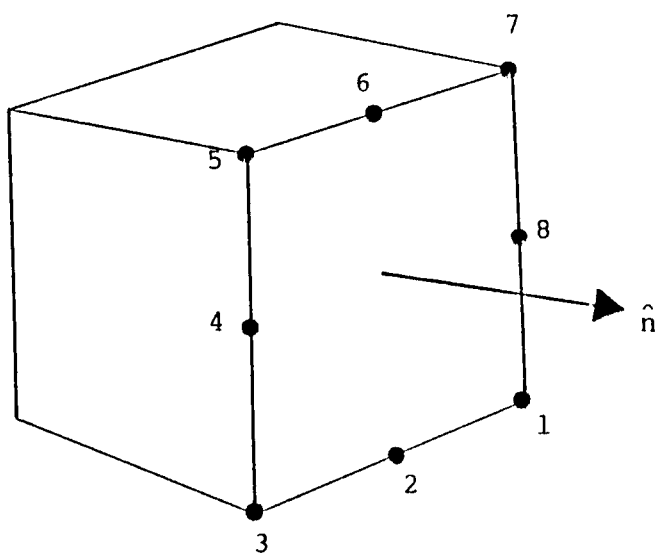


Element groups 1 and 2 are disjoint sections and therefore normals for one element in each section must be defined.

Figure for **GMR: NORM card
Three-dimensional disjoint (unconnected) elements



input shown is for
FLAG = +
on NORM card for 3-D



input shown is for
FLAG = -
on NORM card for 3-D

Figure for **GMR: NORM card

Three-dimensional outer normal convention
(In 3-D a mixed convention is allowed, as long as one and only one normal per disjoint section is defined)

NORMal SIGN

Status - REQUIRED - (for 2-D or axisymmetry analysis)

Full Keyword - **NORMal**

Function - Defines the outer normal direction in the current GMR.

Input Variables -

SIGN (symbol) - REQUIRED

Allowable symbols are " $+$ " or " $-$ "

$+$ defines the outward normal as up when numbering an element from right to left while looking down the z axis (see figure).

$-$ defines the outward normal as down when numbering an element from right to left while looking down the z axis (see figure).

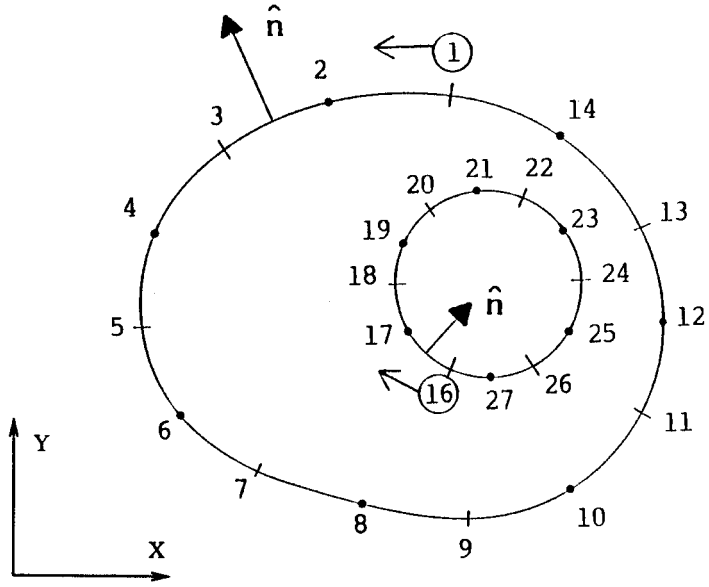
Additional Information -

All elements, including enclosing elements, of a GMR must follow the same convention.

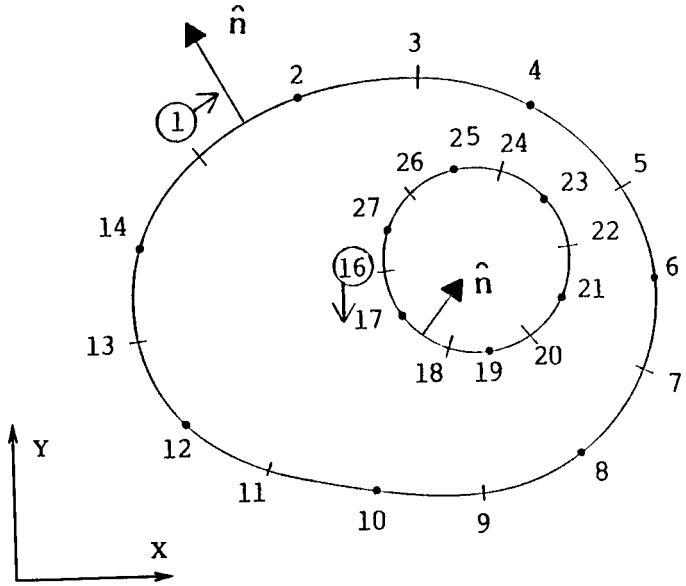
Examples of Use -

1. Define the direction of the outward normal to the surface SURF1 as positive.

```
SURFace  SURF1
TYPE-surface  QUAD
ELEMents-surface
    1    1    2    3
    2    3    4    5
    3    5    6    7
    4    7    8    1
NORMal    +
```



input shown is for
FLAG = +
on NORM card for 2-D



input shown is for
FLAG = -
on NORM card for 2-D

Figure for **GMR: NORM card

Two-dimensional outer normal convention
(In 2-D, all elements in a single GMR must use the same convention)

5.3.6 ENCLOSING ELEMENT DEFINITION

ENCLosing-elements

Status - OPTIONAL

Full Keyword - **`ENCLosing-elements`**

Function - Signals the beginning of the connectivity definition for enclosing elements of the boundary element 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

Additional Information -

In a GMR of infinite or semi-infinite extent, it is neccessary to use respectively, either the EXTErior 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 unmodeled 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 neccessary 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 neccessary 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.

Examples of Use -

1. Define enclosing elements for an axisymmetric body.

```
ENCLosing-elements
  55   95  105  115
  56  115  125  135
```

(NONE) NEL NODE1 .... NODEN

Status - REQUIRED (minimum of one card if ENCLosing-elements 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)

NODE1 . . . NODEN (Integer) - REQUIRED

User number for the node for defining the geometry of the enclosing element.

N = 8 (for 3-D)

N = 3 (for 2-D and Axisymmetry)

Additional Information -

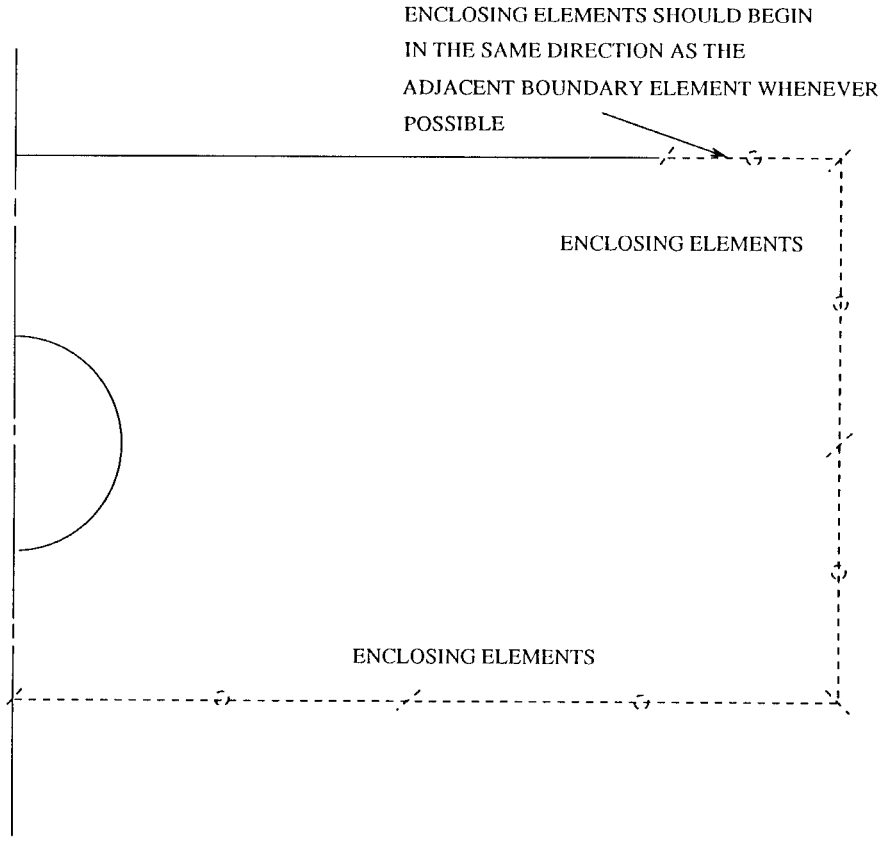
Only **EIGHT** noded enclosing elements are allowed in 3-D and **THREE** noded enclosing elements are allowed in 2-D and axisymmetry.

All of the connectivity for enclosing elements must be defined such that their normals are positive, i.e., NORMal card does not apply to enclosing elements.

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....
```



ENCLOSING ELEMENTS SHOWN FOR AXISYMMETRIC ANALYSIS

ALSO AVAILABLE FOR 3-DIMENSIONAL AND 2-DIMENSIONAL ANALYSES

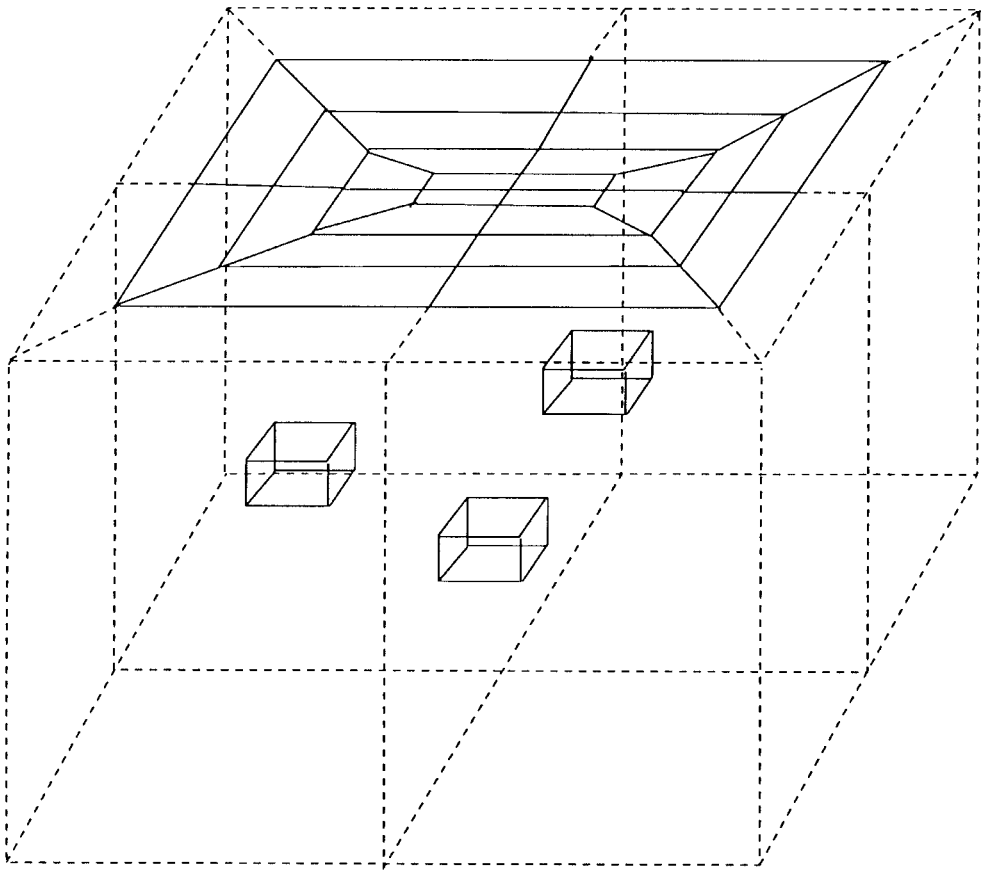


FIGURE FOR THREE-DIMENSIONAL ENCLOSING ELEMENTS
FOR A PROBLEM WITH THREE VOIDS IN A HALF SPACE

——————— BOUNDARY ELEMENTS

- - - - - - - - - - - - - - ENCLOSING ELEMENTS

5.3.7 DISCONTINUITY ELEMENT DEFINITION

DISCcontinuity-elements

Status - OPTIONAL (available only in Elastostatic and Forced Vibration analysis)

Full Keyword - **`DISCcontinuity-elements`**

Function - Signals the beginning of the connectivity definition for discontinuity elements of the GMR. These elements are used in (both elastostatic and forced vibration) stress analysis to model surfaces with known displacement discontinuity.

Input Variables - NONE

Additional Information -

These elements are usually defined inside a GMR and used to model movements induced by faulting or mining.

Only eight-noded elements are allowed in 3D, while only three-noded elements are allowed in 2D. Discontinuity elements are not available in axisymmetric analysis.

The known values of displacement discontinuity must be input as a **BODY force.

Examples of Use -

1. Define discontinuity elements for simulating a seismic disturbance in a two dimensional model. Figure (a) illustrates the problem.

```
DISCcontinuity-elements
  501   41   42   43
  502   43   44   45
```

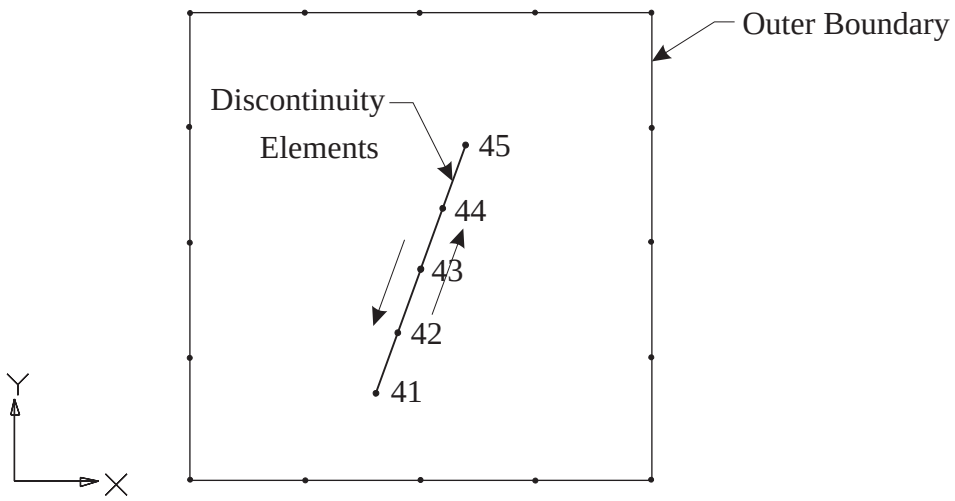


Figure for DISCcontinuity-elements: (a) Two Dimensional Discontinuity Elements

2. Model a sliding discontinuity in a cube. See Figure (b).

```
DISCcontinuity-elements
  41      81   82   83   84   85   86   87   88
```

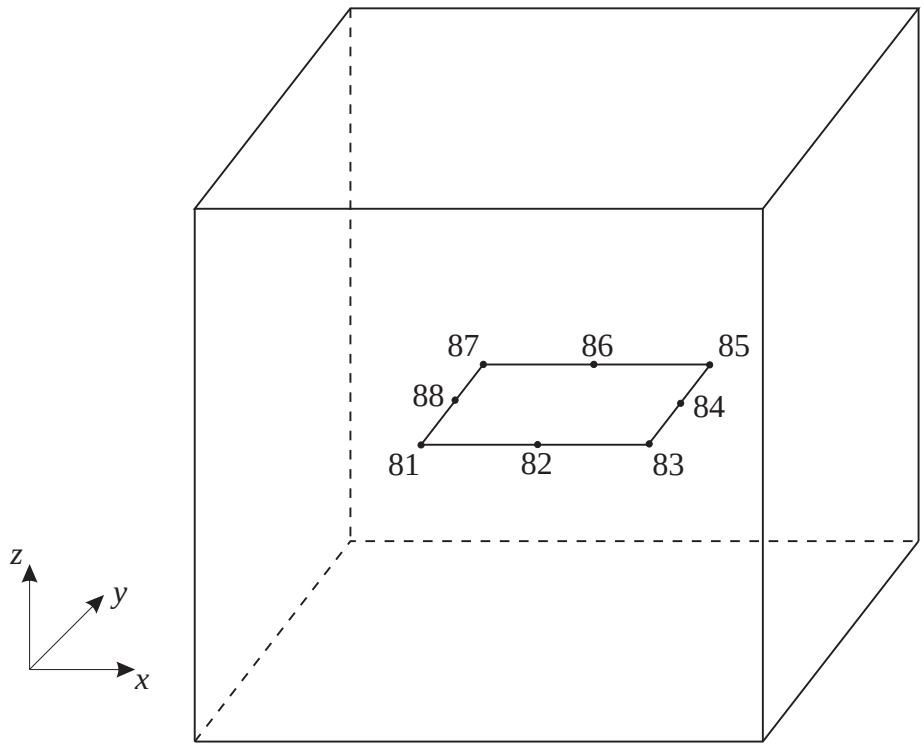


Figure for DISCcontinuity-elements: (b) Three Dimensional Discontinuity Element

(NONE) NEL NODE1 .... NODEN

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

Full Keyword - **NO KEYWORD REQUIRED**

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

Input Variables -

NEL (Integer) - REQUIRED

User element number

NODE1 . . . NODEN (Integer) - REQUIRED

User number for the node for defining the geometry of the discontinuity element.

Additional Information -

Only eight-noded elements are allowed in 3D, while only three-noded elements are allowed in 2D.

The node numbers refer to POINts that are defined in **GMR. They cannot refer to SAMPling-points.

The connectivity of discontinuity elements affects which direction the applied relative motion will act along. See Section 5.6.7 for a discussion of normal direction and orientation of discontinuity elements.

Examples of Use -

1. Define 3D discontinuity elements.

```
DISCcontinuity-elements
  1001  1  2  3  103  203  202  201  101
  1002  3  4  5  105  205  204  203  103
```

5.3.8 LINE ELEMENT DEFINITION

LINE-element-definition

Status - OPTIONAL

Full Keyword - **`LINE-element-definition`**

Function - This card initiates the definition of line elements which are used to represent embedded holes, runners and cooling lines within the current GMR.

Input Variables - NONE

Additional Information -

In the present implementation, holes, runners and cooling lines are 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. However, the curvature is treated in an approximate manner and some loss of accuracy may result. The radius is constant along the length of the element.

Available in 3-D ELASticity, HEAT transfer and thermal DISTortion analysis.

In previous versions of **GPBEST** , the keyword HOLE was used for this card. **GPBEST** will still accept data sets with the keyword HOLE used in place of the keyword LINE-element-definition.

Examples of Use -

1. Define a hole consisting of two quadratic hole elements.

```
LINE-element-definition
POINts-line
  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-line  QUAD
HOLE
ELEMents-surface
   101   0.1  1001  1002  1003
   102   0.1  1003  1004  1005
$(end of hole input)
```

POINts-line

Status - REQUIRED (if LINE-element-definition is input)

Full Keyword - **`POINts-line`**

Function - This card initiates the definition of line nodal points for the current GMR.

Input Variables - NONE

Additional Information -

Line nodal points must be defined separately from the surface nodal points.

(NONE) NNODE X Y Z

Status - REQUIRED (if LINE-element-definition is input)

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for a line node.

Input Variables -

NNODE (Integer) - REQUIRED

User number for the line node.

X,Y,Z (Real) - REQUIRED

Cartesian coordinates of the of the line node.

GPBEST also accepts data defined using the cylindrical coordinate system. See the ‘‘COORdinate-system’’ system card under **CASE input.

Additional Information -

This card is input once for each point.

User node numbering must be unique within the problem. This includes surface, line and sampling points.

All point numbers must be less than or equal to 99999.

All line nodes should be located on the interior of the GMR. Additionally, line nodes should not be coincident with surface or interior points.

Examples of Use -

1. Define a set of node numbers and the cartesian coordinates for nodes on two line elements.

```
LINE-element-definition
POINts-line
   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
```

TYPE-line ETYPE

Status - REQUIRED (if LINE-element-definition is input)

Full Keyword - **`TYPE-line`**

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

Only one TYPE-line card for lines is allowed per GMR.

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

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

All of the three-noded line elements in a single GMR will have the same type of variation specified on the TYPE-line card. This holds true even when different types of line elements (holes, runners, cooling lines) are present in the same GMR.

This card has no effect on the circumferential variation of the field variables about the line which are assumed to be constant in the circumferential direction.

Examples of Use -

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

```
LINE-element-definition
POINts-line
    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
TYPE-line LINEAR
  HOLE
  ELEMents-line
    1001  0.2   101   102   103
    1002  0.2   103   104
```

HOLE

Status - OPTIONAL

Full Keyword - **HOLE**

Function - Specifies that the subsequent line element connectivity definition represents a hole embedded in the current GMR.

Input Variables - NONE

Additional Information -

The kind of line element (HOLE, RUNNer or CLINe) must be defined before the line element connectivity is given. The choices available for line elements are HOLE, RUNNer or CLINe. All three may be contained in the same GMR. However, only one definition per region of each kind is allowed.

In earlier versions of **GPBEST** , HOLE was the only line element option available. Hence, it was not necessary to specify the kind of line element. In order to support **GPBEST** data input created prior to this change, **GPBEST** will assume line elements to be HOLE if nothing is specified.

Examples of Use -

1. Define line element data for a hole, runner and cooling line, each with two quadratic elements.

```
LINE-element-definition

POINts-line
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   0.75  1.0
    . . .
    115  25.0   0.0   1.0

TYPE-line QUADRATIC

  HOLE
  ELEMents-line
    1001  0.2   101   102   103
    1002  0.2   103   104   105

  RUNNer  0.155    1199.0    220.8
  TREFerence-line   450.0
  CONVection-line   1.0E+5
  ELEMents-line
    1003  0.05   106   107   108
    1004  0.05   108   109   110

  CLINe  0.5626     998.5   4184.0
  TREFerence-line   25.0
  CONVection-line   5.0E+4
  ELEMents-line
    1005  0.1   111   112   113   10.0
    1006  0.1   113   114   115   10.0
```

RUNNer COND DENS SPEC

Status - OPTIONAL

Full Keyword - **RUNNer**

Function - Specifies that the subsequent line element connectivity definition represents a runner embedded in the current GMR with the defined material properties. (A runner is a cylindrical region of different material embedded in the current GMR. It is used in injection mold analysis).

Input Variables -

COND (Real) - REQUIRED

Conductivity of the runner.

DENS (Real) - REQUIRED

Density of the runner.

SPEC (Real) - REQUIRED

Specific heat of the runner.

Additional Information -

The kind of line element (HOLE, RUNNer or CLINe) must be defined before the line element connectivity is given. The choices available for line elements are HOLE, RUNNer or CLINe. All three may be contained in the same GMR. However, only one definition per region of each kind is allowed.

Examples of Use -

1. Define line element data for a hole, runner and cooling line, each with two quadratic elements.

```
LINE-element-definition

POINts-line
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   0.75  1.0
    . . .
    115  25.0   0.0   1.0

TYPE-line QUADRATIC

  HOLE
  ELEMents-line
    1001  0.2   101   102   103
    1002  0.2   103   104   105

  RUNNer  0.155    1199.0    220.8
  TREFerence-line   450.0
  CONVection-line   1.0E+5
  ELEMents-line
    1003  0.05   106   107   108
    1004  0.05   108   109   110

  CLINe  0.5626     998.5   4184.0
  TREFerence-line   25.0
  CONVection-line   5.0E+4
  ELEMents-line
    1005  0.1   111   112   113   10.0
    1006  0.1   113   114   115   10.0
```

CLINe COND DENS SPEC

Status - OPTIONAL

Full Keyword - **`CLINe`** (Cooling Line)

Function - Specifies that the subsequent line element connectivity definition represents a cooling line embedded in the current GMR with the defined material properties. (A cooling line is a cylindrical region of fluid moving at a prescribed velocity embedded in the current GMR. It is used in injection mold analysis).

Input Variables -

COND (Real) - REQUIRED

Conductivity of the fluid.

DENS (Real) - REQUIRED

Density of the fluid.

SPEC (Real) - REQUIRED

Specific heat of the fluid.

Additional Information -

The kind of line element (HOLE, RUNNer or LINE-element-definition) must be defined before the line element connectivity is given. The choices available for line elements are HOLE, RUNNer or CLINe. All three may be contained in the same GMR. However, only one definition per region of each kind is allowed.

The velocity of the fluid is defined on the line element connectivity card.

Examples of Use -

1. Define line element data for a hole, runner and cooling line, each with two quadratic elements.

```
LINE-element-definition

POINts-line
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   0.75  1.0
    . . .
    115  25.0   0.0   1.0

TYPE-line QUADRATIC

  HOLE
  ELEMents-line
    1001  0.2   101   102   103
    1002  0.2   103   104   105

  RUNNer  0.155    1199.0    220.8
  TREFerence-line   450.0
  CONVection-line   1.0E+5
  ELEMents-line
    1003  0.05   106   107   108
    1004  0.05   108   109   110

  CLINe  0.5626     998.5   4184.0
  TREFerence-line   25.0
  CONVection-line   5.0E+4
  ELEMents-line
    1005  0.1   111   112   113   10.0
    1006  0.1   113   114   115   10.0
```

TREFerence-line TEMP

Status - OPTIONAL

Full Keyword - **`TREFerence-line`**

Function - This card defines the reference (or initial) temperature (i.e., the datum temperature of the zero stress-strain state) of the runner or cooling line element at the beginning of a temperature- dependent problem.

Input Variables -

TEMP (Real) - REQUIRED

Additional Information -

If this card is not input, the reference temperature is assumed to be the same as the region containing the runner and cooling line elements.

All runner elements in a region must have the same reference temperature, and all cooling line elements must have the same reference temperature. The reference temperature of the runner elements can, however, be different from the reference temperature of the cooling lines.

Examples of Use -

1. Define line element data for a hole, runner and cooling line, each with two quadratic elements.

```
LINE-element-definition

POINts-line
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   0.75  1.0
    . . .
    115  25.0   0.0   1.0

TYPE-line QUADRATIC

  HOLE
  ELEMents-line
    1001  0.2   101   102   103
    1002  0.2   103   104   105

  RUNNer  0.155    1199.0    220.8
  TREFerence-line   450.0
  CONVection-line   1.0E+5
  ELEMents-line
    1003  0.05   106   107   108
    1004  0.05   108   109   110

  CLINe  0.5626     998.5   4184.0
  TREFerence-line   25.0
  CONVection-line   5.0E+4
  ELEMents-line
    1005  0.1   111   112   113   10.0
    1006  0.1   113   114   115   10.0
```

CONVection-line C1
RESIstance-line R1

Status - OPTIONAL

Full Keyword - **`CONVection-line`**

Full Keyword - **`RESIstance-line`**

Function - Identifies an interface with thermal convection resistance between the runner or cooling line and the corresponding surface of the hole surrounding the runner cooling line. The flux across this interface is linearly related to the temperature difference between the two surfaces.

Input Variables -

C1 (Real) - REQUIRED

Thermal convection coefficient (C)

R1 (Real) - REQUIRED

Thermal resistance coefficient ($R = \frac{1}{C}$)

Additional Information -

C1 is the reciprocal of R1.

The CONVection-line option utilizes the relationship:

$$q_1 = C1(\theta_1 - \theta_2)$$

The RESIstance option utilizes the relationship:

$$q_1 = \frac{1}{R1}(\theta_1 - \theta_2)$$

where:

θ_1	local temperature of GMR 1.
θ_2	local temperature of GMR 2.
q_1	local heat flux from GMR 1.

The user is responsible for providing C or R in the proper units consistent with the specification of material properties, geometry and boundary conditions.

The convection (C) or resistance (R) should be a positive real number ($R > 0$). If zero is input, the coefficient will be automatically reset to 1.0E--10.

ELEMents-line

Status - REQUIRED (if LINE-element-definition is input)

Full Keyword - **`ELEMents-line`**

Function - Signals the beginning of the connectivity definition for hole, runner and cooling line elements in the current surface.

Input Variables - NONE

Additional Information - NONE

(NONE) NEL RADIUS NODE1 ... NODEN VELO

Status - REQUIRED (if LINE-element-definition is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the radius and connectivity for a single hole, runner or cooling line element.

Input Variables -

NEL (Integer) - REQUIRED

User element number.

RADIUS (Real) - REQUIRED

Radius of the hole, runner or cooling line.

NODE1 ... NODEN (Integer) - REQUIRED

User node numbers of the two or three hole nodes defining the geometry of the hole.

VELO (Real) - REQUIRED for cooling lines only

Velocity of the fluid in the cooling line.

Additional Information -

This card is input once for each 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 line elements, must be unique within a problem.

All line element numbers must be less than or equal to 99999.

All of the nodes referenced in the line element connectivity must have been defined previously as POINts-line under the LINE-element-definition keyword. That is, line elements can only connect line nodes. No surface nodes can be referenced by a line element.

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

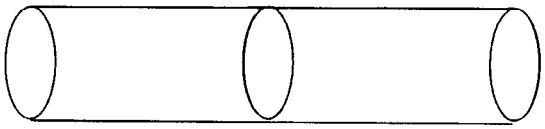
The radius is assumed constant along the entire length of the hole element.

The velocity of every cooling line element in a cooling circuit must have the same velocity. Cooling line elements in the same GMR may have different velocities if they belong to different circuits.

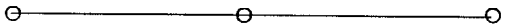
Examples of Use -

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

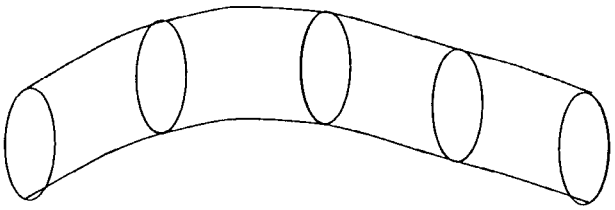
```
LINE-element-definition
POINts-line
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   1.0   1.0
TYPE-line LINEAR
  HOLE
  ELEMents-line
    1001  0.2   101   102   103
  RUNNer   0.015   460.0   7800.0
  ELEMents-line
    1002  0.2   103   104
```



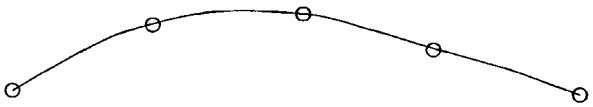
A Straight Hole, Runner or Cooling Line



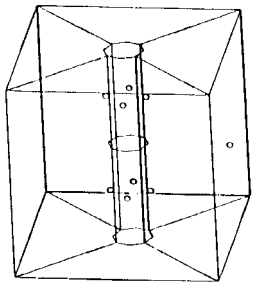
A Line Element



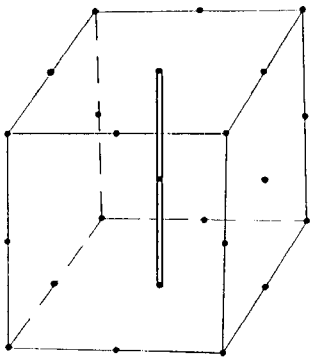
A Curved Hole, Runner or Cooling Line



Modeled With Two Quadratic Line Elements



Conventional 3-D BEM Discretization of Cube and Hole



Discretization Using One Quadratic LINE-element-definition

5.3.9 SAMPLING POINT AND SAMPLING SURFACE DEFINITION

SAMPling-points POIN

Status - OPTIONAL

Full Keyword - **`SAMPling-points`**

Function - This card signals the fact that a set of sampling points for which results are requested at any point on or in the body, will be provided for the current GMR.

Input Variables -

POINTS (Alphanumeric) - REQUIRED

Additional Information -

This card is used to define points at which displacements, stresses, strains, temperatures, pressures and fluxes are to be calculated.

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

The sampling point definition is also used in conjunction with the sampling surface definition. See next subsection.

The sampling point definition has no effect on the solution of the boundary element system.

This card is not valid for a FE GMR in structural-acoustics problems.

Examples of Use -

1. Request result information at three interior points

```
SAMPling-points POINTS
 1001   0.333  0.25  0.0
 1002   0.25   0.1  10.0
 1003   0.2    0.5  20.2
```

(NONE) NNODE X Y Z

Status - REQUIRED (if SAMPling-points 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. For 2-D and axisymmetry problems only x (or r) and y (or z) coordinates are needed.

GPBEST also accepts data defined using the cylindrical or reversed axisymmetric coordinate system. See the ‘‘COORdinate-system’’ system card under **CASE input.

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.

Point numbers must be less than or equal to 99999.

SAMPling-surfaces SURF NAME

Status - OPTIONAL

Full Keyword - **`SAMPling-surfaces`**

Function - This card initiates the definition of a sampling surface for the current GMR (use in post-processing of results).

Input Variables -

SURFACE (Alphanumeric) - REQUIRED

NAME (Alphanumeric) - REQUIRED

This is a user defined name for the current sampling surface. May be up to 8-characters long.

Additional Information -

The name of each sampling surface in a GMR must be unique.

The sampling surface is only used for the post-processing display of results. It has no effect on the solution of the boundary element system.

There may be more than one sampling surface in a GMR. The number of sampling surfaces in a GMR is only limited by the total number of sampling points allowed in the total analysis. See global parameters in section 5.0 or use the PRINtout-control LIMIT card in **CASE input.

Each sampling surface can be generated in a pre-processor as an independent group of elements that are associated with a particular GMR. Then by using the **GPBEST** prebest translators the sampling surface data is automatically placed in the **GPBEST** data set in the appropriate GMR. After **GPBEST** is run, the postbest translator will recreate the independent group (similar to a GMR group) that can be viewed in a post-processor. See the BESTVIEW Manual under the translator section for details. This option is also available in the BESTVIEW graphic program.

Although **GPBEST** does not require any special naming convention for sampling surfaces, the translators for pre- and post-processing require sampling surface names to start with the characters ‘‘SS’’ and to conclude with the ‘‘_’’ character followed by the GMR name which they are associated with. For example, the names of two sampling surfaces in GMR ‘‘TOP’’ would be SS1_TOP and SS2_TOP. The character after the ‘‘SS’’ distinguishes the different surfaces in the GMR. It can be a number 0-9 or a letter A-Z.

In an acoustic analysis, radiated acoustic power is calculated for each sampling surface. For this reason, the outward normal of each element of the sampling surface must be consistent with all other elements in the same sampling surface. The outward normal of an element is defined according to the right-hand-rule with respect to the connectivity.

This card is not valid for a FE GMR in structural-acoustics problems.

Examples of Use -

```
SAMPling-points POINTS
   1001   1.0   2.0   3.0
   1002   1.0   3.0   3.0
   ⋮

SAMPling-surfaces SURFACE SS1_TOP
   ELEMents-sampling-surf
   101   1001   1002   1003   1004   1005   1006   1007   1008
   102   1001   1002   1003   1009   1010   1011   1012   1013
   ⋮

SAMPling-surfaces SURFACE SSB_BACK
   ELEMents-sampling-surf
   202   2001   2002   2003   2004   2005   2006
   ⋮
```

ELEMents-sampling-surf

Status - REQUIRED (if SAMPling-surfaces SURF is input)

Full Keyword - **`ELEMents-sampling-surf`**

Function - Signals the beginning of the connectivity definition for sampling surface elements of the current surface.

Input Variables - NONE

Additional Information - NONE

(NONE) NEL NODE1 NODE2 . . . NODEN

Status - REQUIRED (Minimum of one if SAMPling-surfaces SURF is input)

Full Keyword - **NO KEYWORD REQUIRED**

Function - Each card defines the connectivity of a single sampling surface element.

Input Variables -

NEL (INteger) REQUIRED

User element number

NODE1 . . . NODEN (INteger) REQUIRED

User node numbers may be 3, 4, 6, 8 or 9 for 3D analysis. The elements should be compatible with the graphic program that will be used in the post-processing of results.

Additional Information -

- This card is input once for each element.
- All nodes specified in the connectivity must be input under the SAMPling-pointss POINTS definition.
- User element numbers should be less than or equal to 99999. The element number does not have to be unique, but it is recommended.
- The sequence of the node numbering is similar to a regular boundary element. However, since the sampling surface element's only purpose is for displaying results in a post-processor, the direction of the order (with respect to the normal) is not important.

Example of Use -

```
SAMPling-points POINTS
   1001   1.0   2.0   3.0
   1002   1.0   3.0   3.0
   ⋮

SAMPling-surfaces SURFACE SS-TOP
   ELEMents-sampling-surf
   101   1001   1002   1003   1004   1005   1006   1007   1008
   102   1001   1002   1003   1009   1010   1011   1012   1013
   ⋮

SAMPling-surfaces SURFACE SS-BACK
   ELEMents-sampling-surf
   202   2001   2002   2003   2004   2005   2006
   ⋮
```

5.3.10 HISTORY POINT DEFINITION

HISTory-points NODE1 NODE2 . . . NODEN

Status - OPTIONAL

Full Keyword - **`HISTory-points`**

Function - Identifies which points are to have their results output to the nodal history file called HISTORY.OUT.

Input Variables -

NODE1 (Integer) - REQUIRED

User node number of desired node

NODE2 . . . NODEN (Integer) - OPTIONAL

User node numbers of desired node

Additional Information -

This card can be repeated as often as required. The keyword HISTory-points should be at the beginning of each new input line.

If HISTory-points is input, then HISTory-quantities should be input in **CASE input to specify which result quantities should be output.

The coordinate of the node specified on this card must be defined previously in the main coordinate list, the line-element coordinate list or in the sampling point coordinate list.

Examples of Use -

```
POINts
  21   1.0   0.0   0.0
  22   1.0   0.0   0.5
  23   1.0   0.0   1.0
  24   1.0   0.0   1.5
  ⋮

SURFace
  ⋮

SAMPling-points
  101   5.0   0.0   3.0
  102   6.0   0.0   3.0
  103   7.0   0.0   3.0

HISTory-points   21   22   101   102   103
  ⋮
```

5.3.11 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 **GPBEST** , 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.

Volume cells are not valid in structural-acoustic and periodic acoustic problems.

Examples of Use -

1. Define three, 8-noded quadratic volume cells for two-dimensional analysis.

```
  VOLUme
  TYPE-volume QUAD
  CELLs
   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)
```

TYPE-volume ATYPE

Status - REQUIRED (if VOLUme is input)

Full Keyword - **`TYPE-volume`**

Function - This card defines the variation of field quantities over the volume cells of the current GMR. This variation is independent of the geometry variation of the cell.

Input Variables -

ATYPE (Alphanumeric) - REQUIRED

Allowable values are LINE or QUAD.

LINEAR - Linear shape functions

QUADRATIC - Quadratic shape functions

Additional Information -

Only one TYPE-volume card for cells is allowed per GMR.

If QUAD is used and there are cells present, with linear geometry, then linear variation of field variables will be assumed over 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-Gmr GMR1
  .
  .
  .
  VOLUme
  TYPE-volume QUAD
```

CELLs

Status - REQUIRED (if VOLUme 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 VOLUme 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.

K = 3, 4, 6, or 8 for 2-D and axisymmetry

K = 4, 5, 6, 7, 8, 9, 10, 11, 15, 16, 20 or 21 for 3-D

Additional Information -

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

2-D: Cell numbering must begin at the corner and be numbered consecutively in either direction.

3-D: 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 (i.e., 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.

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.

```
VOLUme
TYPE-volume QUAD
CELLs
   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
```

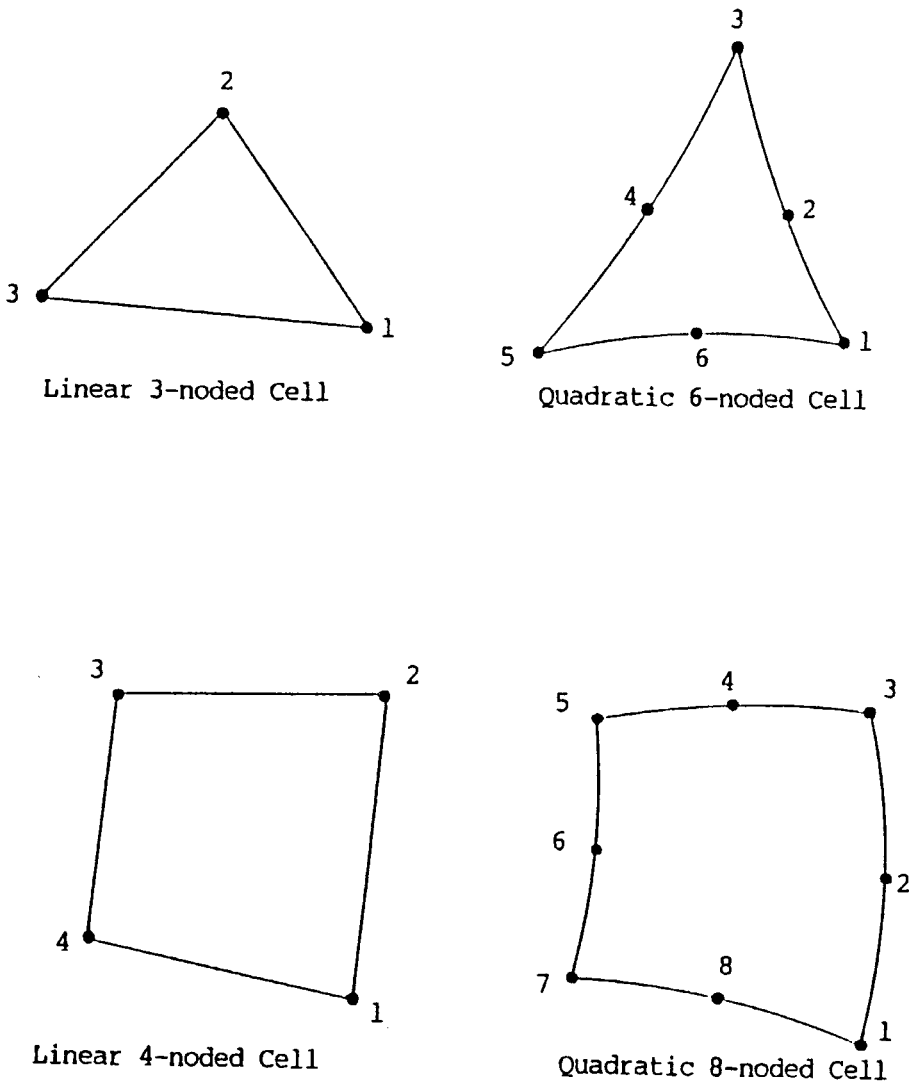


Figure for **GMR: Volume Cell Connectivity
Two-Dimensional Volume Cells

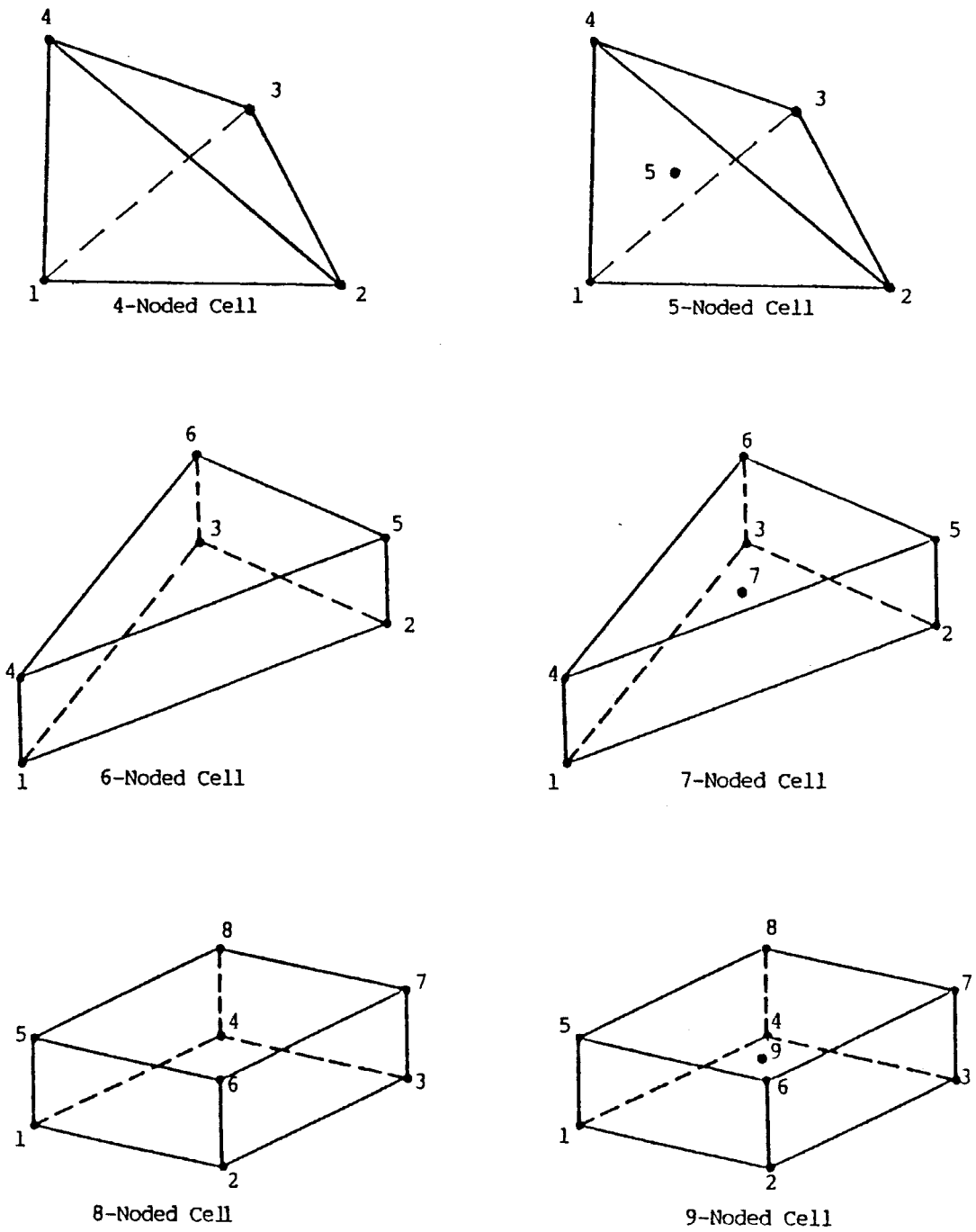


Figure for **GMR: Volume Cell Connectivity
Linear, Three-Dimensional Volume Cells

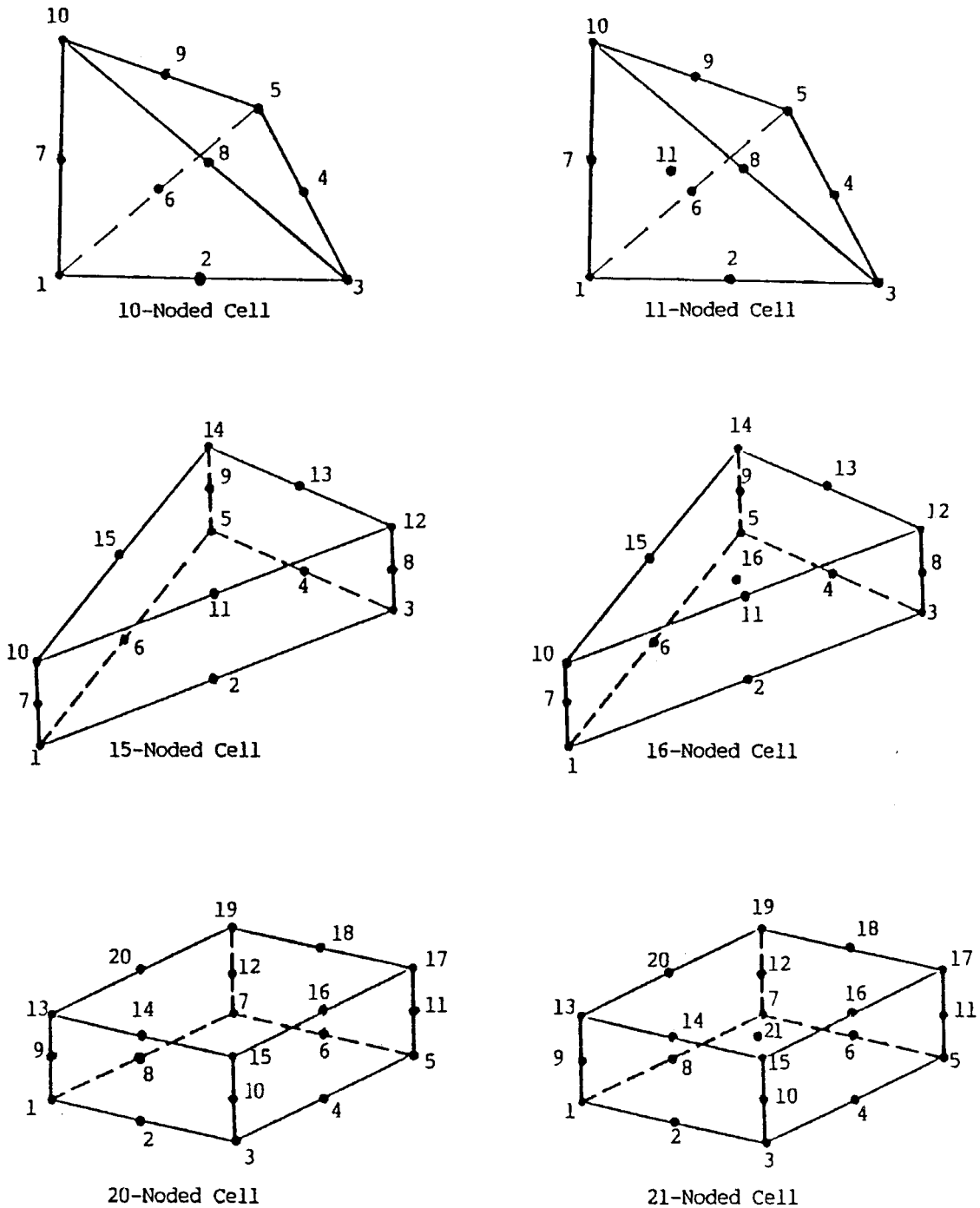


Figure for **GMR: Volume Cell Connectivity
Quadratic, Three-Dimensional Volume Cells

FULL

Status - REQUIRED (if VOLUme is input)

Full Keyword - **FULL**

Function - Identifies that the interior volume of 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 interior of the GMR must be completely filled by cells.

For highly accurate results, it is recommended that a GMR be completely filled with cells and that the FULL card option be used.

Examples of Use -

1. Specify that the GMR (GMR1) is completely filled with volume cells.

```
VOLUme
TYPE-volume QUAD
CELLs
  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
```

5.3.12 PROPERTY DEFINITION FOR FE GMR

PROPerty

Status - REQUIRED (for a **GMR with ID-Gmr having FINITE qualifier. This option is valid only when a FE GMR is defined for use in a structural-acoustics problem.)

Full Keyword - **`PROPerty`**

Function - Identifies a property set for various elements used in the current finite element GMR block.

Additional Information -

This card, followed by the attribute cards, defines the property set to be used by subsequent elements initiated by the ELEMents-surface and ELEMents-line cards. Henceforth, all elements in the current GMR would have the set attribute values until the next PROPerty card is defined, which would update the current attribute values. From that point onward, any further elements (using the ELEMents-surface and ELEMents-line cards) would have the updated property attribute values.

Refer to the following figure for cross-sectional property definitions:

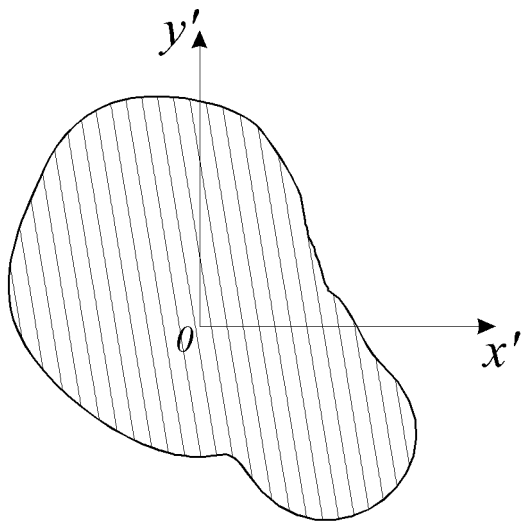


Figure for the PROPerty Card: Cross-Sectional Property Definitions

a) EMODULUS value

Status - REQUIRED

Full Keyword - **`EMODULUS`**

Function - Defines the value of Young's modulus.

Input Variables -

Value (Real) - Required

Additional Information - None

b) POISSON value

Status - REQUIRED

Full Keyword - **`POISSON`**

Function - Defines the value of Poisson's Ratio

Input Variables -

Value (Real) - Required

Additional Information - None

c) DENSITY value

STATUS - REQUIRED

Full Keyword - **`DENSITY`**

Function - Defines the value of the mass density

Input Variables -

Value (Real) - Required

Additional Information - None

d) AREA value

Status - REQUIRED

Full Keyword - **AREA**

Function - Defines the value of the cross-sectional area for frame cross-section.
Input Variables -

Value (Real) - Required

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis.

The equation for AREA is given by:

$$AREA = \int_A dx'dy'$$

e) XPAR value

Status - OPTIONAL

Full Keyword - **XPAREA**

Function - Defines the value of cross-sectional area effective in shear along the x'-axis.

Input Variables -

Value (Real) - Required

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis. If omitted, the value is taken as that given for the attribute AREA.

f) YPAR value

Status - REQUIRED

Full Keyword - **YPAREA**

Function - Defines the value of cross-sectional area effective in shear along the y'-axis.

Input Variables -

Value (Real) - Optional

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis. If omitted, the value is taken as that given for the attribute AREA.

g) IXPR value

Status - REQUIRED

Full Keyword - **IXPRIME**

Function - Defines the moment of inertia about x'-axis (usually the major axis depending on the orientation).

Input Variables -

Value (Real) - Required

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis.

The equation for IXPRIME is given by:

$$IXPRIME = I_{x'} = \int_A \left[(y')^2\right] dx'dy'$$

h) IYPR value

Status - REQUIRED

Full Keyword - **IYPRIME**

Function - Defines the moment of inertia about y'-axis (usually the major axis, depending on the beam orientation).

Input Variables -

Value (Real) - Required

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis.

The equation for IYPRIME is given by:

$$IYPRIME = I_{y'} = \int_A \left[(x')^2\right] dx'dy'$$

i) TORSIONAL value

Status - REQUIRED

Full Keyword - **TORSIONAL**

Function - Defines the moment of torsional inertia of the frame.

Input Variables -

Value (Real) - Required

Additional Information -

This attribute value is used only for 3-noded frame-type elements. If provided for shell-type elements, this value is not used in the analysis. If omitted, the value is taken as the sum of values given for attributes IXPR and IYPR, which is correct for a circular cross-section.

For noncircular cross-sections, TORS must be calculated or looked up in tables. For example, see Roark and Young [126].

Examples of Use -

1. Define property values of frames shown in the Figure.

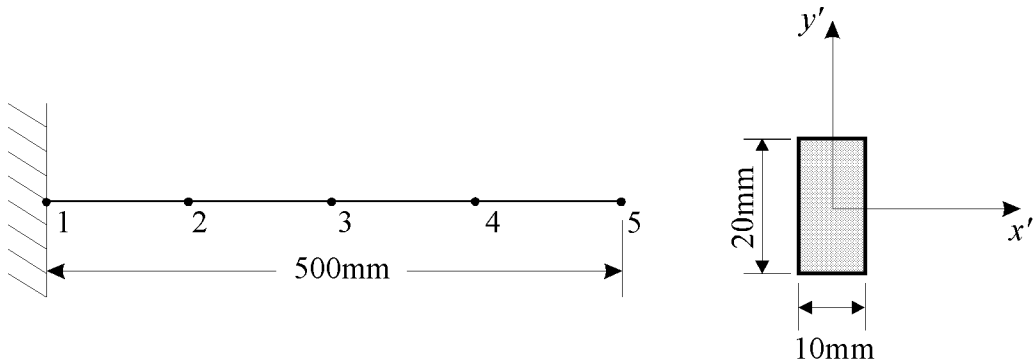


Figure for the FE PROP Card: Frame Elements

```
**GMR
  ID-Gmr  GMR1  FINI
  MATErial  STRUCTURE
  MASS LUMPED
  POINts
   1     0.0    0.0    0.0     $ mm
   2   125.0    0.0    0.0
   3   250.0    0.0    0.0
   4   375.0    0.0    0.0
   5   500.0    0.0    0.0
  PROPerty
  EMODULUS   2.07E+5      $ N/(mm^2)
  POISSON   0.30
  DENSITY   7.854E-9     $ (N s^2)/(mm^4)
  AREA   200.0        $ mm^2
  IXPR   6.667E+03    $ mm^4
  IYPR   1.667E+03    $ mm^4
  TORSIONAL   4.580E+03    $ mm^4
  ELEMents-surface
    1    1   2   3
    2    3   4   5
```

2. Use multiple properties for various elements of an FE GMR:

```
**GMR
  ID-Gmr  GMR1  FINI
  MATErial  STRUCTURE
  MASS CONSISTENT
  OVDAmping   0.03
  THICkness   0.04
  POINts
  .....
  .....
  PROPerty
  EMODULUS   6.82E+07
  POISSON   0.30
  DENSITY   0.80E+01
  AREA   0.30
  IXPR   0.004
  IYPR   0.014
  ELEMents-surface
    1    1   2   3   7   11   10   9   6
    2    3   4   5   8   13   12   11  7
    ⋮
    60   63  64  65
    61   71  72  73
  PROPerty
  EMODULUS   6.82E+07
  POISSON   0.30
  DENSITY   0.80E+01
  AREA   0.45
  IXPR   0.021
  IYPR   0.005
  ELEMents-surface
    49   53   48   45
    59   53   62   63
```

Note that in the above example, once the definition of frame-type elements starts (within the GMR), the shell-type elements (8-noded or 6-noded) can no longer be defined. This has been discussed in the section dealing with the ELEMents-surface and ELEMents-line cards.

MASS ATYPE

Status - OPTIONAL (applicable only for structural-acoustics problems).

Full Keyword - **MASS**

Function - Identifies the formulation to be used for the structural mass matrix.

Input Variables -

ATYPE (alpha) - Required.

Available value is either LUMPED or CONSISTENT.

Additional Information - None.

Examples of use -

1. Use LUMPED mass matrix:

```
**GMR
ID-Gmr  GMR1  FINITE
MATErial  STRUCTURE
MASS LUMPED
POINts
  ........
  ........
ELEMents-surface
  ........
  ........
```

OVDAmping VALUE

Status - OPTIONAL (applicable only for structural-acoustics problems).

Full Keyword - OVDAmping

Function - Defines the overall structural damping ratio for the problem.

Input Variables -

VALUE (real) - Required.

This value defines the overall material damping coefficient of the FE GMR.

Additional Information - None.

Examples of use -

1. Define 3% damping:

```
**GMR
ID-Gmr  GMR1  FINITE
MATErial STRUCTURE
MASS LUMPED
OVDAmping   0.03
POINts
  ........
  ........
ELEMents-surface
  ........
  ........
```

ECCEntricity
N1 V1

Status - Optional (applicable only for structural-acoustics problems)

Full Keyword - **`ECCEntricity`**

Function - Define nodal eccentricity for frame elements acting as stiffeners. Positive eccentrictiy value pushes the frame centerline along the nodal normal (it is assumed that all elements connected to the node have the same sense of the normal).

Input Variables -

N1 (Integer) - REQUIRED

This defines the node number (for a frame element) where the eccentricity is to be declared.

V1 (Real) - REQUIRED

This defines the value of the eccentricity at the node N1.

Additional Information - None

Examples of Use -

1. Define eccentricities at nodes 12 and 71.

```
**GMR
  ID-Gmr  GMR1   FINITE
  MATErial   STRUCTURE
  MASS LUMPED
  POINts
  .....
  .....
  PROPerty
  .....
  .....
  ELEMents-surface
  .....
  .....
  ECCEntricity
    12   0.30
    71   0.15
```

5.4 DEFINITION OF GMR COMPATIBILITY

SECTION	KEYWORD	PURPOSE	PAGE
5.4.1	Interface Definition Input Card		5.4.7
	**INTERFACE	start of interface compatibility condition	5.4.7
	ID-Interface	Name of interface set	5.4.8
	GAP-interface	obsolete - see TOLErance card	5.4.9
	TOLErance	set tolerance used for matching node pairs	5.4.10
	SANDwich	fully coupled structural interface between two acoustic regions	5.4.13
5.4.2	Definition of Interface Surface 1		5.4.15
	GMR-1	name of first GMR	5.4.15
	SURFace-gmr-1	surface on first GMR	5.4.16
	CLINe-gmr-1	cooling line in first GMR	5.4.17
	ELEMents-gmr-1	element of surface	5.4.18
	POINts-gmr-1	nodal point of surface for gap definition	5.4.19
5.4.3	Definition of Interface Surface 2		5.4.20
	GMR-2	name of second GMR	5.4.20
	SURFace-gmr-2	surface on second GMR	5.4.21
	CLINe-gmr-2	cooling line in second GMR	5.4.22
	ELEMents-gmr-2	element of surface	5.4.23
	POINts-gmr-2	nodal point of surface for gap definition	5.4.24
5.4.4	Type of interface conditions		5.4.29
	BONDed	bonded interface connection	5.4.29
	SLIDing	sliding interface connection	5.4.31
	FIT-value	interference fit	5.4.33
	CONTact	nonlinear contact interface	5.4.37
	SPRIng-value	spring condition	5.4.40
	CONVection-value	thermal convection condition	5.4.43
	RESIstance-value	thermal resistance condition	5.4.43
	GENEralized-interface	generalized convection condition	5.4.44
	IMPEdance-value	impedance condition	5.4.46
	ADMIttance-value	admittance condition	5.4.46

Definition of GMR Compatibility

SECTION	KEYWORD	PURPOSE	PAGE
5.4.5	Additional contact interface input		5.4.47
	FRICtion	coefficient of static friction	5.4.47
	EQUIvalence	make equivalent the locations of node pairs	5.4.48
	OFFSet	offset the locations of node pairs	5.4.51
	O-VAlue	offset values	5.4.55
5.4.6	Cyclic Symmetry interface definition		5.4.57
	CYCLic	cyclic symmetry interface definition	5.4.57
	ANGLe	angle for cyclic interface	5.4.60
	DIREction-cyclic	axis of rotation for cyclic interface	5.4.62

Modeling Interfaces Between GMRs

When a body is modeled as an assembly of several GMRs, suitable conditions must be specified in order to define the connections among the various regions. In the present version of **GPBEST** , compatibility is defined between the interface surfaces of each pair of contacting regions. Six types of compatibility are allowed:

1 - Bonded contact : Continuity of all displacement components is imposed across the interface. This is the default if no other interface type is specified.

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 - Interference Fit : Imposed displacement between interface elements.

4 - Nonlinear Contact : When the interface nodes are under compression, continuity of displacement is maintained in the normal direction and tangential tractions are proportional to the normal traction (in the direction resisting movement) where the constant proportionality is the coefficient of static friction. When the interface nodes are in tension, all tractions are set to zero and a gap will appear between the interface nodes.

5 - Spring, resistance/convection, or impedance contact : Spring or thermal resistance (or acoustic impedance) is imposed between regions.

6 - Cyclic symmetric contact : Symmetric elements within a cyclic symmetric part have symmetric deformation imposed on these elements.

Continuity of temperature or pressure, where applicable, is imposed across the interface in a similar manner.

In Version 5.0 of **GPBEST** , a single nodal point location may be a part of any number of GMRs. Thus, nodal points may be referenced in more than one interface definition set. However, a single location must have a unique node number in each GMR. In prior versions, a maximum of two GMRs were allowed to intersect at a point as illustrated in the figure following the **INTERFACE card.

In Version 5.2 of **GPBEST** , traction, flux, torque and load boundary conditions can be applied to elements that are contained in an interface definition. See the INTErface card in Section 5.3.3, ‘Identification of Boundary Condition Type.’

When working with interface conditions, it is important to distinguish between conditions that apply to the undeformed geometry and conditions that apply to the deformed geometry. For instance, the EQUIvalence and OFFSet cards apply only to the undeformed geometry. In other words, they describe what the geometry looks like before any deformation occurs. On the other hand, the FIT-value, SLIDing, and CONTact cards apply to the deformed geometry. They determine how the GMRs interact once deformation starts.

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

GMRs that are involved. Unless the TOLErance card or the CYCLic card is input, the corresponding nodes of the two GMRs must have identical coordinates.

Some examples of valid and invalid GMR interface definitions may assist the user. Figure 5.4.1 illustrates several valid interface definitions for problems involving two GMRs.

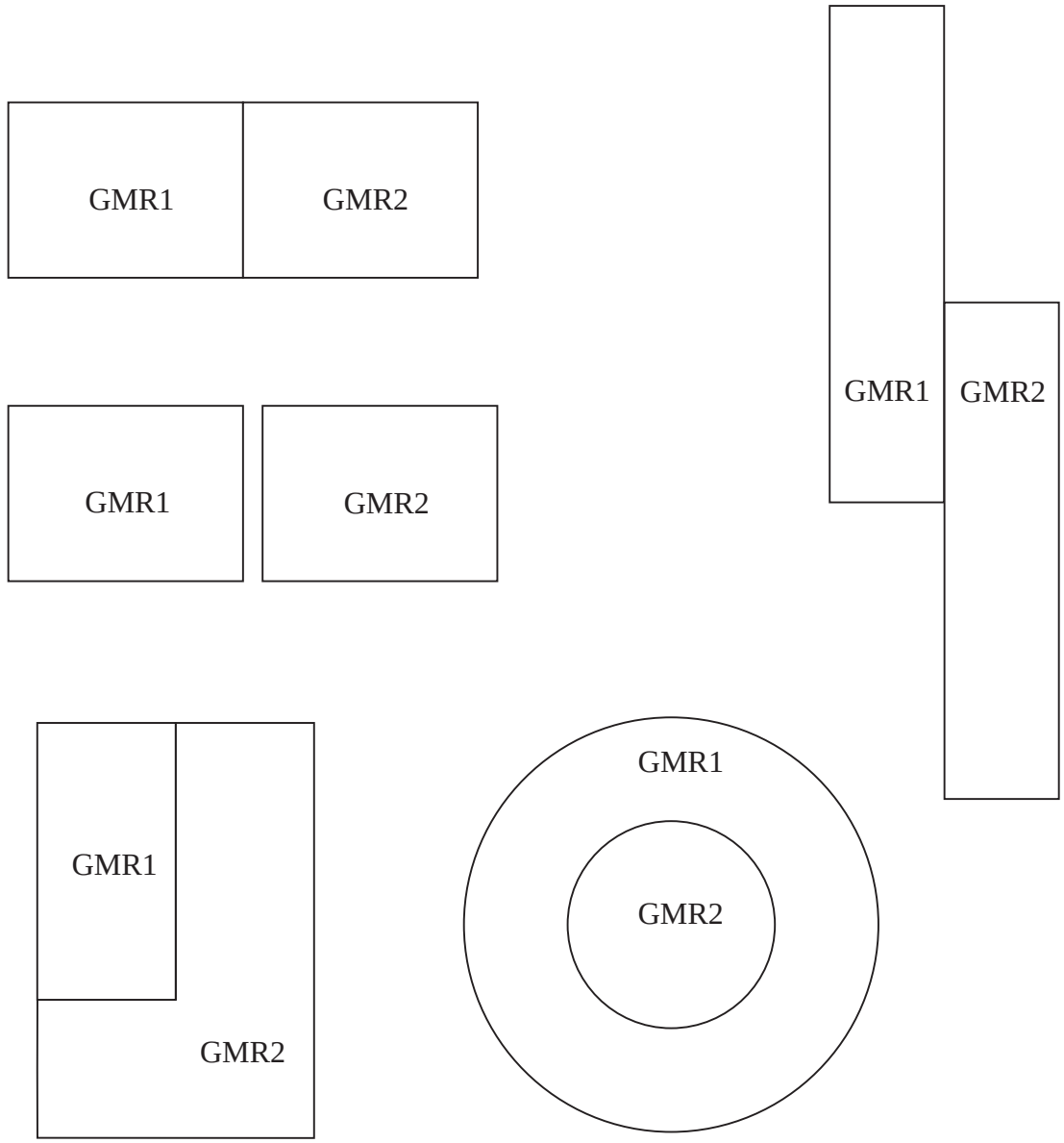


Figure 5.4.1 Valid Interfaces for Two-GMR Problems

Figure 5.4.2 illustrates several multi-GMR problems, with three or more GMRs. In these examples, each interface node is shared by only two GMRs. Therefore, **GPBEST** handles their connection exactly.

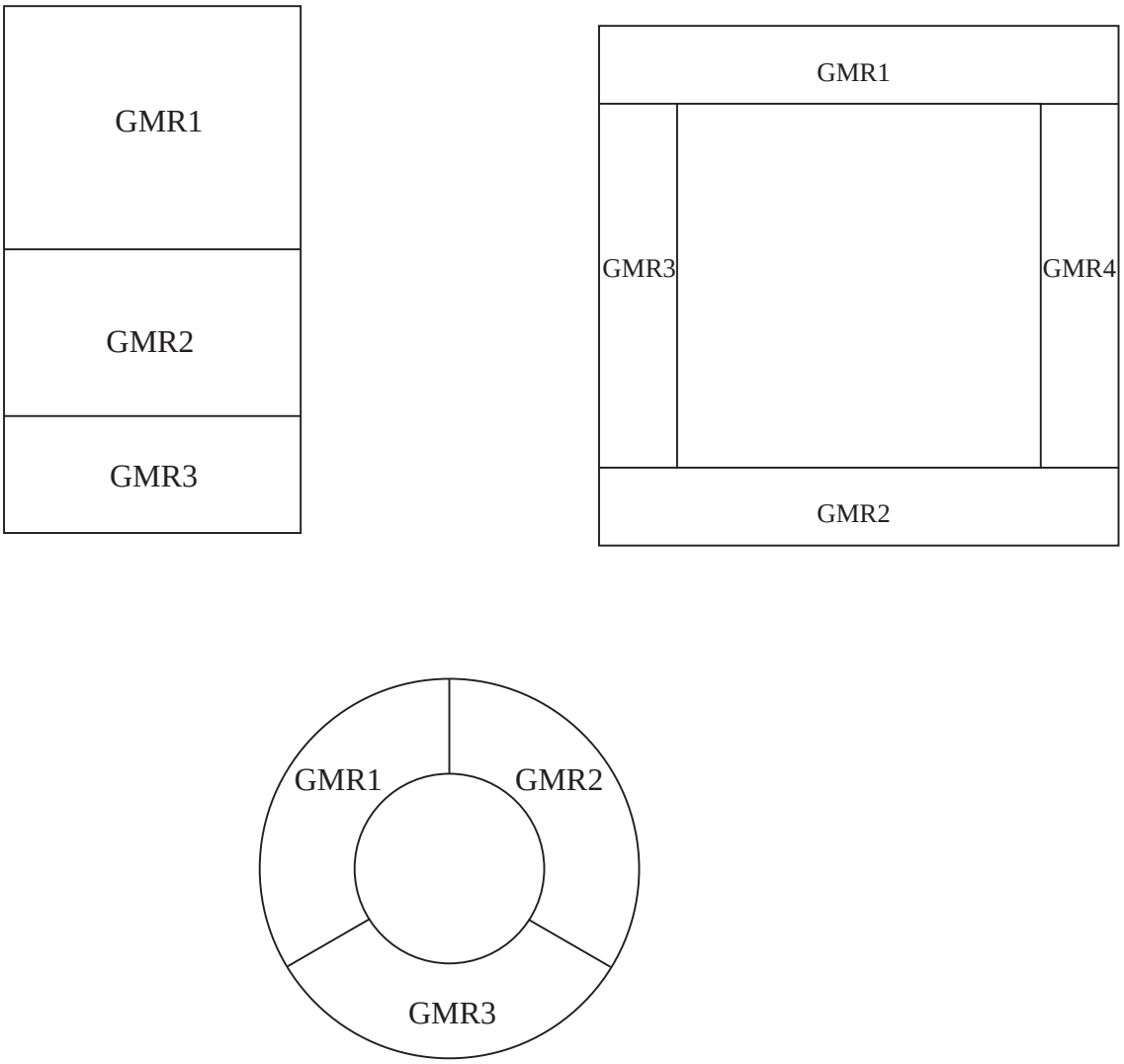


Figure 5.4.2 These GMR Connections are Handled Exactly

On the other hand, Figure 5.4.3 shows different multi-GMR problems. In these, more than two GMRs share a node. In this case, **GPBEST** handles the connecting nodes in an approximate manner.

Definition of GMR Compatibility

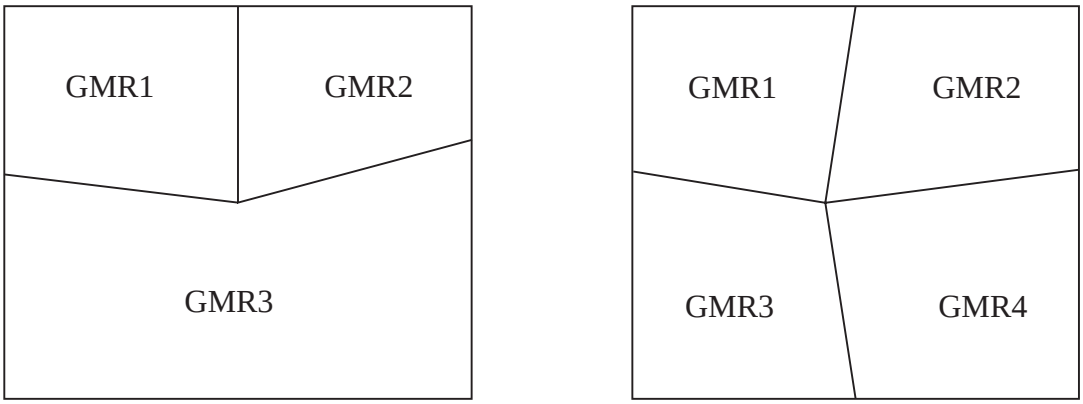
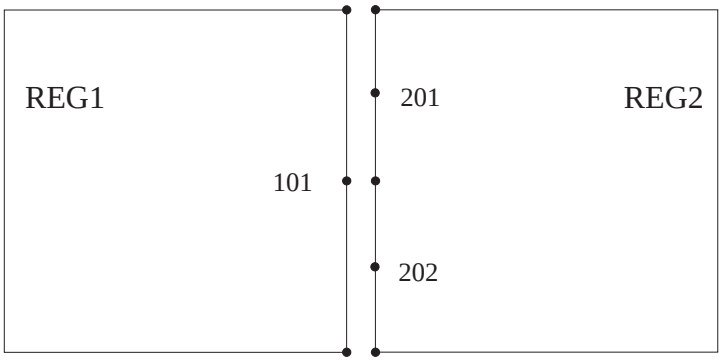


Figure 5.4.3 These Connections are Approximated
at the Node Where More Than Two GMRs Connect

Figure 5.4.4 illustrates an illegal interface in **GPBEST** . An interface cannot be used to ‘‘stitch’’ two non-similar GMR edges together. However, it should be noted that the PREBEST interface will allow a user to join a linear GMR with a quadratic GMR. In this case, PREBEST will output data that will be acceptable to **GPBEST** , employing linear interfaces only.



(Element Numbers Shown)

Figure 5.4.4 Illegal Interface Between Two GMRs

Acoustic problems have some special restrictions on interfaces involving finite element regions. Currently, **GPBEST** is limited to one boundary element GMR in the structural definition and one finite element GMR in the Indirect boundary element formulation. Therefore, there is only a need for the **INTERFACE card to define interfaces between multiple acoustic GMRs when using a Direct boundary element formulation is being used. While dealing with problems of fully-coupled structural-acoustics usind DBEM, at least one SANDwich interface must be defined. All the finite elements have to be accounted for in such SANDwich interface sections.

Throughout the rest of this section are figures illustrating the various types of GMR interfaces. The user is encouraged to browse these examples in order to become familiar with the wide variety of interface options available in **GPBEST** .

5.4.1 INTERFACE DEFINITION INPUT CARD

**INTERFACE

Status - OPTIONAL

Full Keyword - **INTERFACE**

Function - Indicates the beginning of an interface definition.

Input Variables - NONE

Additional Information -

A **INTERFACE 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.

Note that each of the two GMRs 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 GMRs (default is perfectly bonded connection).

```
**INTERFACE
  GMR-1  REG1
     SURFace-gmr-1  TOP
     ELEMents-gmr-1   3  4  5
  GMR-2  REG2
     SURFace-gmr-2  BOTTOM
     ELEMents-gmr-2  103  104  105
```

ID-Interface NAME

Status - OPTIONAL

Full Keyword - **`ID-Interface`**

Function - Defines the identifier for the current interface.

Input Variables -

NAME (Alphanumeric) - REQUIRED

User-specified name for the current data set.

Additional Information -

If the user does not include this card, then an id NAME 'INTERFAC' will be assigned by **GPBEST** .

It is recommended (but not necessary) that the NAME be unique compared to all other interface data set names defined in the problem.

The NAME must be eight or less alphanumeric characters. Blank characters occurring within the NAME are not permitted.

Examples of Use -

1. Define an interface set.

```
**INTERFACE
  ID-Interface  DISP1
  SURFace-gmr-1  SURF1
  ELEMents-bc  104
```

2. Define a bolt and sleeve interface.

```
**INTERFACE
  ID-Iterface BOLTSLEV
  GMR-1 BOLT
  SURFace-gmr-1 SURF7
  ELEMent-gmr-1 106 107
  GMR-2 SLEEVE
  SURFace-gmr-2 SURF20
  ELEMent-gmr-2 206 207
```

GAP-interface

A new keyword TOLErance has been added to GPBEST. (See next card for details). The function of the TOLErance card is similar to the function of the GAP-interface card, however, the TOLErance card is more general, and therefore,

‘‘GAP-interface EXPLICIT’’.

NOTE TO USERS: See the keyword OFFSet for specifying a gap which has not been physically modeled between the two surfaces of an interface.

TOLErance TVAL

Status - OPTIONAL

Full Keyword - **`TOLErance`**

Function - Specify a user-defined tolerance used in matching interface node pairs of this interface set.

Input Variables -

TVAL (REAL) REQUIRED

User defined interface tolerance

Additional Information -

The node matching algorithm in GPBEST first determines the closet two nodes (defined in the interface set) between the two regions and defines them as the first node pair. The algorithm then looks for the closest two remaining no

It may also be necessary to define a tolerance in a contact analysis when an actual gap is modeled across a contact interface. It is also worth noting that if the physical gap is quite large, the node-matching algorithm may have

Example of interface sets containing a TOLErance card can be found in the example problems: **`elas810.dat`**, **`elas812.dat`**, and **`elas813.dat`**.

Examples of Use -

1. Define an initial 0.1 gap between two GMRs. The GMR nodes are actually modeled with the specified gap in the **GMR blocks; the interface nodes are NOT coincident. Figure (a) shows the two GMRs separated by a 0.1 gap.

   To aid the user, Figures (b) and (c) illustrate what the user can expect to view on the screen while in a post-processor such as BESTVIEW, PATRAN, etc.

```
**INTERFACE
    TOLErance  0.11
    GMR-1 REG1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1  101  102
    GMR-2 REG2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2  201  202
    CONTact
```

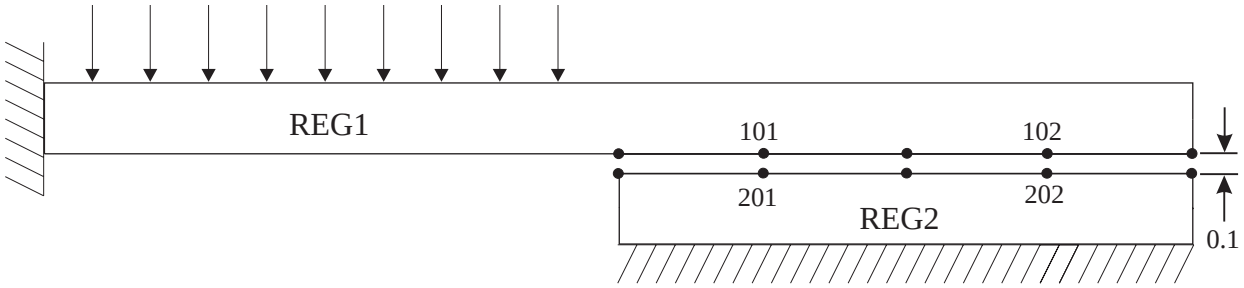
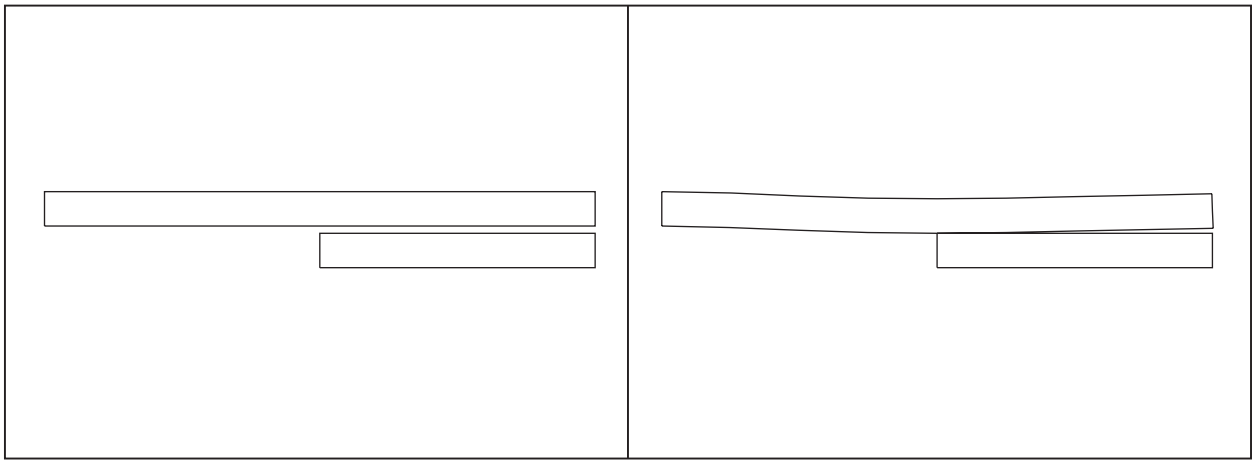


Figure for TOLErance Card: (a) Interface Modeled using the TOLErance Card



(b) Undeformed Geometry (c) Deformed Geometry After Running **GPBEST**

Figure for TOLErance Card: Results Viewed in a Post-Processor

2. In modeling the Hertz contact problem, the nodes from the two GMRS are not coincident at the interface. Therefore, the TOLErance option is employed. Four elements and nine nodes are used to model the interface where contact occurs.

```
**INTERFACE
  TOLErance  0.35
  GMR-1 GMR1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1   1   2   3   4
    POINts-gmr-1     1   2   3   4   5   6   7   8   9
  GMR-2 GMR2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2  69  68  67  66
    POINts-gmr-2    81  82  83  84  85  86  87  88  89
  CONTact
```

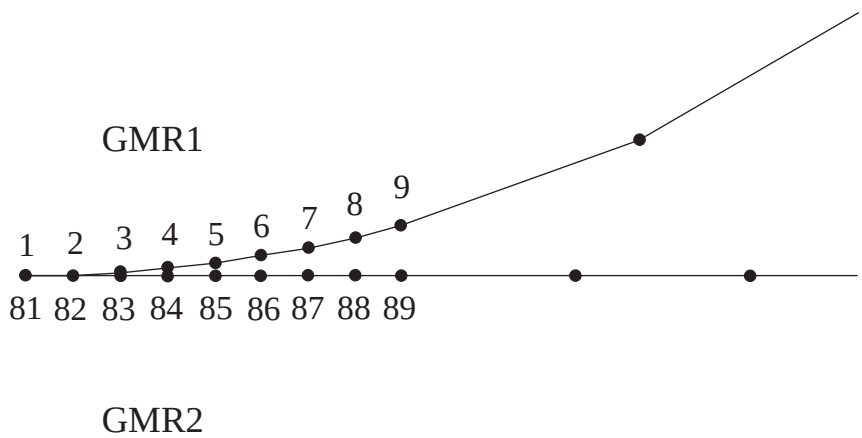


Figure for TOLErance Card: (d) TOLErance Option (Node Numbers Shown)

SANDwich

Status - OPTIONAL

Full Keyword - **SANDwich**

Function - Identifies two acoustic regions containing a finite element shell structure between them. The shell structure is ‘‘sandwiched’’ between the two acoustic regions.

Input Variables - NONE

Additional Information -

This keyword is valid only in the context of a fully-coupled multi-GMR structural acoustics problem using the direct boundary element method.

Note that all structural shell elements must be in contact with the fluid regions.

A SANDwich interface always behaves as a BONDed type interface. Currently, this is the only interface type available when there is a structure between two acoustic regions.

Examples of Use -

1. Top surface of a cube vibrating in air with the other five surfaces rigid. Since the five walls are rigid, there does not need to be any interface defined between IZONE (interior) and EZONE (exterior) GMRs. Figure (a) illustrates the cube modeled with four elements on each face. Element numbers are 1 through 24.

```
**INTERFACE
    SANDwich
    GMR-1 IZONE   $ Interior
      SURFace-gmr-1 IZONE
      ELEMents-gmr-1  1  2  3  4
    GMR-2 EZONE   $ Exterior
      SURFace-gmr-2 EZONE
      ELEMents-gmr-2  1  2  3  4
```

Definition of GMR Compatibility

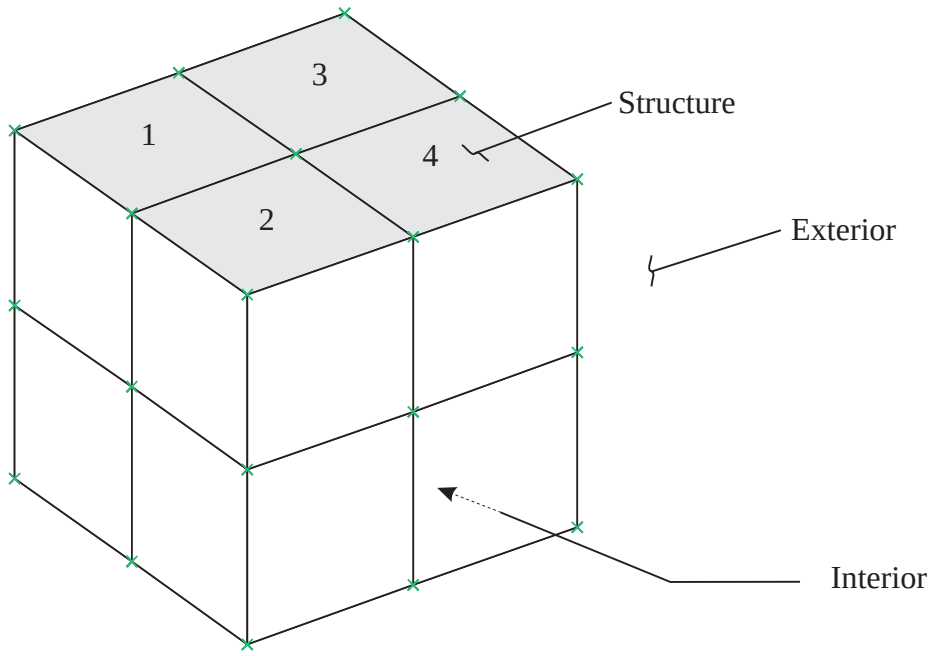


Figure for SANDwich Card: (a) Interior and Exterior GMRs

NOTE: Since the walls of the container are rigid, separate **INTERFACE conditions do not need to be defined for them. However, the structure between the interior and exterior GMRs must be defined using an **INTERFACE.

5.4.2 DEFINITION OF INTERFACE SURFACE 1

GMR-1 IDGMR

Status - REQUIRED

Full Keyword - **GMR-1**

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 (i.e., NAME on ID-Gmr 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 set must be defined for each case.

Examples of Use -

1. Identifies the first GMR, i.e. GMR1, of which the interface surface is a part.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 101   102   103   104
```

SURFace-gmr-1 IDSUR

Status - REQUIRED

Full Keyword - **`SURFace-gmr-1`**

Function - Identifies the surface within the (first) selected GMR-1 which embodies the interface surface (i.e., name on SURFace 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 set must be defined for each surface involved.

The SURFace-gmr-1 card may conclude the required input for a interface definition. If the SURFace-gmr-1 card is not followed by a ELEMents-gmr-1 card, then GPBEST will apply the interface compatibility condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the interface surface, i.e. SURF1, as part of the first GMR-1.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 109   110
  GMR-2 GMR2
    SURFace-gmr-2  SURF2
```

CLINe-gmr-1

Status - REQUIRED

Full Keyword - **`CLINe-gmr-1`** (Cooling Line)

Function - Identifies a cooling line within the (first) selected GMR-1 which is to be connected to another cooling line.

Input Variables -

NONE

Additional Information -

If the interface compatibility condition is to be applied to more than one cooling line, then a separate **INTERFACE set must be defined for each cooling line involved.

Examples of Use -

1. Example of an interface involving cooling lines.

```
**INTERFACE
  GMR-1 GMR1
    CLINe-gmr-1
    ELEMents-gmr-1  109
    POINts-gmr-1    101
  GMR-2 GMR2
    CLINe-gmr-2
    ELEMents-gmr-2  209
    POINts-gmr-2    201
```

ELEMents-gmr-1 EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - **`ELEMents-gmr-1`**

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 form 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 ELEMents-gmr-1.

If the ELEMents-gmr-1 card is specified, GPBEST will apply the interface compatibility condition only to the elements specified on this list.

In the present version of **GPBEST** , interface compatibility cannot 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-gmr-1 card.

```
**INTERFACE
  GMR-1  GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 120   121   122
  GMR-2 GMR2
```

POINts-gmr-1 P1 P2 ... PN

Status - OPTIONAL (required only when node matching algorithm fails, or ‘‘OFFSet VARIABLE’’ is input, or for a cooling line interface)

Full Keyword - **POINts-gmr-1**

Function - Defines the point order used in matching corresponding nodes on the two surfaces (or cooling line) of the interface.

Input Variables -

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

Additional Information -

The list of points for this surface (or cooling line) will be matched consecutively to the list of points for the other surface (or cooling line) in the interface definition.

Elements of each region do not have to be listed in order.

This card may be repeated as often as required. Each card must begin with the keyword POINts-gmr-1.

This card is not valid for structural-acoustics problems.

Examples of Use -

1. Define the one-to-one correspondence between interfacial nodes, which are initially separated by an explicitly modeled gap.

```
**INTERFACE
  TOLErance  1.1
  GMR-1 REG1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 1001
  POINts-gmr-1  97  58  33  24  82  67  66  43
  GMR-2 REG2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 2003
  POINts-gmr-2  2057  2063  2007  2006  2035  1068  1072  2099
```

2. Example of an interface involving cooling lines.

```
**INTERFACE
  GMR-1 GMR1
    CLINe-gmr-1
    ELEMents-gmr-1  109
    POINts-gmr-1    101
  GMR-2 GMR2
    CLINe-gmr-2
    ELEMents-gmr-2  209
    POINts-gmr-2    201
```

5.4.3 DEFINITION OF INTERFACE SURFACE 2

GMR-2 IDGMR

Status - REQUIRED

Full Keyword - **GMR-2**

Function - Identifies the second GMR-2 for which the interface surface is to be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR-2 as input during the geometry definition (i.e., NAME on ID-Gmr 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 set must be defined for each case.

Examples of Use -

1. Identifies the second GMR-2, i.e. GMR2, of which the GMR surface is a part.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 101   102
  GMR-2 GMR2
    SURFace-gmr-2  SURF2
    ELEMents-gmr-2 201   202
```

SURFace-gmr-2 IDSUR

Status - REQUIRED

Full Keyword - **`SURFace-gmr-2`**

Function - Identifies the surface within the (second) selected GMR-2 which embodies the interface surface (i.e., name on SURFace 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 set must be defined for each surface involved.

The SURFace-gmr-2 card may conclude the required input for a interface definition. If the SURFace-gmr-2 card is not followed by a ELEMents-gmr-2 card, then GPBEST will apply the interface compatibility condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the interface, i.e. SURF2, as part of the second GMR-2.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 101   102
  GMR-2 GMR2
    SURFace-gmr-2  SURF2
    ELEMents-gmr-2 201   202
```

CLINe-gmr-2

Status - REQUIRED

Full Keyword - **`CLINe-gmr-2`** (Cooling Line)

Function - Identifies a cooling line within the (second) selected GMR-2 which is to be connected to another cooling line.

Input Variables -

NONE

Additional Information -

If the interface compatibility condition is to be applied to more than one cooling line, then a separate **INTERFACE set must be defined for each cooling line involved.

Examples of Use -

1. Example of an interface involving cooling lines.

```
**INTERFACE
  GMR-1 GMR1
    CLINe-gmr-1
    ELEMents-gmr-1  101
    POINts-gmr-1    105
  GMR-2 GMR2
    CLINe-gmr-2
    ELEMents-gmr-2  201
    POINts-gmr-2    205
```

ELEMents-gmr-2 EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - **`ELEMents-gmr-2`**

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 ELEMents-gmr-2 card is specified, GPBEST will apply the interface compatibility condition only to the elements specified on this list.

In the present version of **GPBEST** , interface compatibility cannot be specified at individual nodes.

Examples of Use -

1. Specifies three elements, 210, 211 and 212, for interfacial compatibility on the surface identified by the preceding SURFace-gmr-2 card.

```
**INTERFACE
  GMR-1  GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 101   102   103
  GMR-2  GMR2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2 210   211   212
```

POINts-gmr-2 P1 P2 ... PN

Status - OPTIONAL (required only when node matching algorithm fails or ‘‘OFFSet VARIABLE’’ is input, or for a cooling line interface)

Full Keyword - **POINts-gmr-2**

Function - Defines the point order used in matching corresponding nodes on the two surfaces (or cooling lines) of the interface.

Input Variables -

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

Additional Information -

The list of points for this surface (or cooling line) will be matched consecutively to the list of points for the other surface (or cooling line) in the interface definition.

This card may be repeated as often as required. Each card must begin with the keyword POINts-gmr-2.

This card is not valid for structural-acoustics problems.

Examples of Use -

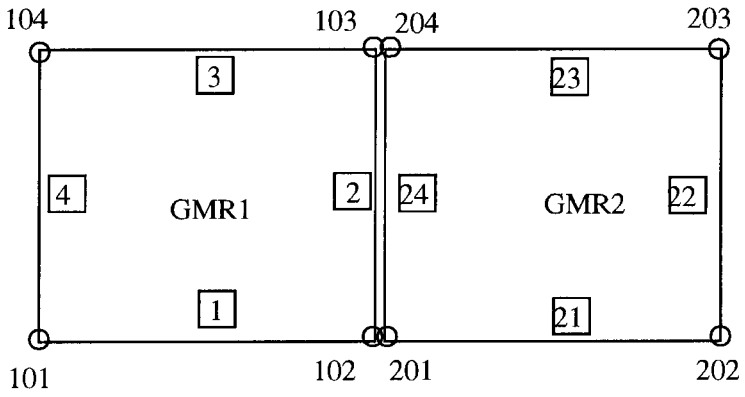
1. Define the one-to-one correspondence between interfacial nodes which are initially separated by an explicitly modeled gap.

```
**INTERFACE
  TOLErance   1.1
  GMR-1 REG1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 1001
  POINts-gmr-1  97  58  33  24  82  67  66  43
  GMR-2 REG2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 2003
  POINts-gmr-2  2057  2063  2007  2006  2035  1068  1072  2099
```

2. Example of an interface involving cooling lines.

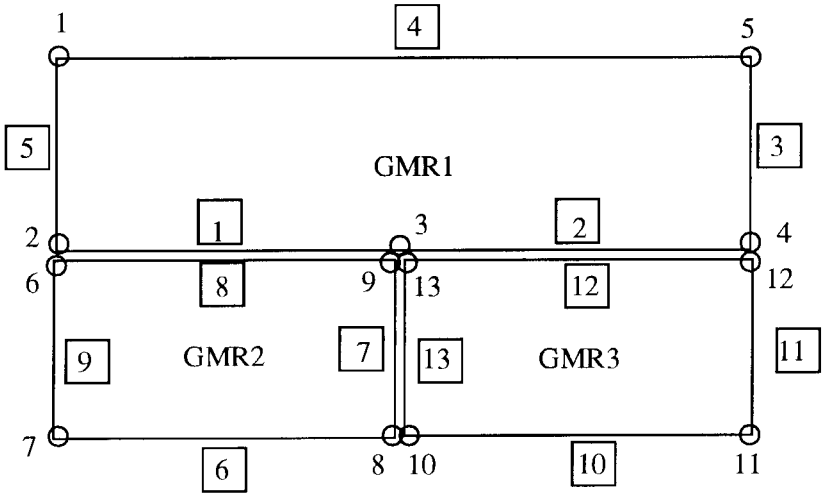
```
**INTERFACE
  GMR-1 GMR1
    CLINe-gmr-1
    ELEMents-gmr-1  101
    POINts-gmr-1    105
  GMR-2 GMR2
    CLINe-gmr-2
    ELEMents-gmr-2  201
    POINts-gmr-2    205
```

SOME EXAMPLES OF THE **INTERFACE CARD



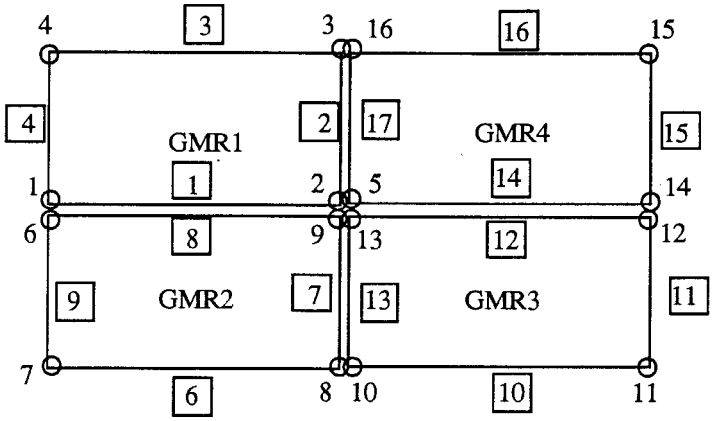

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 2
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 24
```

SOME EXAMPLES OF THE **INTERFACE CARD
CONTINUED



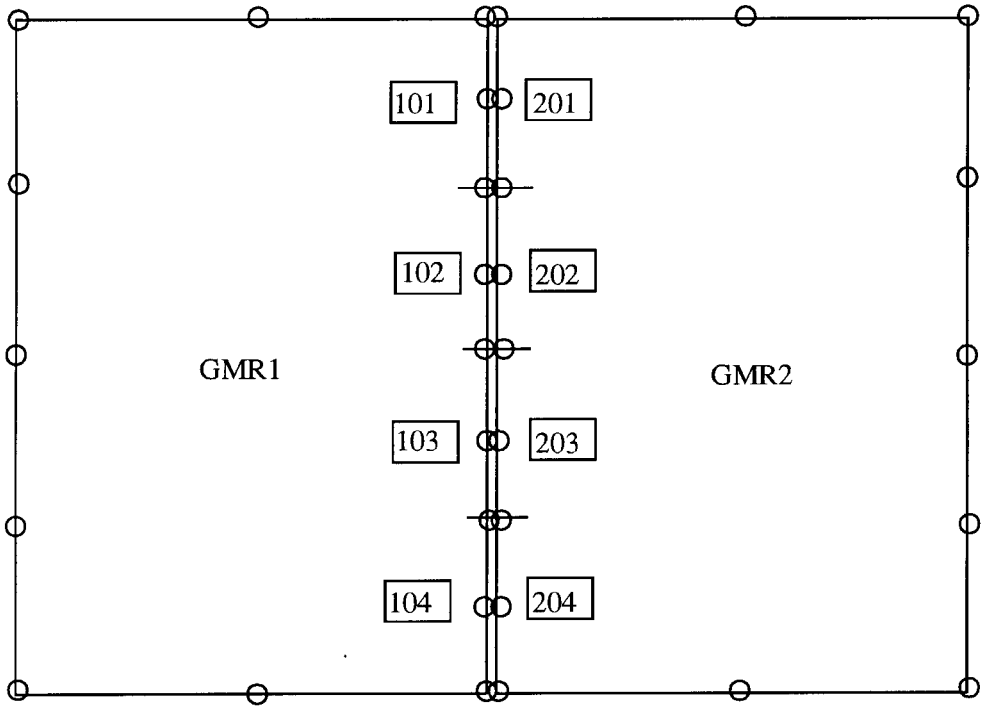

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 1
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 8

**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 2
  GMR-2 GMR3
  SURFace-gmr-2 SURF3
  ELEMents-gmr-2 12

**INTERFACE
  GMR-1 GMR2
  SURFace-gmr-1 SURF2
  ELEMents-gmr-1 7
  GMR-2 GMR3
  SURFace-gmr-2 SURF3
  ELEMents-gmr-2 13
```

SOME EXAMPLES OF THE **INTERFACE CARD
CONTINUED




```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 1
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 8

**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 2
  GMR-2 GMR4
  SURFace-gmr-2 SURF4
  ELEMents-gmr-2 17

**INTERFACE
  GMR-1 GMR2
  SURFace-gmr-1 SURF2
  ELEMents-gmr-1 7
  GMR-2 GMR3
  SURFace-gmr-2 SURF3
  ELEMents-gmr-2 13

**INTERFACE
  GMR-1 GMR3
  SURFace-gmr-1 SURF3
  ELEMents-gmr-1 12
  GMR-2 GMR4
  SURFace-gmr-2 SURF4
  ELEMents-gmr-2 14
```

SOME EXAMPLES OF THE **INTERFACE CARD
CONTINUED




```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1  101  102  103  104
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2  201  202  203  204
```

NOTE: Multi-GMR Connections Shown In Previous Two Pages May Also Involve Multiple Elements

5.4.4 TYPE OF INTERFACE CONDITION

BONDed

Status - OPTIONAL

Full Keyword - **BONDed**

Function - Identifies a fully bonded interface.

Input Variables - NONE

Additional Information -

When this card is input, continuity of all primary response variables (e.g., displacement, temperature) is imposed across the interface.

This is the default condition when the type of interface is not explicitly defined.

The BONDed keyword cannot be used in an **INTERFACE definition which also includes a GENEralized-interface CONV, RESIstance-value, SLIDing, FIT-value, CONTact or SPRIng-value keyword.

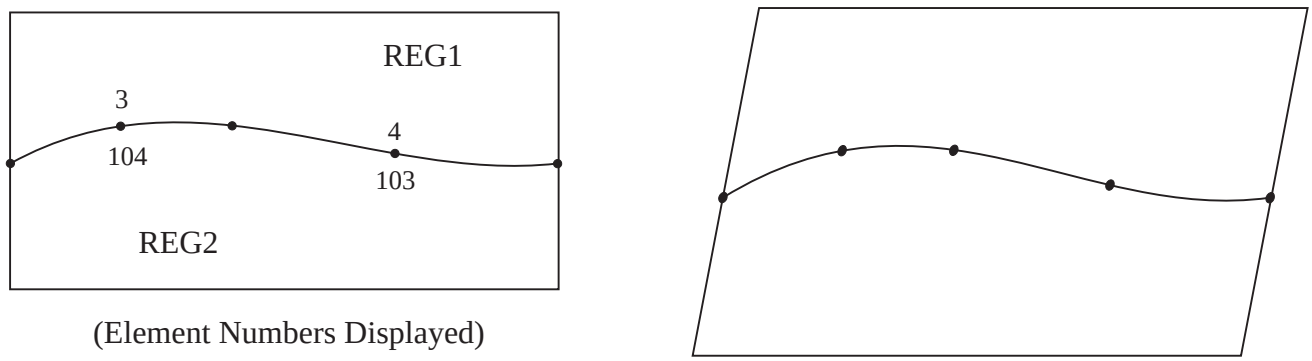
SANDwich interfaces can only behave as BONDed interfaces.

Definition of GMR Compatibility

Examples of Use -

1. Defines a perfectly bonded interface of two boundary elements. Figure (a) shows that the two GMRs are modeled with coincident nodes at the interface. Figure (b) displays the deformed geometry where the interface remains bonded.

```
**INTERFACE
  GMR-1  REG1
    ELEMents-gmr-1   3   4
  GMR-2  REG2
    ELEMents-gmr-2  103 104
  BONDed     $ This card is optional. If no other type of
             $ interface is specified, bonded is assumed.
```



(a) (b)

Figures for BONDed Card: (a) BONDed Interface, (b) Deformed Geometry

SLIDing

Status - OPTIONAL

Full Keyword - **`SLIDing`**

Function - Identifies a sliding interface.

Input Variables - NONE

Additional Information -

When this card is input, only normal displacement compatibility is imposed across the interface. The two GMRs are free to move in the plane tangent to the interface. However, the surfaces remain in contact even under tension. This freedom may require the specification of additional boundary conditions to restrain rigid body motion.

The SLIDing keyword cannot be used in an **INTERFACE definition which also includes a CONTact, BONDed or SPRIng-value keyword. Note, however, that a SLIDing condition on a CONTact interface can be accomplished by setting the contact friction to zero (see CONTact and FRICtion statements in this section), and a SLIDing condition with a SPRIng-value connection can be accomplished by setting the tangential spring constant to zero (see SPRIng-value statement in this section).

It is important to realize that the SLIDing card applies only to the deformed geometry. It determines how the GMRs interact once deformation starts. The SLIDing card does not apply to the undeformed geometry, unlike the EQUIvalence and OFFSet cards.

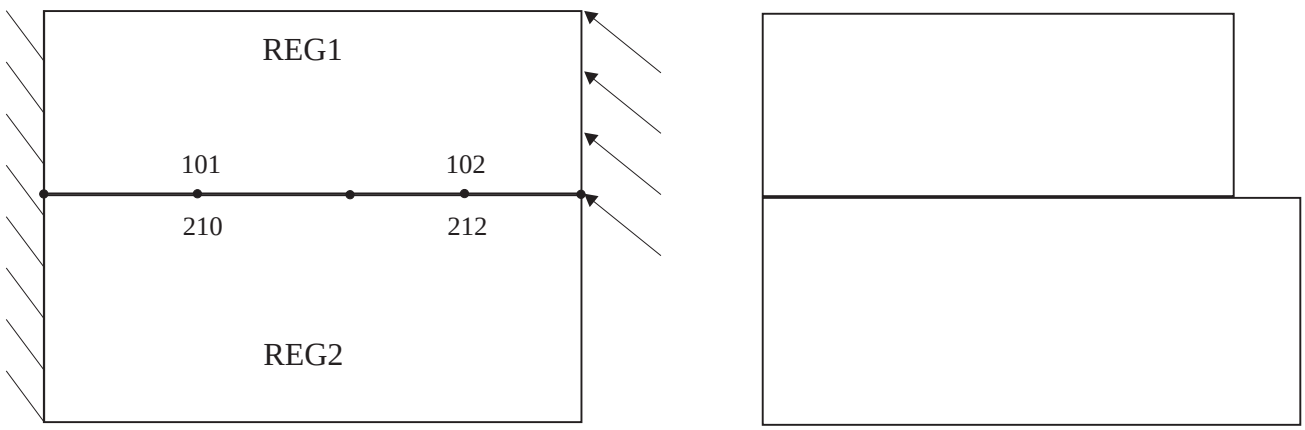
SLIDing uses a local normal coordinate system.

Definition of GMR Compatibility

Examples of Use -

1. Defines a sliding interface of two boundary elements. Figure (a) illustrates the SLIDing interface. The nodes of the two GMRs are coincident at the interface. Figure (b) displays the deformed geometry. In particular, notice that although the load in Figure (a) would tend to separate the two GMRs, the SLIDing interface keeps them together. However, lateral motion is permitted as shown in Figure (b).

```
**INTERFACE
  GMR-1  REG1
    ELEMents-gmr-1   101   102
  GMR-2 REG2
    ELEMents-gmr-2   210   212
  SLIDing
```



(a) (b)

Figures for SLIDing Card: (a) SLIDing Interface, (b) Deformed Geometry

FIT-value DELTA

Status - OPTIONAL

Full Keyword - **`FIT-value (INTERFERENCE)`**

Function - Impose a normal displacement (gap or overlap) in the deformed shape between the nodes of corresponding elements in the interface set.

Input Variables -

DELTA (Real) - REQUIRED

Defines the displacement offset of the interference fit.

Additional Information -

FIT-value, by itself, is a linear interface condition. However, it can be used in conjunction with nonlinear CONTact.

Resultant normal surface tractions will develop in the direction resisting the imposed interference fit.

When FIT-value is present in an interface definition, the tangential component is assumed to be bonded (i.e., it assumes a zero DELTA value for the unspecified components). SLIDing may be used in conjunction with FIT-value if tangential displacement compatibility is not desired.

BONDed and SPRIng-value should not appear in an interface set that contains FIT-value. If SPRIng-value and FIT-value are desired, then perhaps the user may accomplish this modeling option by using interface SPRIng-value with an initial normal spring deflection (or prestress). See the SPRIng-value card in this (**INTERFACE) section for details.

A positive DELTA value will create a gap or positive displacement between nodes of the corresponding elements. A negative DELTA will create an overlap or negative displacement between the nodes.

If FIT-value and CONTact are used in the same interface set, then the DELTA value appearing on the FIT-value card must have a positive value since a negative value implies that a tensile stress can develop on the interface which is impossible on a CONTact interface. See the OFFSet card for initial interface alignment.

If different DELTA values are input in different interface sets, it is the user's responsibility to assure that every pair of nodes (on an interface) has only one unique DELTA value.

It is important to realize that the FIT-value card applies only to the deformed geometry. It determines how the GMRs interact once deformation starts. The FIT-value card does not apply to the undeformed geometry, unlike the EQUIvalence and OFFSet cards.

FIT-value is not available in generalized axisymmetric analysis.

Definition of GMR Compatibility

FIT-value uses a local normal coordinate system.

Example of interface sets containing both FIT-value and CONTact can be found in the example problems: **`elas814.dat`** and **`elas842.dat`** through **`elas847.dat`** .

Examples of Use -

1. Specify an interference fit of 1.0×10^{-3} in the normal direction between the nodes of the corresponding elements in an interface set and a sliding condition in the tangential direction. Figure (a) shows the elements along the interface. The nodes of the GMRs are coincident along the interface. Figure (b) depicts the deformed shape, whereby the imposed gap pushes the interface elements apart by the specified distance. Note that the interference fit is uniform along the interface, i.e., each node moves 1.0×10^{-3} apart in the normal direction.

   To aid the user, Figures (c) and (d) illustrate what the user can expect to view on the screen while in a post-processor such as BESTVIEW, PATRAN, etc.

```
**INTERFACE
  GMR-1   REG1
    SURFace-gmr-1  SURF11
    ELEMents-gmr-1     25   26
  GMR-2   REG2
    SURFace-gmr-2  SURF22
    ELEMents-gmr-2     32   31
  FIT-value      1.0E-03
  SLIDing
```

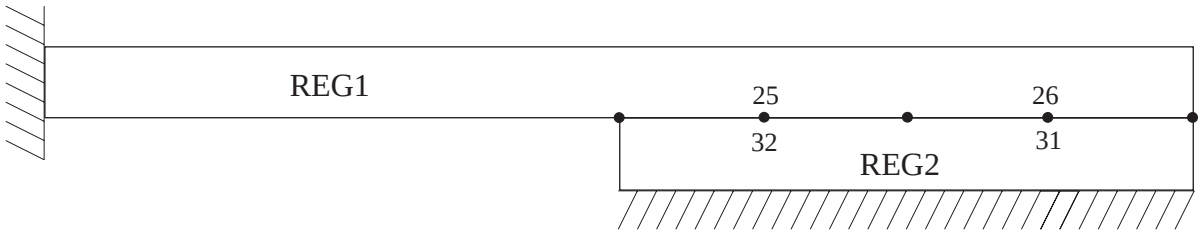


Figure for FIT-value Card: (a) Initially, There is No Spacing Between GMRs

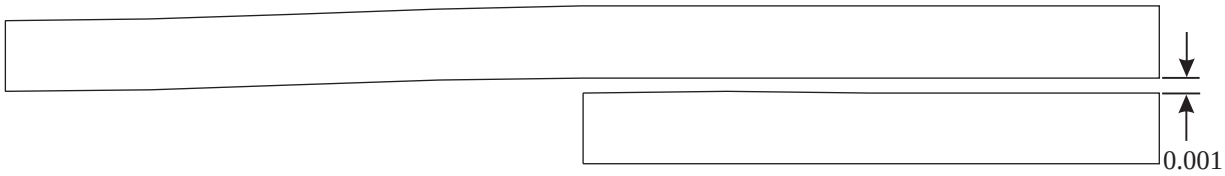
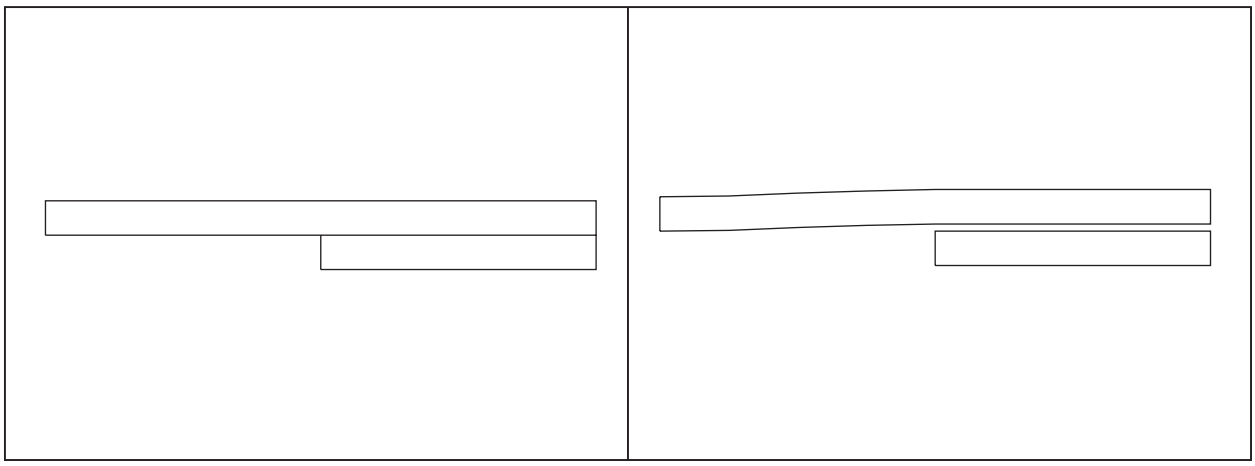


Figure for FIT-value Card: (b) After Deforming, the Imposed FIT-value Spacing Results



(c) Undeformed Geometry (d) Deformed Geometry

Figures for FIT-value Card: Results Viewed in a Post-Processor

2. Two cylinders are press fit together by an interference of 1.0×10^{-3}. Figure (e) shows a 2-D model of a 20 degree slice of the cylinders. The nodes at the interface are modeled as being coicident. Figure (f) depicts the deformed shape, whereby the effect of the press fit is simulated. Note that the interference fit is uniform along the interface, i.e., each node moves 1.0×10^{-3} apart in the normal direction.

```
**INTERFACE
  GMR-1   REG1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1     2
  GMR-2   REG2
    SURFace-gmr-2  SURF2
    ELEMents-gmr-2     8
  FIT-value       1.0E-03
```

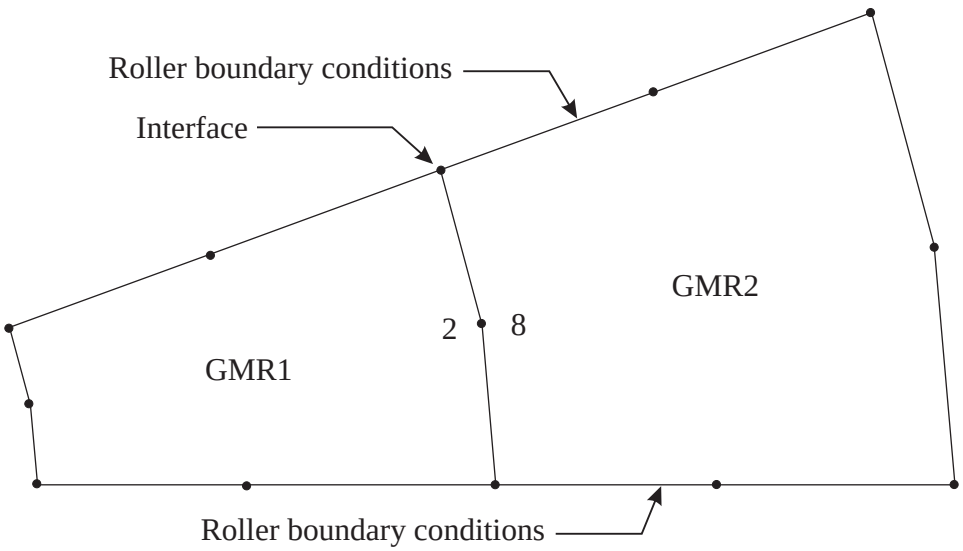


Figure for FIT-value Card: (e) Initially, There is No Spacing Between GMRs

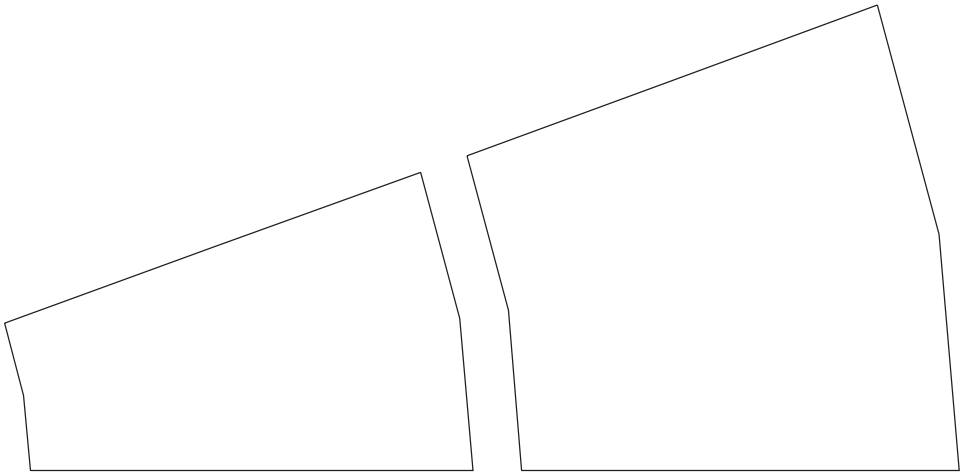


Figure for FIT-value Card: (f) Deformed Shape Illustrates Press Fit

CONTact

Status - OPTIONAL

Full Keyword - **`CONTact`**

Function - Identifies a nonlinear contact interface between corresponding surfaces.

Input Variables - NONE

Additional Information -

If this card is included, an iterative nonlinear algorithm is used to solve the nonlinear system. The TOLErance-convergence and MAXImum-iterations cards in **CASE input may be used to control the nonlinear algorithm.

When the interface nodes are under compression, continuity of displacement is maintained in the normal direction and a normal traction will develop. The tangential displacement of the corresponding interface nodes will, in general, have different displacements. The tangential tractions will be proportional to the normal traction in a direction resisting the relative displacements of the corresponding nodes, where the constant of proportionality is the coefficient of static friction which is input on the FRICtion card. When the interface nodes are in tension, all tractions will be zero and the displacements of the corresponding interface nodes will, in general, have different values.

Elements in adjacent regions that are not in contact in the undeformed shape but may come into contact in the deformed shape may be defined in the contact interface (see TOLErance card). In the situation of an initial gap, tractions will not develop on the contact interface until the initial displacement of the gap is closed. The situation of an initial overlap is somewhat fictitious. **GPBEST** handles the situation as though the corresponding nodes were coincident. Therefore, the tractions are assumed to be zero if the bodies deform in a direction that would separate them and the tractions are nonzero. The relative normal displacement between the two bodies is zero if the bodies deform in a direction that cause them to intersect further.

Keep in mind that **GPBEST** is based on small rotation and small deformation theory.

In order to simulate sliding conditions, use a FRICtion card with a zero coefficient of friction, rather than a SLIDing card.

If FIT-value and CONTact are used in the same interface set, then the DELTA value appearing on the FIT-value card must have a positive value since a negative value implies that a tensile stress can develop on the interface and this is not possible on a CONTact interface. See the OFFSet card for initial interface alignment.

It is important to realize that the CONTact card applies only to the deformed geometry. It determines how the GMRs interact once deformation starts. The CONTact card does not apply to the undeformed geometry, unlike the EQUIvalence and OFFSet cards.

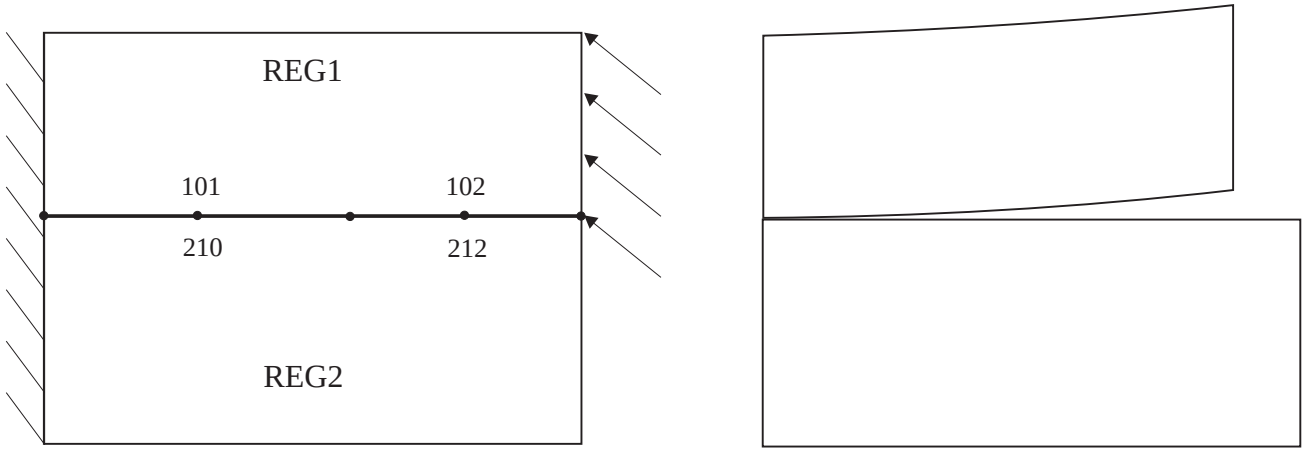
Definition of GMR Compatibility

CONTact uses a local normal coordinate system.

Examples of Use -

1. A nonlinear contact interface with no initial gap between the GMRs. The interface nodes are modeled as coincident, as shown in Figure (a). The deformed geometry is displayed in Figure (b). Note that the load in (a) has caused GMR REG1 to slide laterally and lift up, as Figure (b) clearly shows.

```
**INTERFACE
  GMR-1  REG1
    SURFace-gmr-1 TOP
    ELEMents-gmr-1   101   102
  GMR-2  REG2
    SURFace-gmr-2 BOT
    ELEMents-gmr-2   210   212
  CONTact
    FRICtion      0.2
```

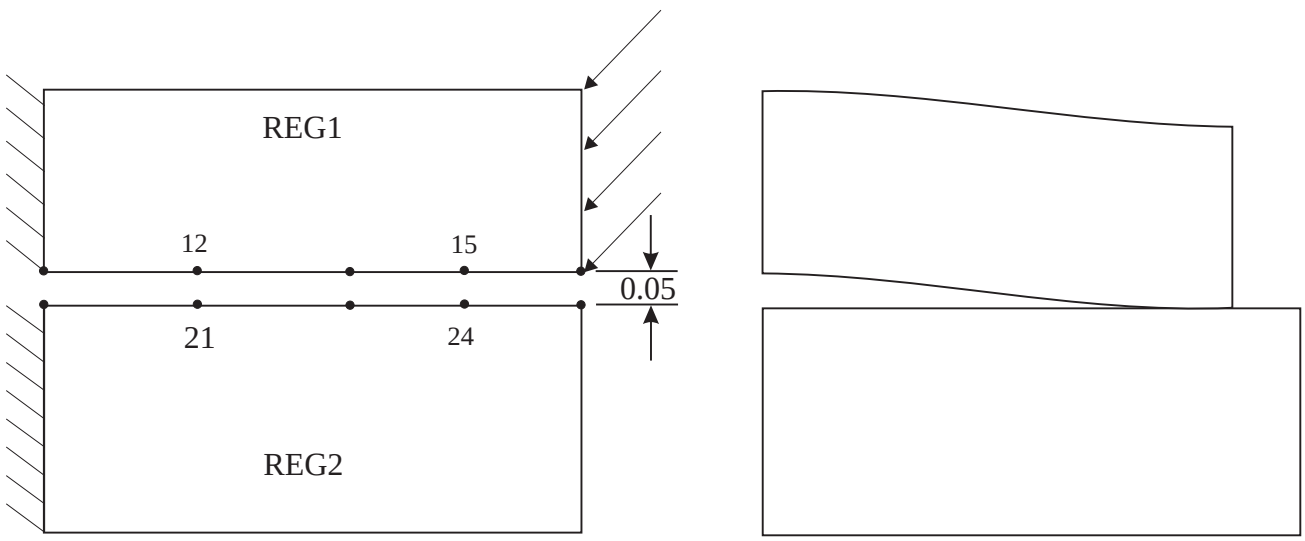


(a) (b)

Figures for CONTact Card: (a) CONTact Interface with no initial gap, (b) Deformed Geometry

2. A nonlinear contact interface with an initial gap of 0.05 between the GMRs. The interface nodes are not coincident. The interface elements are shown in Figure (c). The deformed geometry is displayed in Figure (d). Note that the load in (c) has caused GMR REG1 to slide laterally and contact at the right tip, as Figure (d) shows.

```
**INTERFACE
  TOLErance 0.051
  GMR-1  REG1
    SURFace-gmr-1 TOP
    ELEMents-gmr-1   12   15
  GMR-2  REG2
    SURFace-gmr-2 BOT
    ELEMents-gmr-2   21   24
  CONTact
```



(c) (d)

Figures for CONTact Card: (c) CONTact Interface with Initial gap, (d) Deformed Geometry

SPRIng-value KN KT DN

Status - OPTIONAL

Full Keyword - **`SPRIng-value`**

Function - Defines the spring coefficients on an interface between two regions. The tractions across the interface are linearly related to the relative displacement between the two surfaces. An initial spring deflection normal to the surfaces can also be defined.

Input Variables -

KN (Real) - REQUIRED

Spring coefficient normal to the interface

KT (Real) - REQUIRED

Spring coefficient tangent to the interface

DN (Real) - OPTIONAL

Initial deformation in the spring normal to the surfaces.

Additional Information -

The spring relation in the normal direction is

$$t_n^I = t_n^{II} = k_n(d_n + \Delta u_n)$$

where

t_n^I, t_n^{II} are the tractions normal to the surfaces of regions I and II, respectively,

Δu_n is the relative displacement between the two surfaces, where a positive value represents a separation,

k_n is the spring coefficient normal to the surfaces, and

d_n is the initial deformation in the spring layer normal to the surfaces.

A negative value of d_n indicates the spring is in compression in the underform (initial) state. A positive value of d_n indicates the spring is in tension in the underform (initial) state. From this we see that a spring that is compressed (negative d_n value) before placing it between two regions that are externally restrained, will produce a negative normal traction which will act to force the regions apart.

The spring relation in the tangential direction is

$$t_t^I = -t_t^{II} = k_t \Delta u_t$$

where the subscript t indicates the tangential direction. The tractions act in a direction that opposes the relative tangential displacement. In 3-D analysis, **GPBEST** assumes only one value for the spring coefficient tangent to the interface. This is because the tangential directions of the local coordinate system are defined arbitrarily by **GPBEST**

. Therefore, the tangential spring relationship applies to all directions tangent to the interface.

Note that the dimensions of the SPRIng-value constants in **GPBEST** are traction/length or force/length^3 (NOT force/length as in a physical one-dimensional spring).

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

The spring coefficients K_n and K_t should be positive real numbers. If zero is input, the coefficient will be automatically reset to 1.0E--10.

If DN is not specified, then a value of zero is assumed.

An initial deformation in the tangential spring is not permitted.

SPRIng-value uses a local normal coordinate system.

Interface springs are NOT permitted in mixed region problems involving an axisymmetric region and a 2-D planar region, nor are they permitted in problems with 2-D planar regions with thickness other than one.

The SPRIng-value keyword cannot be used in a **INTERFACE definition which also includes a CONTact, BONDed or SLIDing keyword. Note, the tangential components of the spring can be set to a large value for a BONDed condition or a zero value for a SLIDing condition.

Examples of Use -

1. Defines an interface between two regions connected by a spring which acts only in the direction normal to the surfaces. A condition of sliding (i.e., no resistance) is created by choosing the spring coefficient in the tangential direction to be zero.

```
**INTERFACE
  GMR-1 REG1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1  101  102
  GMR-2 REG2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2  201  202
      SPRIng-value  10.0E+06  0.0
```

2. Define an interface between two regions connected by springs in which the normal spring has an initial compressed deflection of 0.1. Since the spring constant is 1000 N/m^3, this is equivalent to a spring that is prestressed to a level of 100 N/m^2.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1  101  102
  GMR-2 GMR2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2  201  202
      SPRIng-value  1000.0   100.0   0.1
```

CONVection-value C1
RESIstance-value R1

Status - OPTIONAL

Full Keyword - **`CONVection-value`**

Full Keyword - **`RESIstance-value`**

Function - Identifies an interface with thermal convection/resistance between the corresponding surfaces. The flux across this interface is linearly related to the temperature difference between the two surfaces.

Input Variables -

C1 (Real) - REQUIRED

Thermal convection coefficient (C)

R1 (Real) - REQUIRED

Thermal resistance coefficient ($R = \frac{1}{C}$)

Additional Information -

C1 is the reciprocal of R1.

The CONVection option utilizes the relationship:

$$q_1 = C1(\theta_1 - \theta_2)$$

The RESIstance option utilizes the relationship:

$$q_1 = \frac{1}{R1}(\theta_1 - \theta_2)$$

where:

θ_1	local temperature of GMR-1.
θ_2	local temperature of GMR-2.
q_1	local heat flux from GMR-1.

The user is responsible for providing C or R in the proper units consistent with the specification of material properties, geometry and boundary conditions.

The convection (C) or resistance (R) should be a positive real number ($R > 0$). If zero is input, the coefficient will be automatically reset to 1.0E--10.

GENEralized-interface CONV A1 B1 TA1
GENEralized-interface CONV A2 B2 TA2

Status - OPTIONAL (Available only in HEAT conduction analysis)

Full Keyword - **`GENEralized-interface CONVECTION`**

Function - Identifies an interface incorporating a generalized convection condition defined by six parameters.

Input Variables -

A1, B1, TA1 (R)
A2, B2, TA2 (R)
Coefficients in the generalized convection interface relation.

Additional Information -

The interface relationshps can be written as

$$q_1 = A1(TA1 - \theta_1) + B1(\theta_1 - \theta_2)$$

$$q_2 = A2(TA2 - \theta_2) + B2(\theta_2 - \theta_1)$$

where:

θ_1 local temperature of the first GMR defined in the interface
θ_2 local temperature of the second GMR defined in the interface
q_1 local heat flux from the first GMR defined in the interface
q_2 local heat flux from the second GMR defined in the interface

Two cards must be specified. The first card defines the coefficients for the flux q_1 of the first GMR defined in the interface definition, while the second card defines the coefficients for the flux q_2 of the second GMR defined in the interface definition.

The GENEralized-interface CONVECTION condition is only valid for HEAT analysis. The GENEralized convection condition is available in uncoupled thermal-elastic (CTHErmal) analysis, however, it is not fully supported, and must be used with caution.

The GENEralized-interface CONVECTION keyword cannot be used in an **INTERFACE definition which also includes a BONDed, CONVection, RESIstance-value or CONTact keyword.

Examples of Use -

1. Define a generalized convective interface which defaults to the standard resistance interface with $R = 0.2$

```
**INTERFACE
  GMR-1  GMR1
     SURFace-gmr-1 SURF1
     ELEMents-gmr-1  11  12
  GMR-2  GMR2
```

```
   SURFace-gmr-2 SURF2
   ELEMents-gmr-2  21  22
GENEralized-interface CONV   0.0  5.0  0.0
GENEralized-interface CONV   0.0  5.0  0.0
```

IMPEdance-value R1

ADMIttance-value A1

Status - OPTIONAL

Full Keyword - **`IMPEdance-value`**

Full Keyword - **`ADMIttance-value`**

Function - Identifies an interface with impedance between the corresponding surfaces. The velocity across this interface is linearly related to the acoustic pressure difference between the two surfaces.

Input Variables -

A1 - REQUIRED

Admittance coefficient (A1)

R1 - REQUIRED

Impedance coefficient (R1 = 1/A1)

Additional Information -

R1 is the reciprocal of A1.

The IMPEdance-value and ADMIttance-value options utilize the relationship:

$$v_2 = \frac{1}{A1}(p_2 - p_1) \qquad \text{and} \qquad v_1 = -v_2$$

where:

p_1	local acoustic pressure of GMR-1.
p_2	local acoustic pressure of GMR-2.
v_1	local normal velocity from GMR-1.
v_2	local normal velocity from GMR-2.

The user is responsible for providing A1 or R1 in the units consistent with the specification of material properties, geometry and boundary conditions.

If the admittance/impedance value input is less than 1.0E-10, then it is set to 1.0E-10; if it is greater than 1.0E+10, then it is set to 1.0E+10.

This card is only relevant to periodic ACOUstic analysis.

Examples of Use -

1. Defines impedance at the interface between two regions.

```
**INTERFACE
  GMR-1 GMR1
    SURFace-gmr-1  SURF1
    ELEMents-gmr-1 12
  GMR-2 GMR2
    SURFace-gmr-2  SURF2
    ELEMents-gmr-2 21
  IMPEdance-value 1.0
```

5.4.5 ADDITIONAL INTERFACE INPUT

FRICtion FVAL

Status - OPTIONAL (for CONTact only)

Full Keyword - **FRICtion**

Function -

Defines the coefficient of static friction between the corresponding surfaces.

Input Variables -

FVAL (Real) - REQUIRED

Coefficient of static friction

Additional Information -

The value of FVAL should be greater than or equal to zero.

If the CONTact card is input on interface definition and the FRICtion card is not input, then a zero coefficient of static friction is assumed.

This card can only be used in conjunction with the nonlinear CONTact interface definition.

Examples of Use -

1. Defines a nonlinear contact interface with friction between two regions. Tractions will only develop on the interface nodes that are in contact when the body deforms.

```
**INTERFACE
  GMR-1  REG1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1   101   102
  GMR-2 REG2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2   201   202
  CONTact
    FRICtion      0.2
```

EQUIvalence

Status - OPTIONAL (for CONTact only)

Full Keyword - **`EQUIvalence`**

Function - Specify that the distance between interface node pairs are assumed to be zero at the start of a contact analysis.

Input Variables - NONE

Additional Information -

Gaps and/or overlaps (whether physically existing in the model or defined using the OFFSet card) have an effect on the results when they are present in a nonlinear CONTact analysis. The EQUIvalence card assumes that the nodes are in contact at the start of a CONTact analysis even if an actual gap exists between nodal pairs. The EQUIvalence card has no effect on a linear interface (BONDed, SLIDing SPRIng-value, etc.).

The EQUIvalence option is used to negate an actual gap or overlap that may exist between regions (GMRs) in the model but was created unintentionally.

The OFFSet card, on the other hand can be used to specify a gap that does not actually exist in the model (or a different gap size is desired).

If neither the OFFSet card nor the EQUIvalence card is input in the **INTErface set, then the actual physical distance (in the average normal direction) between the coordinate of each node pair is assumed. If this distance is neg

This card must not appear before the keyword CONTACT appears in an interface set. The keywords OFFSet and EQUIvalence must not appear in the same interface set.

It is important to realize that the EQUIvalence card applies only to the undeformed geometry. In other words, it describes what the geometry looks like before any deformation occurs. It is as if the nodal coordinates are updated before the loads are applied. No tractions or stress develops in the body due to the use of this card. This is in contrast to the FIT-value option which is similar to an applied displa

Example of interface sets containing OFFSet and/or EQUIvalence can be found in the example problems:

`elas225.dat`, **`elas226.dat`**, **`elas227.dat`**, **`elas228.dat`**, **`elas818.dat`**, **`elas819.dat`**, **`elas848.dat`** and **`elas849.dat`** .

Examples of Use -

1. This illustrates the use of the EQUIvalence. The interface elements between two regions (REG1 and REG2) have been modeled with coincident nodes (see Figure (a)). However, in translating the model from a CAD system, the nodes do not actually line up as Figure (b) shows. In order to accommodate this, the EQUIvalence

card is used to straighten out the mismatch. Figure (c) illustrates the effect of the EQUIvalence option on the $undeformed$ geometry. The nonlinear contact algorithm is run in **GPBEST** , and the final deformed geometry is displayed in Figure (d).

To aid the user, Figures (e) and (f) illustrate what the user can expect to view on the screen while in a post-processor such as BESTVIEW, PATRAN, etc.

```
**INTERFACE
  GMR-1  REG1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1   101   102
  GMR-2 REG2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2   201   202
  CONTact
    EQUIvalence
```

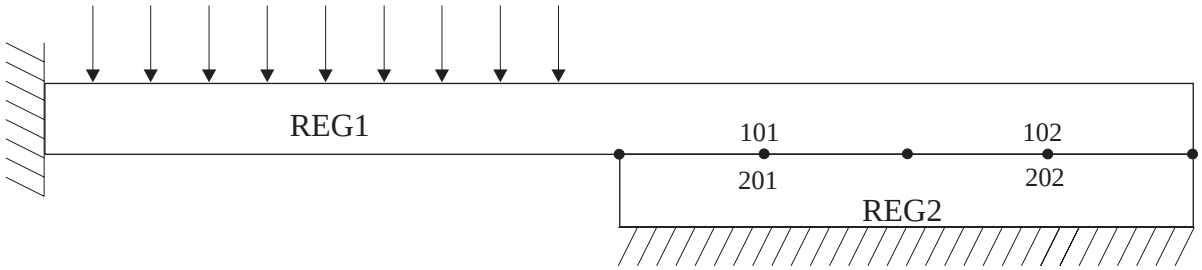


Figure for EQUIvalence Card: (a) Modeled Geometry.

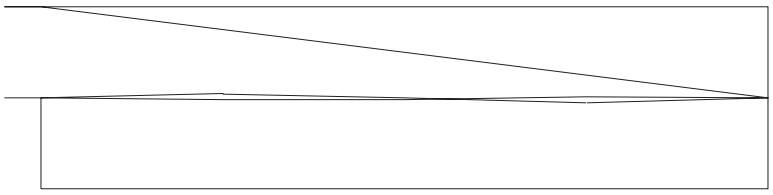


Figure for EQUIvalence Card: (b) Zoomed in View Illustrates the Mismatch between GMRs at the Interface.

Figure for EQUIvalence Card: (c) Effect of Applying the EQUIvalence Card. This is a Plot of the Undeformed Geometry.

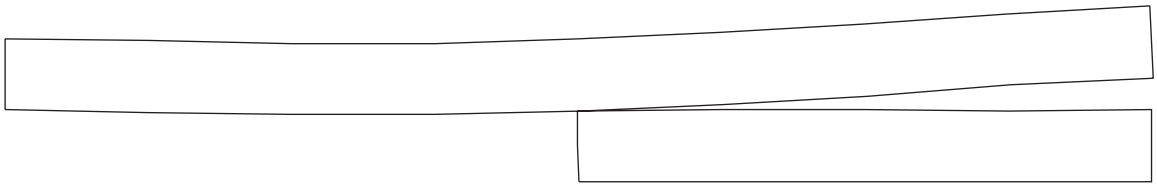
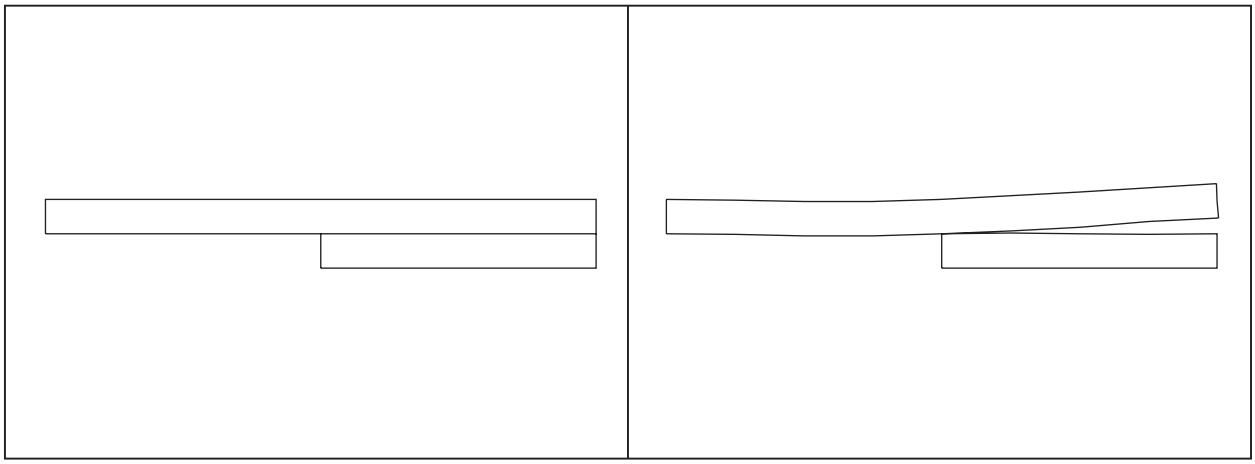


Figure for EQUIvalence Card: (d) Effect After Running Nonlinear CONTact Algorithm. This is a Plot of the Final Deformed Geometry.



(e) Undeformed Geometry (f) Deformed Geometry After Running CONTact

Figure for EQUIvalence Card: Results Viewed in a Post-Processor

OFFSet TYPE

Status - OPTIONAL (for CONTact only)

Full Keyword - **`OFFSet`**

Function - Indicates that the location of the nodes across a CONTact interface are assumed to be OFFSet from one another by a specified distance, and specify the type (CONSTANT or VARIABLE) of OFFSet to be input.

Input Variables -

TYPE (Alphanumeric) - REQUIRED

Allowable values are CONSTANT or VARIABLE

CONSTANT - Specifies that all node pairs contained in the interface set will have the same value.

VARIABLE - Specifies that an offset value will be input for each node pair contained in the interface set.

Additional Information -

Gaps and/or overlaps (whether physically existing in the model or defined using the OFFSet card) have an effect on the results when they are present in a nonlinear CONTact analysis. The OFFSet card is used to specify a gap that does not actually exist in the model (or when a different gap size is desired). The OFFSet card has no effect on a linear interface (BONDed, SLIDing SPRIng-value, etc.).

If an OFFSet definition is contained in an interface set, then the OFFSet values are taken to be the (averaged) normal distance between the nodes. If an actual gap exists between the coordinates of the node pairs, this gap is ign

The EQUIvalence card can be used to negate an actual gap or overlap that may exist between nodes in the model but was created unintentionally.

If neither the OFFSet card nor the EQUIvalence card is input in the **INTErface set, then the actual physical distance (in the average normal direction) between the coordinate of each node pair is assumed. If this distance is neg

This card must not appear before the keyword CONTACT appears in an interface set. The keywords OFFSet and EQUIvalence must not appear in the same interface set.

It is important to realize that the OFFSet card applies only to the undeformed geometry. In other words, it describes what the geometry looks like before any deformation occurs. It is as if the nodal coordinates are updated before the loads are applied. No tractions or stress develops in the body due to the use of this card. This is in contrast to the FIT-value option which is similar to an applied displa

Example of interface sets containing OFFSet and EQUIvalence cards can be found in the example problems:

`elas225.dat`, **`elas226.dat`**, **`elas227.dat`**, **`elas228.dat`**,

elas818.dat, **elas819.dat**, **elas848.dat** and **elas849.dat** .

Examples of Use -

1. The interface elements between two regions (REG1 and REG2) have been modeled with coincident nodes (see Figure (a)). However, the physical components actually have a constant gap of 0.1 in the undeformed state, as shown in Figure (b). Include the effect of this gap in the nonlinear contact algorithm. Assume that the surfaces slide freely when in contact. The load forces the two beams together, then causes the REG1 beam to lift off at the tip (see Figure (c)).

   To aid the user, Figures (d) and (e) illustrate what the user can expect to view on the screen while in a post-processor such as BESTVIEW, PATRAN, etc.

```
**INTERFACE
  GMR-1  REG1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1   101   102
  GMR-2 REG2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2   201   202
  CONTact
    FRICtion  0.0
    OFFSet    CONSTANT
    O-VAlue   0.1
```

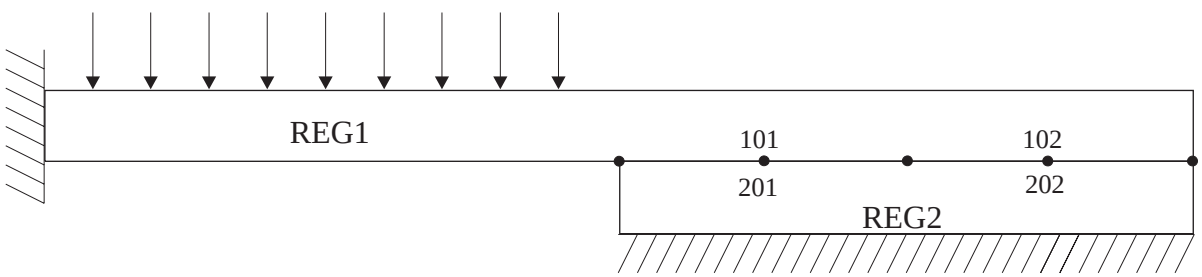


Figure for OFFSet Card: (a) Modeled Geometry.

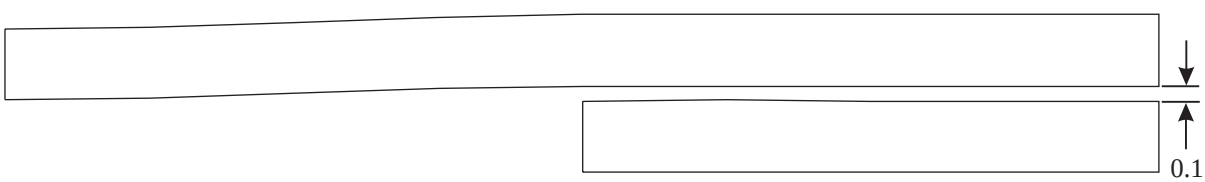


Figure for OFFSet Card: (b) Effect on $Undeformed$ Geometry After Applying OFFSet 0.1.

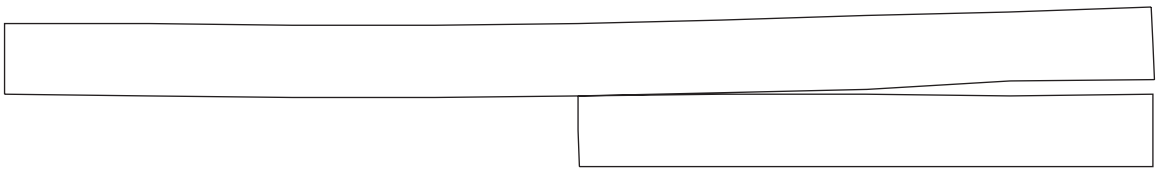
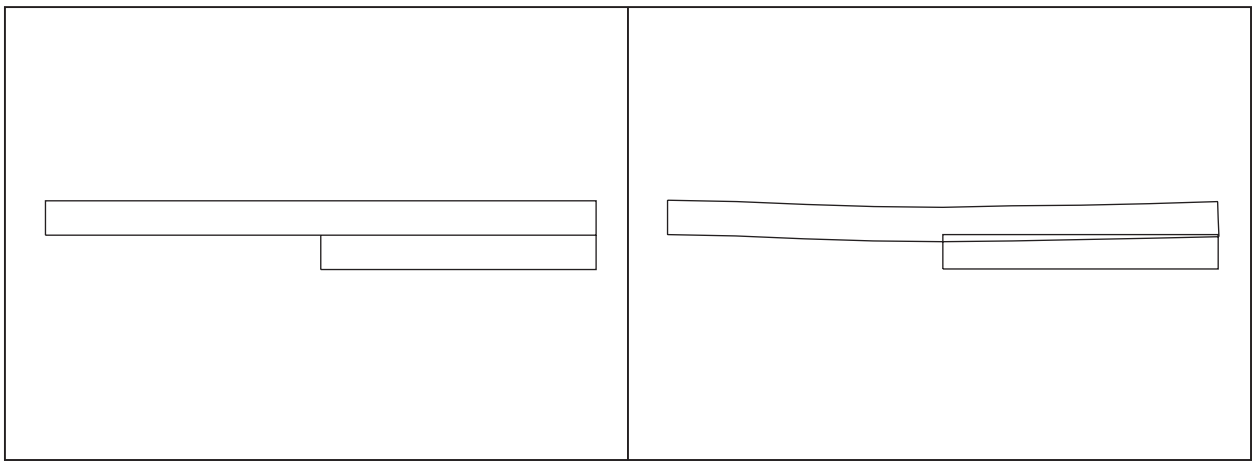


Figure for OFFSet Card: (c) Final Deformed Geometry After Running Nonlinear CONTact.



(d) Undeformed Geometry (e) Deformed Geometry After Running CONTact

Figure for OFFSet Card: Results Viewed in a Post-Processor

NOTE: While post-processing this problem, the deformed geometry plot (Figure (e)) appears as if the upper beam overlaps the lower beam. This is because the post-processor does not apply the ‘‘OFFSet’’ gap to the undeformed geometry. The user must be aware that this is happening. Figures (a), (b), and (c) indicate what is actually taking place inside **GPBEST** .

O-VAlue FVAL1 (FVAL2 FVAL3 . . . FVALN)

Status - REQUIRED if OFFSet is input

Full Keyword - **`O-VAlue`**

Function - To specify the OFFSet value (or values) for an interface in a contact analysis. An OFFSet represents a user defined gap between the nodes of two regions (connected by a CONTact interface) which during deformation must

Input Variables -

FVAL1 (Positive-Real) - Only one value required if OFFSet CONSTANT was input on the previous card.

FVAL1 - Is the (constant) OFFSet value of every node pair in the interface.

FVAL1 - FVALN (Positive-Real) - If OFFSet VARIABLE was input than one value is required for every N source point pairs contained in the interface definition.

FVAL1 is the OFFSet value of the first node pair defined using the POINt-gmr cards. FVAL2 is the value of the second node pair defined, and so on.

Additional Information -

All OFFSet values input should be a positive real number (or zero). A negative number would represent an overlap of the two nodes of the interface. This would indicate that the surface is already in contact. Therefore, if a neg

If ‘‘OFFSet VARIABLE’’ is contained in the interface set, then the POINt-interface cards must be input for both GMRs defined in the set and one offset value must be entered for each node pair. The POINt-gmr-1 list for the first G

The O-VAlue card may be repeated as often as necessary to complete the definition for all node pairs. Each time the card is repeated, it must begin with the keyword O-VAlue and must follow immediately after the last card.

Examples of Use -

1. The interface elements between two regions (REG1 and REG2) have been modeled with coincident nodes (see Fig. (a)). However, the physical components actually have a variable gap which varies from zero at the edges to 0.1 at the ce

```
**INTERFACE
  GMR-1   REG1
    SURFace-gmr-1   SURF1
    ELEMents-gmr-1  101   102   103
    POINt-gmr-1     110   111   112   113   114   115   116
  GMR-2   REG2
    SURFace-gmr-2   SURF2
    ELEMents-gmr-2  203   202   201
    POINt-gmr-2     224   223   222   221   220   219   218
  CONTACT
    FRICTION        0.2
    OFFSet          VARIABLE
    O-VAlue         0.0   0.3   0.7   0.1   0.7   0.3   0.0
```

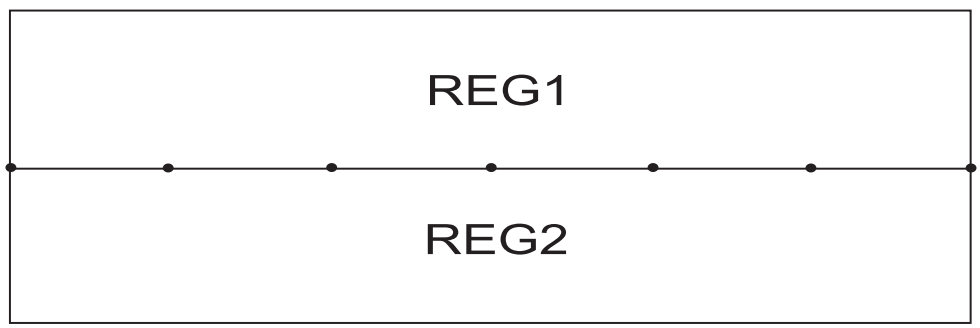


Figure (a) Modeled Geometry

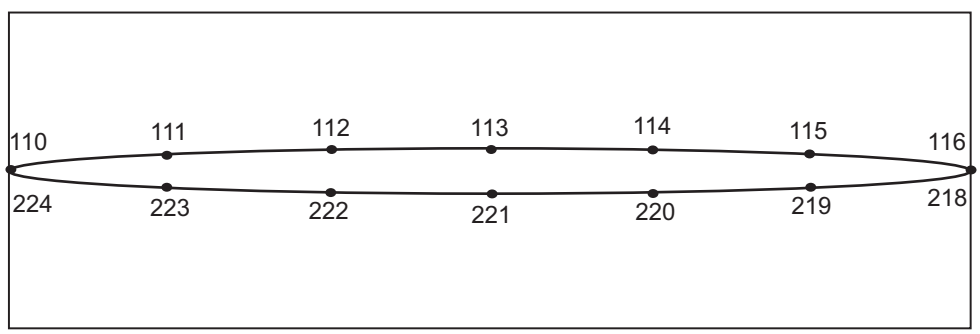


Figure (b) Effect on Stress-free, Undeformed Geometry
after Applying a Variable Offset

5.4.6 CYCLIC SYMMETRY INTERFACE DEFINITION

CYCLic

Status - OPTIONAL

Full Keyword - **`CYCLic`**

Function - Identifies a cyclic symmetry interface capability.

Input Variables - NONE

Additional Information -

This type of interface condition establishes a BONDed relationship between two boundary surfaces. In order for this condition to be applied, the elements and nodes of the two boundary surfaces involved must be such that one can be exactly superimposed on the other by a rotation about an axis of the global coordinate system. Further, the imposed boundary conditions of the problem must be such that the deformed shape of one boundary surface can be exactly superimposed on the other by the same rotation. This option is intended for the analysis of (periodic) structures subjected to periodic loading.

Rigid body translation is automatically prevented in the radial direction (i.e. perpendicular to the cyclic axis). However, rigid body translation along the cyclic axis as well as rigid body rotation about that same axis are **not** automatically prevented by invoking the CYCLic option. Consequently, these motions must be constrained explicitly by the user.

Since a cyclic interface condition involves all components of displacement and traction, **no other boundary condition may be applied to the elements that are involved.**

Local coordinate systems are established for each node on the second boundary surface defined in the cyclic interface definition. Displacement and spring boundary conditions applied to elements adjacent to a cyclic symmetry interface should be verified for correctness when a solution is obtained. If rigid body rotation about the cyclic axis is not correctly prevented, displacements may have erratic behavior. (Tractions, however, may be correct.)

If the CYCLic card is included, a BONDed interface is assumed. The SLIDing, FIT-value, CONTact, SPRIng-value, RESIstance-value, or GENEralized-interface keywords cannot be used in an **INTERFACE definition which contains the CYCLic keyword.

The CYCLic symmetry interface definition allows cyclic compatibility between elements that are contained in the same GMR and in the same SURFace-gmr-1 definition as well as elements that may be part of different regions.

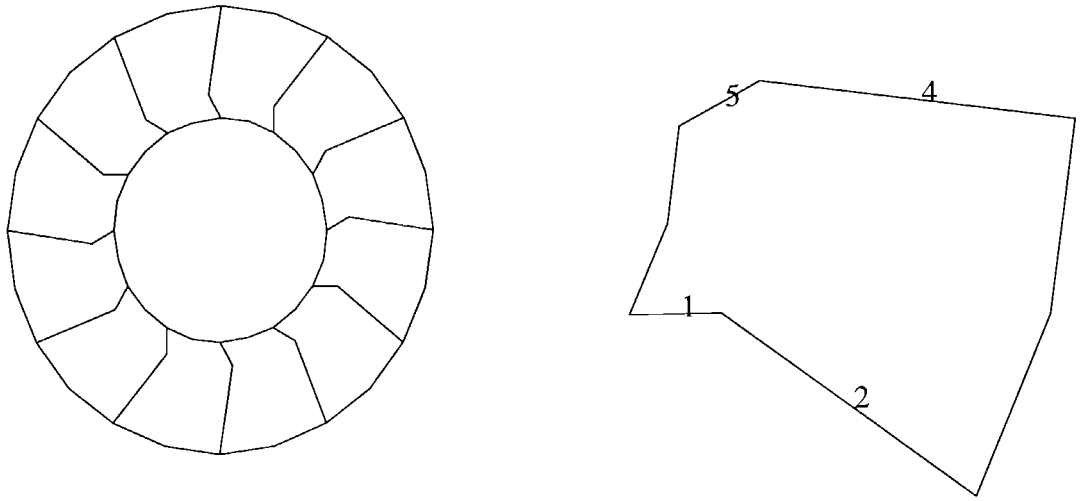
Definition of GMR Compatibility

CYCLic uses a local normal coordinate system.

Examples of Use -

1. Activate option for cyclic symmetry. Only a 30-degree section is modeled in a single GMR. (Note that the matching edges do not have to be straight. However, they must line up with each other when rotated through ANGLe.)

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
    ELEMents-gmr-1    1    2
  GMR-2 GMR1
  SURFace-gmr-2 SURF1
    ELEMents-gmr-2    4    5
  CYCLic
    ANGLe    30.
    DIREction-cyclic    0.0   0.0   1.0
```



Entire Problem Single GMR (30 degrees)

Figure for **INTERFACE with CYCLic SYMMETRY Option - 1 GMR

2. Activate option for cyclic symmetry. Base region contains two GMRs.

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
    ELEMents-gmr-1    1    2
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
    ELEMents-gmr-2   24   25
  CYCLic
    ANGLe    60.
    DIREction-cyclic    0.0   0.0   1.0
```

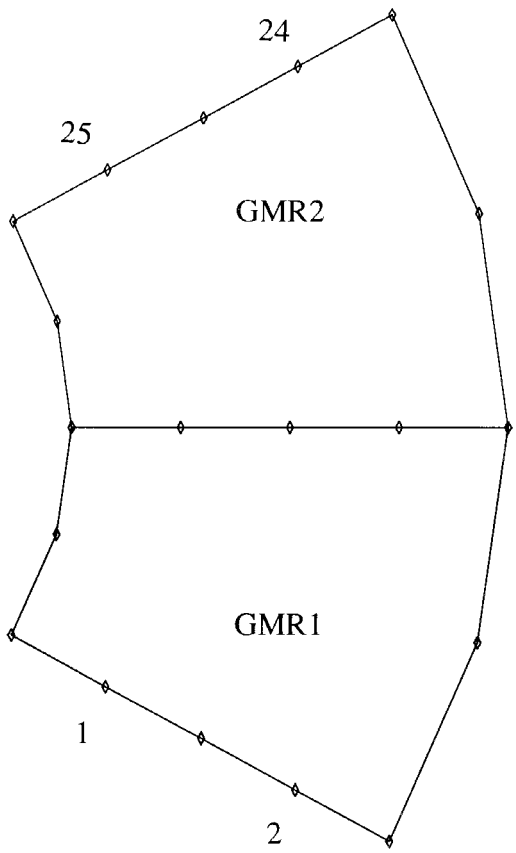


Figure for **INTERFACE with CYCLic SYMMETRY Option - 2 GMRs

ANGLe THETA

Status - REQUIRED (if CYCLic 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 in the positive axis direction (using the right-hand rule)

See DIREction-cyclic card (next page) for specification of the positive cyclic axis direction.

Additional Information - NONE

Examples of Use -

1. Specifies an angle of 72 degrees between the two surfaces referenced in the cyclic symmetry condition.

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1 3
  GMR-2 GMR1
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2 5
  CYCLic
  ANGLe 72.0
  DIREction-cyclic  0.  0.  1.
```

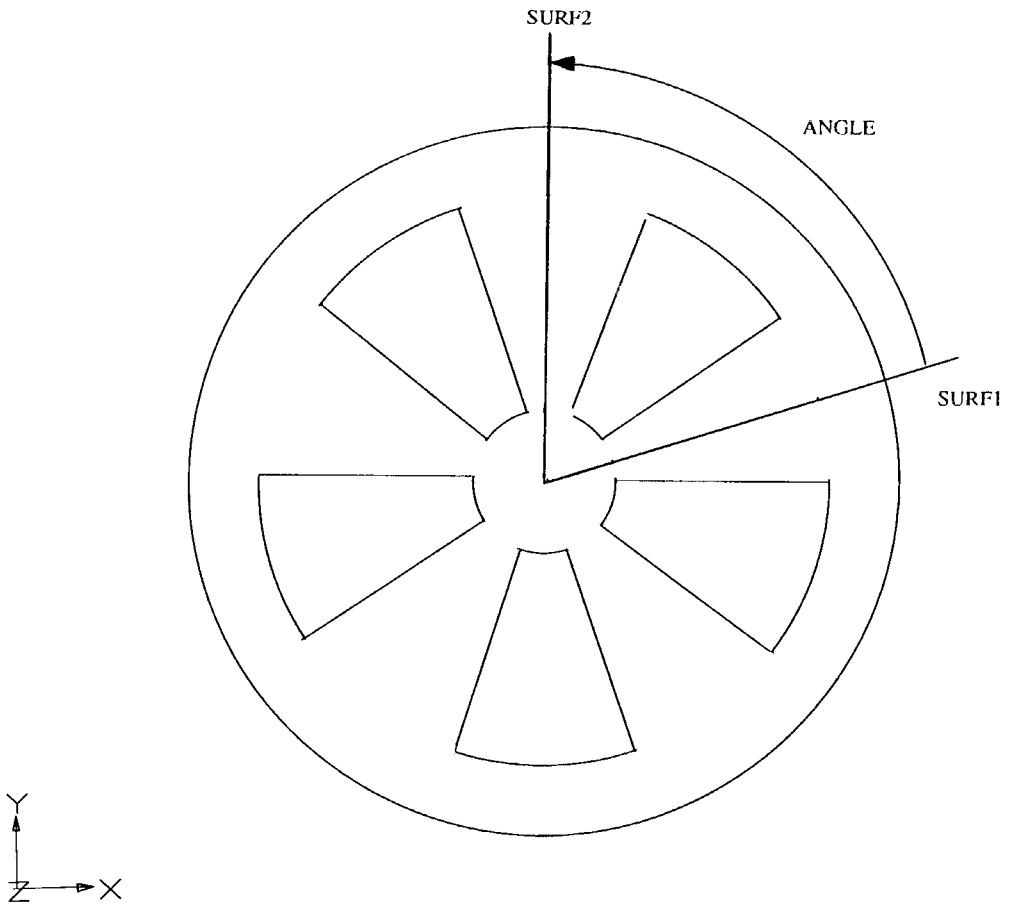


Figure for **INTERFACE with CYCLic SYMMETRY / ANGLe Option

(The positive angle of 72 degrees is defined by a right-hand rule rotation about the z-axis.)

DIREction-cyclic X Y Z

Status - OPTIONAL (if CYCLic is specified, for 3D only)

Full Keyword - **`DIREction-cyclic`**

Function - Defines the axis of rotation for cyclic symmetry.

Input Variables -

X,Y,Z (Real) - REQUIRED

Components of a vector along the positive direction of the axis of rotation.

Additional Information -

The axis of rotation must be one of the three axes of the global coordinate system. Therefore, two of the three components on the DIREction-cyclic must be zero.

If CYCLic is input and this card is omitted, then rotation is assumed to be about the positive z-axis.

In 2D problems this card is ignored and the positive z-axis is assumed to be the axis of rotation.

Examples of Use -

1. Use a 22.5-degree slice to model a solid cylinder using CYCLic symmetry. Rotation is around the Y-axis. Note that this illustrates a 3-D problem.

```
**INTERFACE
  GMR-1  GMR1
    SURFace-gmr-1  SURF11
    ELEMents-gmr-1   101
  GMR-2  GMR1
    SURFace-gmr-2  SURF11
    ELEMents-gmr-2   103
  CYCL
    ANGLe 22.5
    DIREction-cyclic 0.0 1.0 0.0     $ ROTATION AROUND Y-AXIS
```

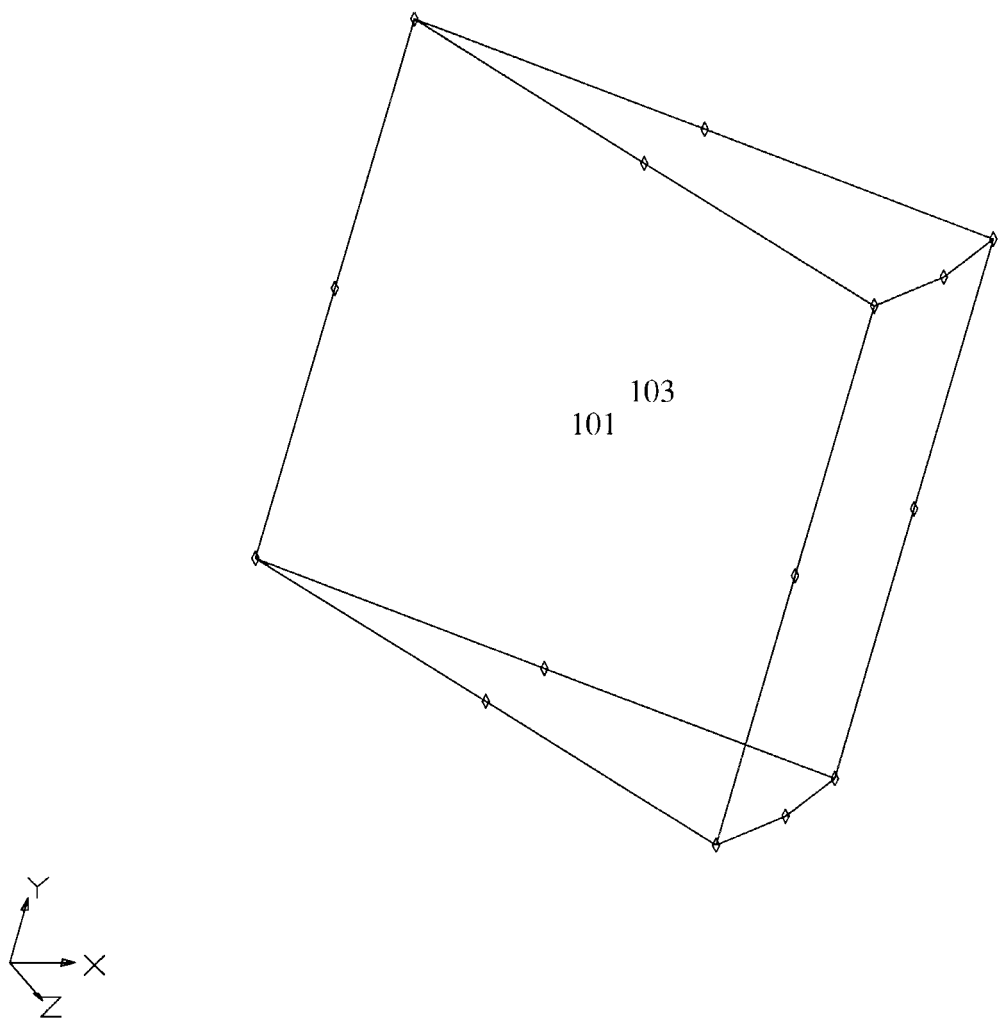


Figure for **INTERFACE with CYCLic symmetry / DIREction-cyclic Option

5.5 DEFINITION OF BOUNDARY CONDITIONS

SECTION	KEYWORD	PURPOSE	PAGE
5.5.1	Boundary Condition Set Card		5.5.18
	**BCSET	start of the B.C. definition	5.5.18
5.5.2	Boundary Condition Identification		5.5.19
	ID-Bcset	name of B.C. set	5.5.19
5.5.3	Identification of Boundary Condition Type		5.5.21
	VALUe	for specified B.C. value input	5.5.21
	RELAtion	for B.C. relation between boundary quantities	5.5.22
	VARIable	for specified B.C. value input which may change during subsequent RESTart-gmr runs	5.5.23
	MOVIng	for moving heat source specification	5.5.25
	INTErface	boundary condition on interface	5.5.27
	LOCAl	for local definition of B.C.	5.5.30
	DIREction-bc	input for local tangential directions in 3-D	5.5.38
	(NONE)	explicit definition for local directions	5.5.40
5.5.4	Definition of Surface for Application of Boundary Conditions		5.5.43
	GMR-bc	identifies a GMR	5.5.44
	SURFace-bc	identifies the surface for this B.C. set	5.5.45
	ELEMents-bc	identifies surface elements	5.5.48
	POINts-bc	identifies surface points	5.5.49
	TIMEs-bc	defines the time for input	5.5.51
	FREQuency-bc	defines the time for input	5.5.54
5.5.5	Value Boundary Condition for Surface Elements		5.5.56
	TRACtion-bc	traction B.C. input	5.5.56
	LOAD-bc	load B.C. input	5.5.62
	GLOAd-bc	load B.C. input	5.5.65
	TORQue-bc	torque B.C. input	5.5.67
	DISPlacement-bc	displacement B.C. input	5.5.70
	FLUX-bc	flux B.C. input	5.5.72
	TEMPeratures-bc	temperature B.C. input	5.5.74
	PRESsure-bc	pressure B.C. input	5.5.75
	VELOcity-bc	velocity B.C. input	5.5.77

SECTION	KEYWORD	PURPOSE	PAGE
5.5.6	Definition of Hole/Cooling Line for Application of Boundary Conditions		5.5.78
	GMR-line-bc	identifies a GMR	5.5.78
	HOLE-line-bc	indicates B.C. on holes	5.5.79
	CLINe-line-bc	indicates B.C. on clines	5.5.79
	ELEMents-bc	identifies the hole/cline elements	5.5.80
	POINts-line-bc	identifies the hole/cline points	5.5.81
	TIMEs-line-bc	defines the time for input	5.5.82
5.5.7	Value Boundary Condition for Hole/Cooling Line Elements		5.5.83
	FLUX-line-bc	flux B.C. input	5.5.83
	TEMPeratures-line-bc	temperature B.C. input	5.5.84
5.5.8	Definition of Space/Time Variation		5.5.85
	SPLIst	source (field variable) point list	5.5.85
	T-VAlue	nodal value of B.C.	5.5.87
5.5.9	Relation Boundary Condition		5.5.91
	SPRIng	spring relation (between displacement and traction)	5.5.91
	CONVection	convection relation (between temperature and flux)	5.5.95
	RESIstance	resistance relation (between temperature and flux)	5.5.95
	IMPEdance	impedance condition	5.5.97
	ADMIttance	admittance condition	5.5.100
5.5.10	Moving Boundary Condition		5.5.101
	SOURce	heat source	5.5.101
	TIMEs-moving	defines time	5.5.102
	ENERgy-rate	defines energy rate	5.5.103

SECTION	KEYWORD	PURPOSE	PAGE
5.5.11	Node Based Boundary Conditions		5.5.105
	LOCAl-fem FEM	defines the direction cosines of the local axes at certain nodes for defining displacement or forces along the local axes (FE GMR)	5.5.105
	POINts-fem	list of nodes over which displacements and forces are defined	5.5.107
	DISPlacement-fem	displacement BC input (FE GMR)	5.5.108
	STIFfness-fem	spring BC input (FE GMR)	5.5.110
	FORCe-fem	force BC input (FE GMR)	5.5.112
	DROP-fem	acoustic pressure drop BC input (Indirect boundary element acoustic GMR)	5.5.114

Modeling Boundary Condtions

Boundary conditions are constraints and loads specified on the surface of the **GPBEST** model. They act only on boundary elements. For conditions at the interface between two GMRs, see Section 5.4 of this manual. For loads that act throughout a body, see Section 5.6 (Body Forces). Models can also be constrained through symmetry options (see Section 5.1).

Boundary conditions are specified in **GPBEST** through the **BCSET data block. Since there are so many different types of boundary conditions, the **BCSET data block can be somewhat complex. However, for problems of a simpler nature, the boundary condition input is quite basic.

In elastic boundary element analysis, the primary variables are displacements and tractions. For heat transfer analysis, the primary variables are temperature and flux. For potential analysis, the primary variables are the potential and flux. All of these quantities can be specified on boundary elements using the **BCSET data block.

In addition to specifying the primary variables, boundary conditions can also specify a relationship between variables. This is known as a RELAtion, and is useful to model spring-type boundary conditions and convection conditions in heat transfer analysis. Also, boundary conditions can remain constant in the case of static analyses or they can vary over time for time-dependent problems.

Boundary element boundary conditions in **GPBEST** are typically specified on elements as opposed to nodes. This is somewhat different than the finite element method, where all boundary conditions are specified on nodes. Nevertheless, boundary conditions can be applied to specific nodes in **GPBEST** by referencing the element and then the node in the **BCSET data block. For coupled structural-acoustics problems, the structure is modeled by finite elements and the boundary conditions are in the form of nodal quantities. This is described in the last section of this chapter.

The concept of 'tractions' is new to many engineers, because finite element programs typically endorse applying point forces at nodes. Tractions are merely loads distributed over a boundary element. Its units are (force/area). For 3-D problems, tractions applied on surface elements have units of psi (English), or N/mm^2 (metric). In 2-D, tractions have units of lb/in (English), or N/mm (metric). Either way, the traction is a *distributed* force. Tractions can also have a direction of action. Figure 5.5.1a shows a uniform pressure load on a 3-D surface element, while Figure 5.5.1b illustrates a traction applied at a skewed angle in relation to a 2-D boundary element.

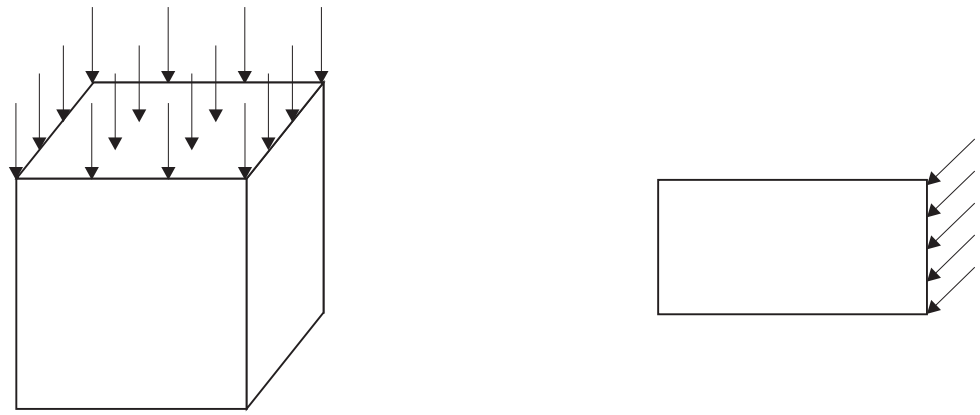


Figure 5.5.1 (a) Traction on 3-D Element (b) Traction on 2-D Element

In finite element analysis, distributed loads can be manually applied as point forces if the magnitude at the nodes take into account the shape functions. For instance, in Figure 5.5.2, a uniform pressure is applied to a 3-node finite element beam element in the ratio of 1/6, 4/6, 1/6, which accounts for the larger shape function at the middle node. However, as shown in the figure, the boundary element uniform pressure is *not weighted differently at each node!* In boundary elements, the exact value of the traction is specified on the element (or at each node), and **GPBEST** takes care of any interpolation internally.

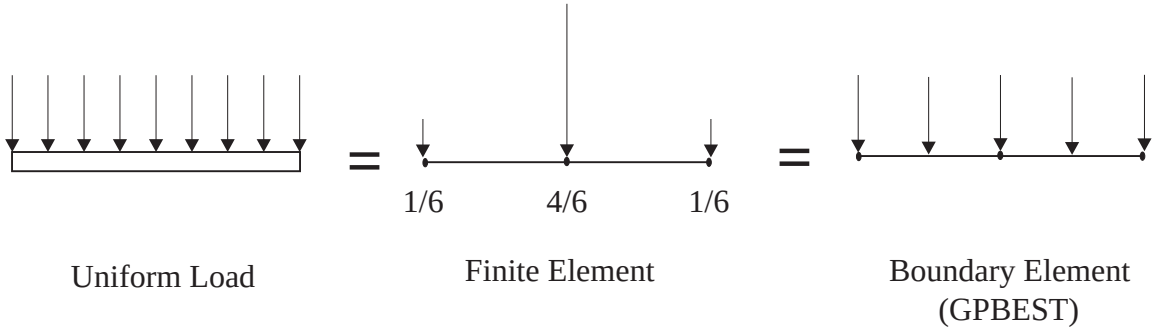
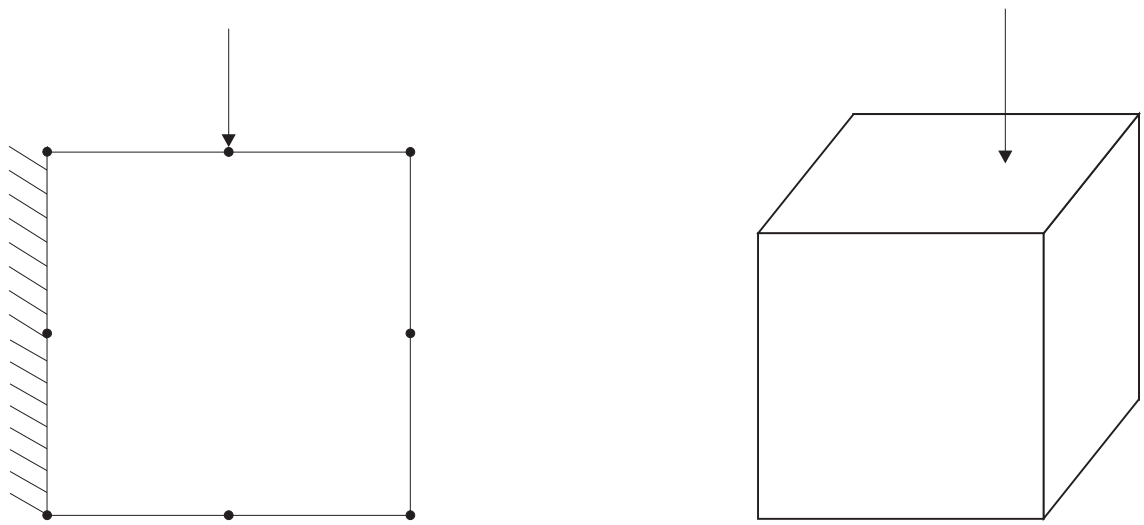


Figure 5.5.2 Applying a Uniform Pressure Over an Element

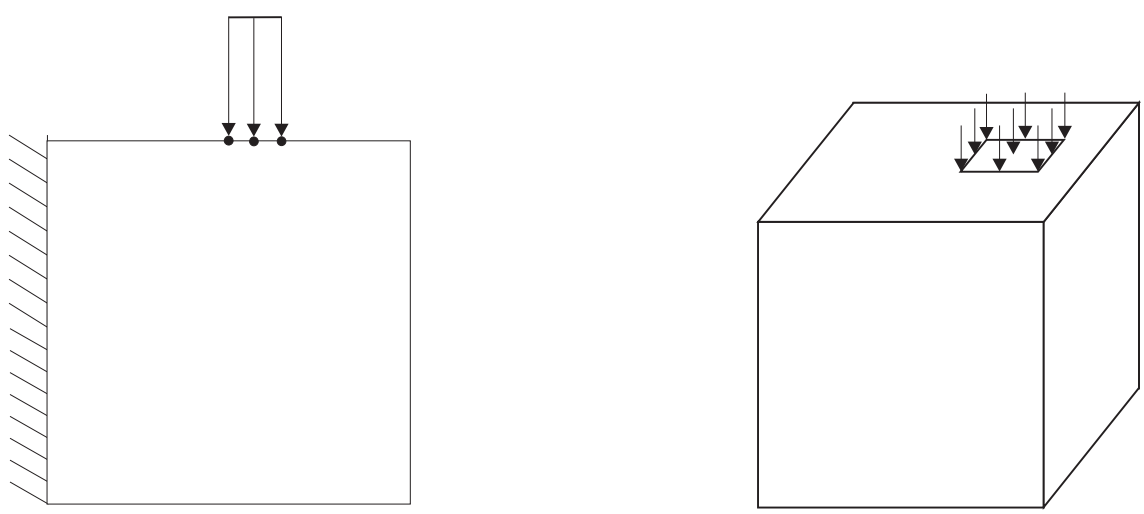
Frequently, an engineer will want to idealize a given load and represent it as a concentrated force in a model. This is often done with finite element models. Figure 5.5.3 illustrates a concentrated force at the edge of a 2-D and a 3-D region. It is important to realize that physically, there is no such thing as a concentrated force. Forces are always *distributed*. Even a needle on a table top results in a distributed force, where the load is distributed over the small area of needle and table contact.



(a) (b)

Figure 5.5.3 Theoretical Concentrated Load at Boundary
(a) 2-Dimensional Model (b) 3-Dimensional Model

GPBEST does not allow concentrated forces on the edges. In order to simulate a concentrated force, the user should distribute the given load over the length of a boundary element. This more closely resembles the physical problem. Figure 5.5.4 (a) shows a distributed load over a short length of a 2-D model, while Figure 5.5.4 illustrates the 3-D case.

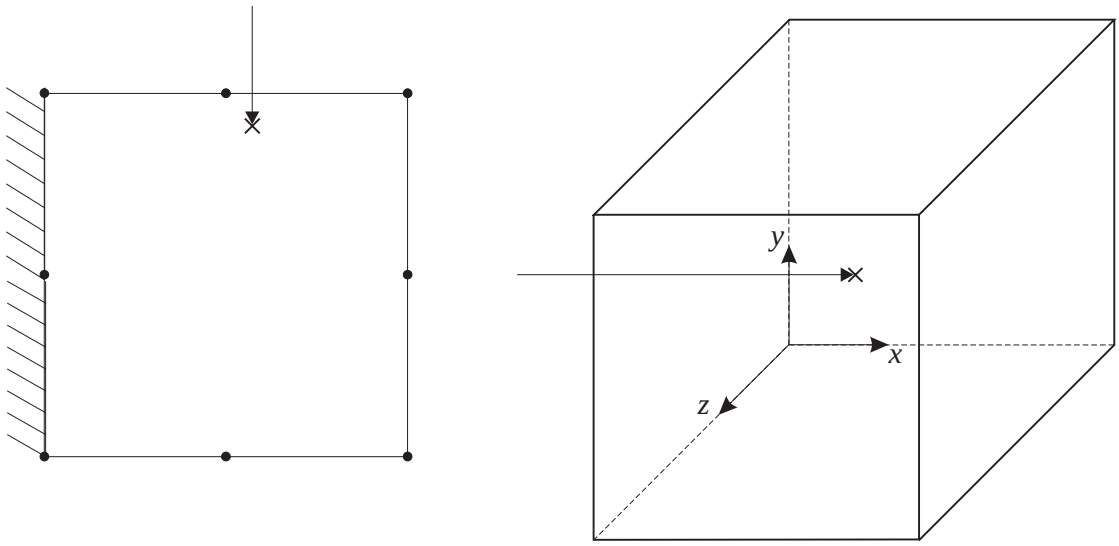


(a) (b)

Figure 5.5.4 Distributing the Load Over a Small Boundary Element
(a) 2-Dimensional Model (b) 3-Dimensional Model

Another capability that exists in **GPBEST** is a concentrated force $inside$ a region. This is not specified in the **BCSET data block, but rather in the **BODY data block with the CONCentrated

card. (See Section 5.6 for more information on the CONCentrated card.) Figure 5.5.5 illustrates a concentrated force inside 2-D and 3-D regions.



(a) (b)

Figure 5.5.5 Concentrated $Force$ Inside Regions
(a) 2-Dimensional Model (b) 3-Dimensional Model

At any rate, the user should not expect to get accurate stress/strain results near a concentrated force or near a sharply varying load field. Near the force, the solution is changing rapidly and the results are very sensitive. Further away from the load field, the results are much more stable.

Another basic concept that must be explained is the distinction between ‘global’ and ‘local’ boundary conditions. Global boundary conditions are the default in **GPBEST** , and are very straight forward. The global directions are x, y, z for 3-D analysis, x, y for 2-D analysis, and r, z for axisymmetric analysis. These are demonstrated in Figure 5.5.6. To specify a boundary condition (constraint or traction) in a global direction, you merely specify the magnitude as components in the global directions.

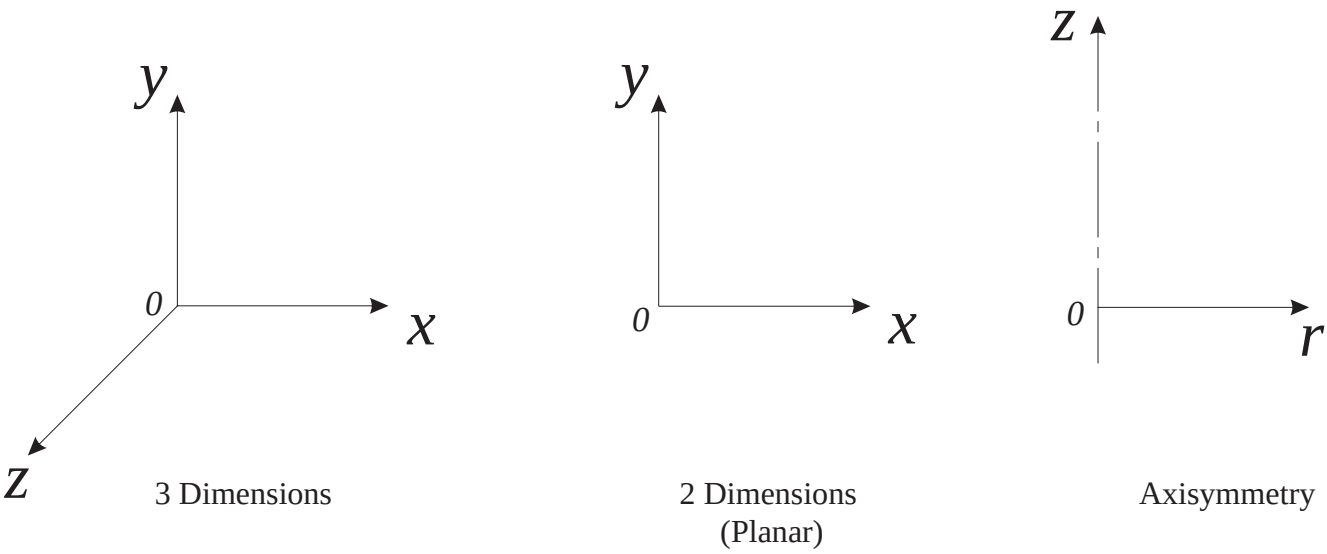


Figure 5.5.6 Global Boundary Condition Coordinate Directions

Local boundary conditions are introduced with the LOCAl card. The first local coordinate direction is in the direction of the outward element normal. For instance, Figure 5.5.7 illustrates the local coordinate direction for a 2-D and a 3-D boundary element.

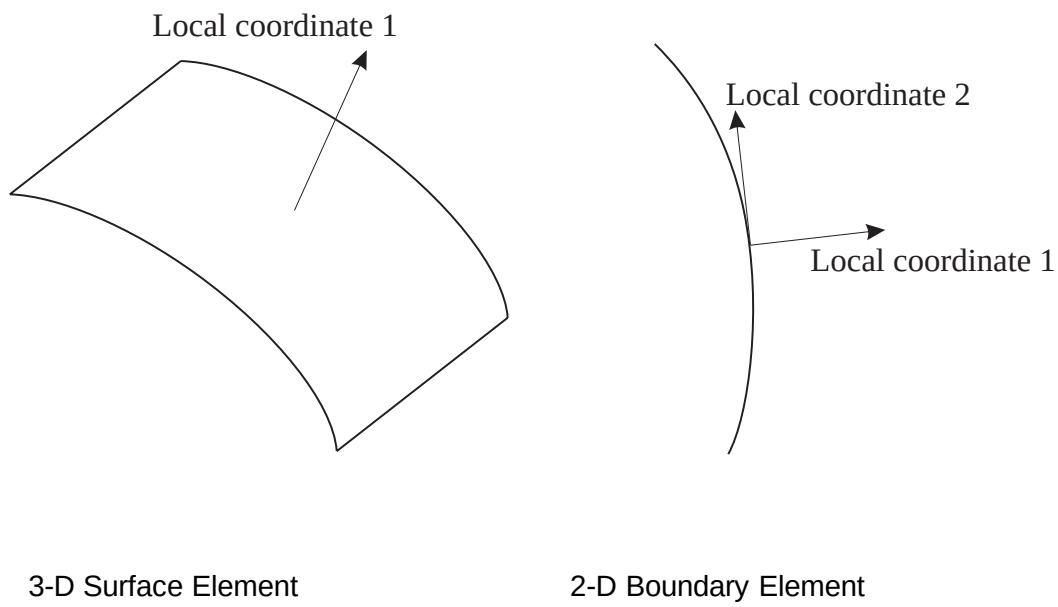


Figure 5.5.7 Local Boundary Condition Coordinate Directions

Rigid Body Constraints

An important consideration in preparing **GPBEST** models is the restraint of the model against what is known as ‘rigid body’ motion. It is perhaps easiest to illustrate this concept by referring to a

stress analysis problem. If a **GPBEST** stress analysis model is constructed but is not 'tied down,' then the entire model can satisfy an infinite number of equilibrium solutions. Not being 'tied down,' the model can fly off into space under the action of the slightest load. This is illustrated in Figure 5.5.8a. Figure 5.5.8b illustrates the same problem where the rigid body motion has been removed through proper application of boundary conditions. Mathematically, rigid body motion involves a singular system matrix which cannot be uniquely solved. In this case, **GPBEST** will continue to run the problem; but the primary degrees of freedom (i.e., displacements) will be incorrect. If the problem runs to completion, the tractions, stresses, strains, etc. may be correct.

No Constraint

(a)

Proper Constraint

(b)

Figure 5.5.8 (a) Model Not Constrained, (b) Model Constrained

How can it be determined whether rigid body modes have not been properly accounted for? The first thing that may be noticed is that the operating system issues a warning message such as "System issued IEEE error: Inexact, Underflow ..." This is a good indication that a singularity was encountered while running the problem.

Another habit the **GPBEST** user should adopt is to check the LOAD BALANCE section in the **GPBEST** results file. The following is a typical load balance output for a model that was not constrained in the Y-direction:

```
TOTAL LOAD BALANCE AND APPLIED LOAD BALANCE:

                        X              Y              Z
  TOTAL BALANCE  -0.19621E+00    0.20000E+01    0.78523E+00
   APPLIED LOAD   0.00000E+00    0.20000E+01    0.00000E+00
  PERCENT ERROR   0.98107E+01    0.10000E+03    0.39262E+02

THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS
CALCULATED WITH RESPECT TO THE
MAGNITUDE OF VECTOR OF APPLIED LOAD BALANCE =     0.20000E+01
```

The above output indicates a 100 percent error in the Y-displacement, a sure sign that something is wrong.

Another indicator that rigid body modes have not been accounted for is when very large displacement results appear in the output. For instance, an X- displacement of 8.45837956E+23

is ridiculously large, and is a sure indication that something is wrong with the model. However, boundary conditions may not be the only source of large values; improper material property values are another common cause. For instance, if the elastic modulus of steel is specified as 3.E-7 instead of 3.0E+7, very large numbers can result. Therefore, it is always wise to check for these types of errors from a number of sources. The following is a sample of the displacement output in the **GPBEST** result file. Note that the Y-displacement of node 1 is very large, indicating a possible problem. Note also that the stresses all have reasonable values:

```
    JOB TITLE:    BRACKET EXAMPLE - brac2.dat
    NODAL OUTPUT AT TIME =      1.000000      FOR REGION = GMR1

NODE       DISPLACEMENT    --------- STRESS ---------
              X/Y/Z          XX/YY/ZZ       XY/XZ/YZ
-------------------------------------------------------
  1         0.10750E+02   -0.13831E+02    0.76223E+01
           -0.83628E+08   -0.90718E+00   -0.13009E+01
           -0.16492E+03   -0.27411E+01    0.22111E+01

  2         0.66802E+01    0.20795E+01   -0.17487E+01
           -0.83628E+08    0.78333E+00   -0.18506E+01
           -0.15113E+03   -0.39437E+01    0.00000E+00
```

For the most part, the analyst does not want rigid body motion. The exception would be in the case of free vibration problems, where the rigid body modes are valid considerations. These are known as ‘‘free-free’’ boundary condition problems, and must be handled differently in **GPBEST** . However, for most other problems, the rigid body modes must be suppressed through proper boundary conditions.

Usually, if proper boundary condition constraints are applied to a model, this will be sufficient to guard against rigid body motion. However, sometimes the analyst must add additional constraints to suppress the rigid body motion. Figure 5.5.9 shows a 3-D model of a simply supported beam, with a uniform pressure load in the $-y$ direction. It is desired to find the displacement in the $x-y$ plane. The motion in the z-direction is immaterial. If the analyst specified boundary conditions at the two ends that only constrained the x and y motion, **GPBEST** could give incorrect results. Even though the motion in the z-direction is not a concern, it must be suppressed nevertheless in order to prevent rigid body motion in the z-direction.

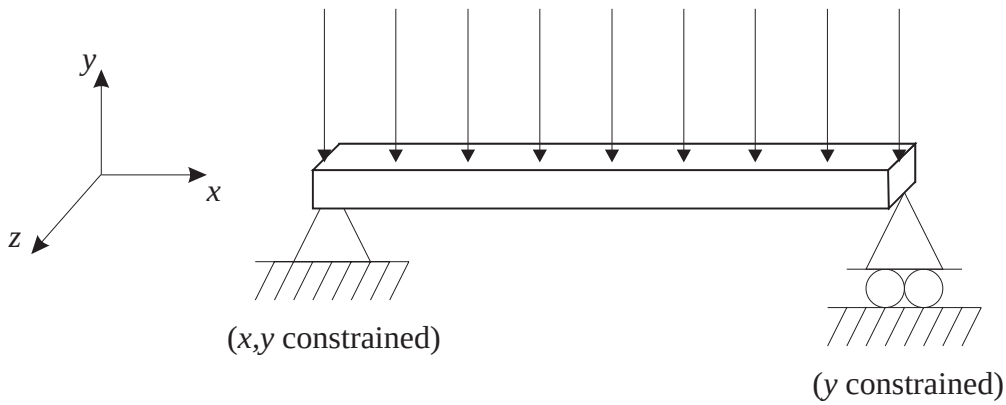


Figure 5.5.9 Rigid Body Mode Unconstrained in z-Direction

If there is symmetry in a model, then this can be used to partially constrain the rigid body motion. Symmetry edges and planes can be explicitly modeled and constrained through proper selection of boundary conditions. However, a better method would be to use the SYMMetry card in **CASE. When using the SYMMetry card, the symmetric edges do not need to be modeled with elements. For example, the 2-D model in Figure 5.5.10 only needs to be constrained at any node in the y-direction. The symmetry condition prevents translation in the x-direction and rotation about the z-direction.

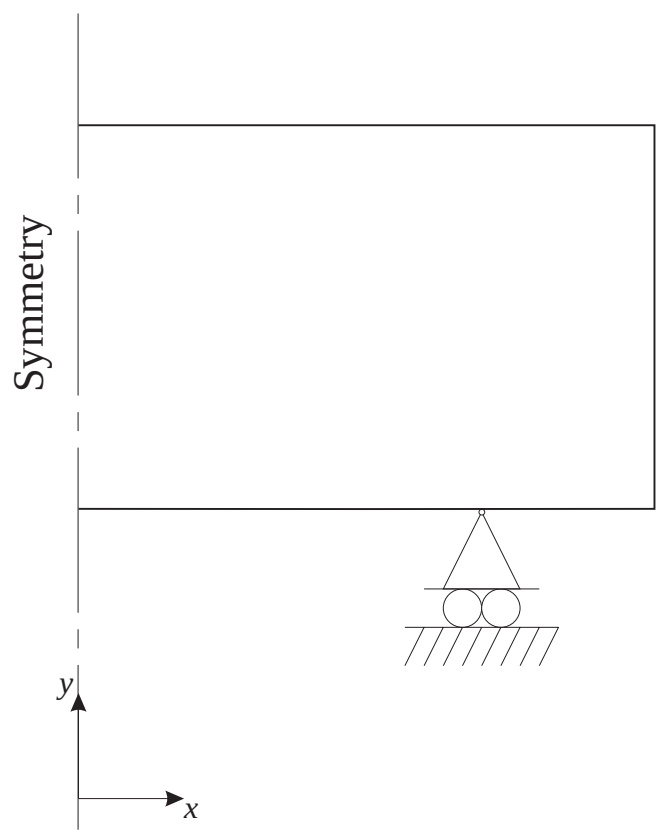


Figure 5.5.10 Constraining a Symmetric Model

Definition of Boundary Conditions

Each kind of analysis requires special consideration of proper rigid body constraint. For instance, in a stress analysis problem, the model must be constrained against rigid body motions in each translation direction. However, possible rotational modes must be considered as well. Figure 5.5.11 shows a 2-D model constrained in the x and y directions, but free to rotate about node 1. In this case, another boundary condition must be imposed on at least one of the other nodes. For instance, the x direction could be constrained at node 5, thus preventing model rotation.

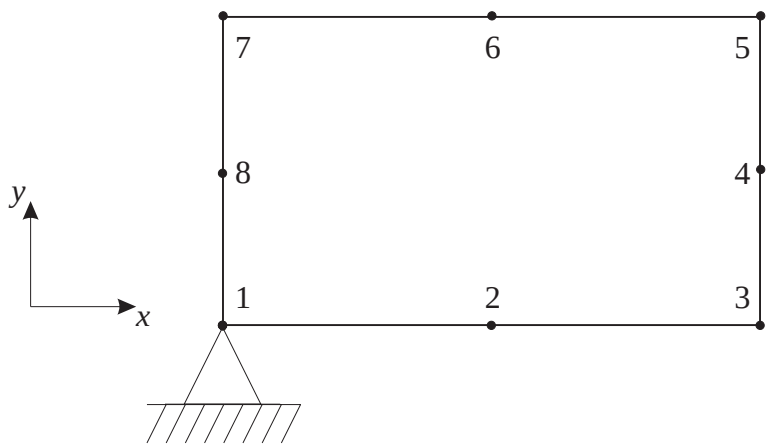


Figure 5.5.11 2-D Model Not Constrained to Rotate About z Axis

In 3-D stress analysis, possible XYZ translations and XYZ rotations must be avoided. In a planar 2-D stress analysis, the xy translations and rotation about the z axis direction must be constrained. Many analysts occasionally forget to apply the translation constraints while applying numerous traction conditions.

The case of an axisymmetric model is a little different. **GPBEST** automatically imposes a condition of zero radial displacement at the origin. Therefore, the radial displacement does not need to be set to zero at the origin or at any other point for the purpose of preventing the arbitrary rigid body motion in the radial direction. However, the axial direction (z) must still be fixed at some point in order to prevent rigid body translation in this direction. This is illustrated in Figure 5.5.12, where the axial rigid body motion has been suppressed.

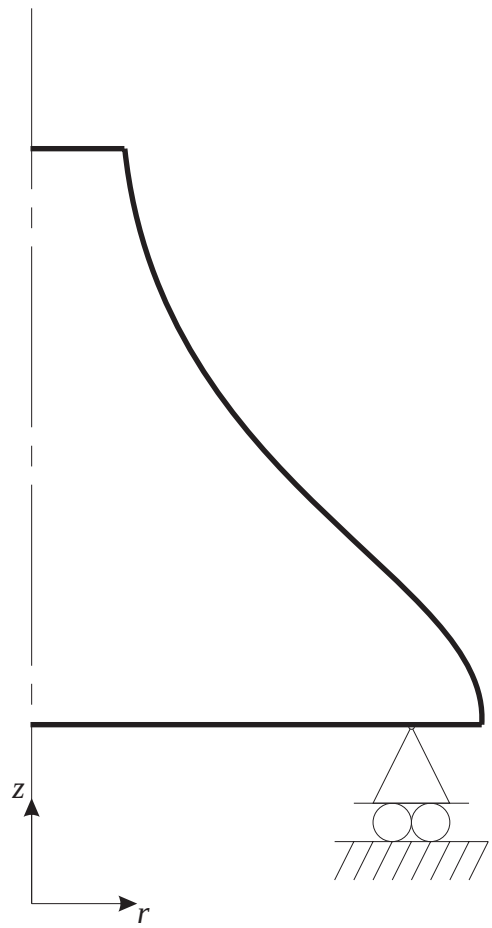


Figure 5.5.12 Constraining the Axial Motion in an Axisymmetric Model

Models using cyclic symmetry are also automatically prevented from moving in the radial direction. However, rigid body translation along the cyclic axis as well as rigid body rotation about the same axis is not prevented by invoking the CYCLic option. Consequently, these motions must be constrained by the user. The user must be careful since further boundary conditions are not allowed to be specified on the cyclic interfaces.

In the case of heat transfer (or other potential analyses), the primary degree of freedom is called ‘temperature’ or ‘potential.’ Somewhere in the model, the temperature must be explicitly defined through a boundary condition. Otherwise, an infinite number of valid solutions could result. Explicitly defining temperature for at least one node in the model is analogous to ‘tying down’ a stress analysis model. A particular situation to watch out for is illustrated in Figure 5.5.13, where heat flux is specified on all edges. This is not sufficient to constrain the temperature, and **GPBEST** will likely produce incorrect temperatures while the heat fluxes may be correct.

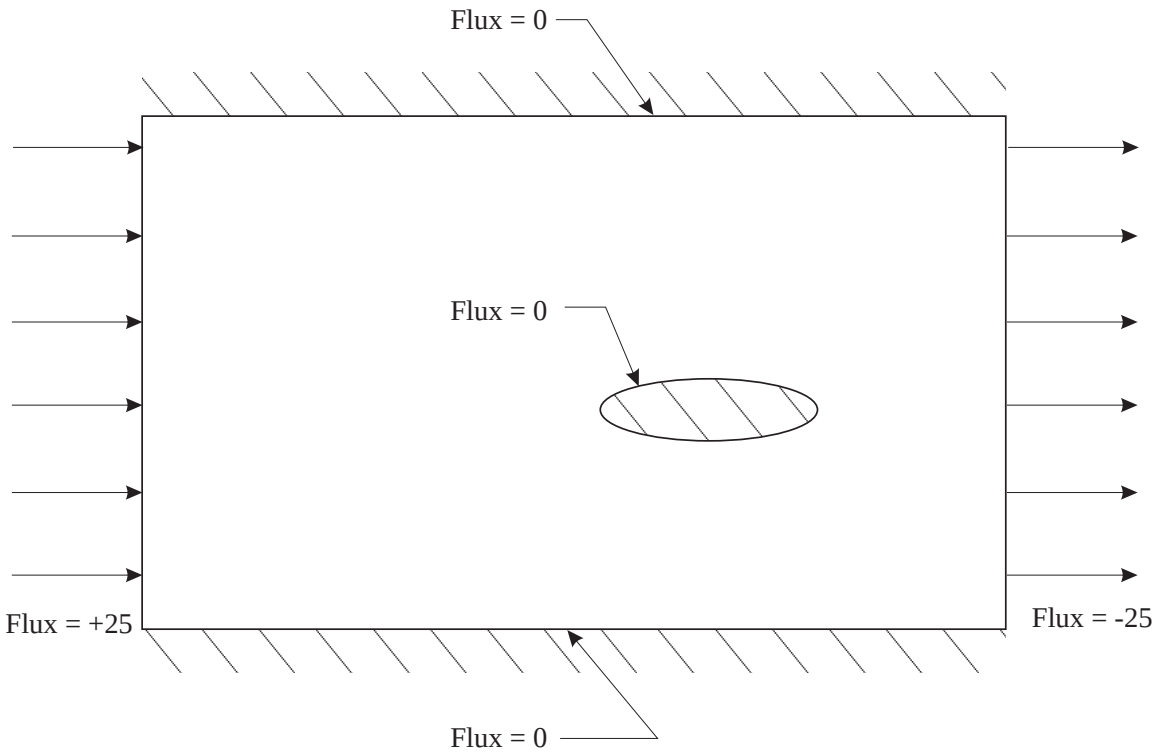
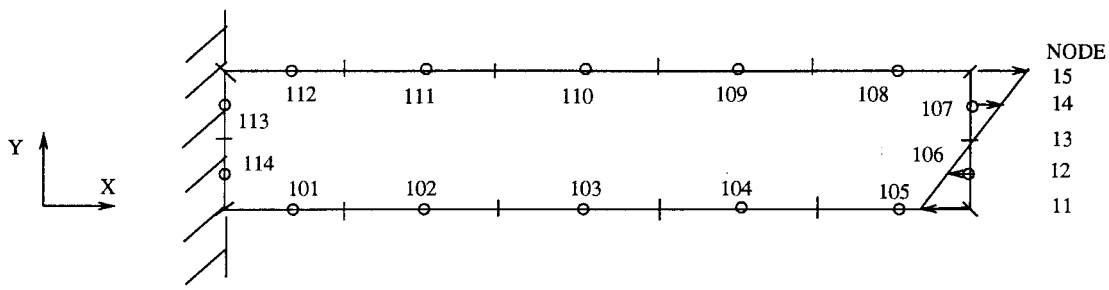


Figure 5.5.13 Heat Transfer Problem With a ‘‘Rigid Body’’ Mode

EXAMPLES OF BOUNDARY CONDITIONS

Some examples of the application of boundary conditions for a specific class of problems are presented in this section.

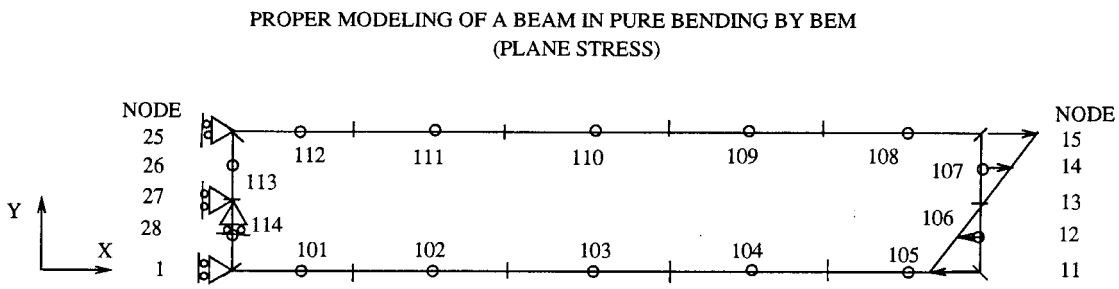
IMPROPER MODELING FOR A BEAM IN PURE BENDING BY BEM
(PLANE STRESS)




```
**BCSET
  ID-Bcset BC001
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  113  114
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1  0.0
  DISPlacement-bc 2
    SPLIst ALL
    T-VAlue 1  0.0

**BCSET
  ID-Bcset BC002
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  106  107
  TRACtion-bc 1
    SPLIst  11  12  13  14  15
    T-VAlue 1  -100.0  -50.0  0.0  50.0  100.0
```

Note: **GPBEST** will produce correct results for the boundary conditions specified above. However, these results will deviate (when $\nu \neq 0.0$) from the results from beam theory. The fixed constraints in the y-direction will produce a resulting shear traction which is non-zero at the top and bottom edges. This is contrary to the beam theory. This shear will have an effect on the deformation near the fixed end and introduce a slight rotation, altering the results even for a long, slender beam.

Also note that quartic elements give better results for beam problems than do quadratic elements.




```
**BCSET
  ID-Bcset BC001
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  113  114
  POINts-bc   1   27   25
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1  0.0

**BCSET
  ID-Bcset BC002
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  113  114
  POINts-bc  27
  DISPlacement-bc 2
    SPLIst  ALL
    T-VAlue 1  0.0

**BCSET
  ID-Bcset BC003
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  106  107
  TRACtion-bc 1
    SPLIst  11  12  13  14  15
    T-VAlue 1  -100.0  -50.0  0.0  50.0  100.0
```

Note: The boundary conditions shown above will produce results which are very close to the results obtained from beam-bending theory. The roller boundary conditions allow the beam to expand vertically such that the shear tractions do not develop at the top and bottom edges of the vertical end of the beam.

Results for a beam subjected to shear tractions at the free end are generally difficult to reproduce using the boundary element method, especially by lower order elements, since it requires a

parabolic shape shear traction across the free end and the variations of displacements along the span are cubic. It is, therefore, necessary to apply the correct boundary conditions and use appropriate variations of displacements along the span to obtain accurate results. As seen in the second illustration, deformation is fixed only at one node in the vertical direction (generally, it is recommended in larger problems that a greater number of nodes be restrained) and hence, the proper shear traction does not develop. On the other hand, the completely clamped beam of the first illustration will produce non-zero shear tractions at the top and bottom but should only be used to model long, slender beams with Poisson's ratio equal to zero.

Although it may be defeating the purpose of the analysis, correct results may be obtained by using boundary conditions similar to the second illustration and imposing the proper shear traction distribution at the fixed end, instead of fixing it (See Reference 123).

5.5.1 BOUNDARY CONDITION SET CARD

**BCSET

Status - REQUIRED

Full Keyword - **BCSET**

Function - Identifies the beginning of a boundary condition data set.

Input Variables - NONE

Additional Information -

Boundary condition data sets may be input as often as is required. Each must begin with this card.

The boundary condition data sets must follow all GMR-bc and INTErface definitions, and must precede any **BODY data.

Examples of Use -

1. Fix the normal (local) displacement of all elements on surface SIDE1 of GMR REG2 for all time. (No TIMEs-bc card is required.)

```
**BCSET
  ID-Bcset U1FIX
  VALUe
  LOCAl
  GMR-bc REG2
    SURFace-bc SIDE1
    DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0
```

5.5.2 BOUNDARY CONDITION IDENTIFICATION

ID-Bcset NAME [NODAL]

Status - OPTIONAL

Full Keyword - **`ID-Bcset`**

Function - Defines the identifier for the current boundary condition data set.

Input Variables -

NAME (Alphanumeric) - REQUIRED

User-specified name for the current data set.

NODAL - OPTIONAL. The qualifier NODAL identifies that the subsequent data set would describe boundary specifications over nodes. This is valid while running periodic acoustics using the INDIRECT boundary element method and structural-acoustics problems.

Additional Information -

If the user does not include this card, then an id NAME 'BCIDxxxx' will be assigned by **GPBEST** , where xxxx is a unique integer number.

It is recommended (but not necessary) that the NAME 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 occurring within the NAME are not permitted.

The NODAL qualifier may be associated with the following analysis types:

1. Acoustic analysis using INDIRECT boundary element formulation where pressure drop is to be specified.
2. Structural-acoustics problems using the INDIRECT BEM where structural constraints and loads are defined on nodes as well as acoustic pressure-drop.

Examples of Use -

1. Define a set of displacement-type boundary conditions with the name DISP1.

```
**BCSET
  ID-Bcset  DISP1
  GMR-bc GMR1
  SURFace-bc  SURF1
  ELEMents-bc  104
  POINts-bc 108
  DISPlacement-bc 1
  SPLIst 108
  T-VAlue   1   0.0
```

2. Define a set of global forces over nodes of an FE structural GMR.

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  POINts-fem   47   49  51
  FORCe-fem  3
    T-VAlue   1   100.0
```

5.5.3 IDENTIFICATION OF BOUNDARY CONDITION TYPE

VALUe

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 VALUe, RELAtion, VARIable or MOVIng do not appear in a boundary condition set, VALUe is assumed.

For complex loading, both real and imaginary part must be input.

Examples of Use -

1. Used here to indicate that the value of a local traction type of boundary condition is specified.

```
**BCSET
  ID-Bcset  TRAC12
  VALUe
  LOCAl
  GMR-bc GMR1
  SURFace-bc SURF1
  ELEMents-bc 17
    TRACtion-bc 1
  SPLIst ALL
    T-VAlue 1   -100.0
```

RELAtion

Status - OPTIONAL

Full Keyword - **RELAtion**

Function - Identifies the boundary condition as one which will define a relationship between field variables (e.g. spring, convection or impedance 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, CONVection and IMPEdance boundary conditions.

If VALUe, RELAtion, VARIable or MOVIng do not appear in a boundary condition set, VALUe is assumed.

Examples of Use -

1. The RELAtion card is used in the following example to indicate specification of convection type of boundary condition.

```
**BCSET
  ID-Bcset  BCS1
  RELAtion
  GMR-bc  GMR1
  SURFace-bc  SURF1
    ELEMents-bc   1   2   3   4
    CONVection  1.26   -100.0  $ H= 1.26, TEMP(AMBIENT)= - 100.0
```

2. A spring boundary condition in a plane stress problem is specified using the RELAtion card.

```
**BCSET
  ID-Bcset SPRING1
  RELAtion
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc   5
    SPRIng  2  1180     $ Y-DIRECTION      K=1180 N/(m*m)
```

3. Impedance boundary conditions are specified using the RELAtion card.

```
**BCSET
  ID-Bcset IMPE11
  RELAtion
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  11  12
    IMPEdance  53.2       $ Real value only
```

VARIable

Status - OPTIONAL

Full Keyword - **VARIable**

Function - Identifies the boundary condition as one which will define the numerical values of field variables for SAVE-gmr SOLVER and RESTart-gmr SOLVER analyses.

Input Variables - NONE

Additional Information -

This card must not be used for relational boundary condition sets.

This card is only applicable for elastostatic, steady-state heat conduction and steady-state concurrent thermoelasticity.

The keyword **BCSET VARIable card is used in conjunction with SAVE-gmr SOLVER and RESTart-gmr SOLVER case control options when for a series of analyses, only the numerical values of the boundary condition field variables changes. Include the VARIable card in each **BCSET for which the values may change during subsequent analyses.

If VALUe, RELAtion, VARIable or MOVIng do not appear in a boundary condition set, VALUe is assumed.

Examples of Use -

1. On the initial run, the SAVE-gmr SOLVER option is used for subsequent restarts. Here, the VARIable card is used to flag a boundary condition that will be changed on the restart run. For clarity, both the **CASE and **BCSET data blocks are displayed.

```
**CASE
  TITLe   3-D HEAT TRANSFER - INITIAL RUN
  HEAT
  RESTart WRITE
  SAVE SOLVER

**BCSET
  ID-Bcset TEMP12
  VARIable
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  9
    POINts-bc  13  15  18  16  95
  TEMPeratures-bc
    SPLIst  13  15  18  16  95
    T-VAlue 1    0.3 0.8 0.8 0.3 0.5
```

2. On the subsequent restart run, the temperature boundary conditions are doubled.

```
**CASE
  TITLe   3-D HEAT TRANSFER - RESTART RUN
  HEAT
  RESTart SOLVER

**BCSET
  ID-Bcset TEMP12
  VARIable
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  9
    POINts-bc  13  15  18  16  95
  TEMPeratures-bc
    SPLIst  13  15  18  16  95
    T-VAlue 1    0.6 1.6 1.6 0.6 1.0
```

MOVIng

Status - OPTIONAL

Full Keyword - **MOVIng**

Function - Identifies the boundary condition set as one which will define a moving heat source.

Input Variables - NONE

Additional Information -

The keywords VALUe, RELAtion, VARIable and LOCAl are not valid for moving heat source boundary condition specifications.

A MOVIng heat source **BCSET specifies the passing of a point heat source along an axisymmetric surface element. A separate BCSET is required for each pass of the source across each element.

The MOVIng heat source **BCSET should include the definition of the GMR-bc, SURFace-bc, ELEMents-bc and POINts-bc involved.

The MOVIng **BCSET is only available for axisymmetric analysis. It is not available for 2-D and 3-D analyses.

The MOVIng heat source is only available for transient HEAT and CTHErmal analyses.

A positive value for the heat source implies that heat is entering the region at the moving source point.

Examples of Use -

1. Specify a moving heat source condition of constant strength. Notice how the heat source moves from one element to the next.

```
**BCSET
  ID-Bcset FLUX101
  MOVIng
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  10
    POINts-bc    101  102  103
  SOURce
    TIMEs-moving    0.00  2.50  5.00
    ENERgy-rate    6.2832   6.2832   6.2832
**BCSET
  ID-Bcset FLUX102
  MOVIng
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  11
    POINts-bc    103  104  105
  SOURce
    TIMEs-moving    5.00  7.50 10.00
    ENERgy-rate    6.2832   6.2832   6.2832
```

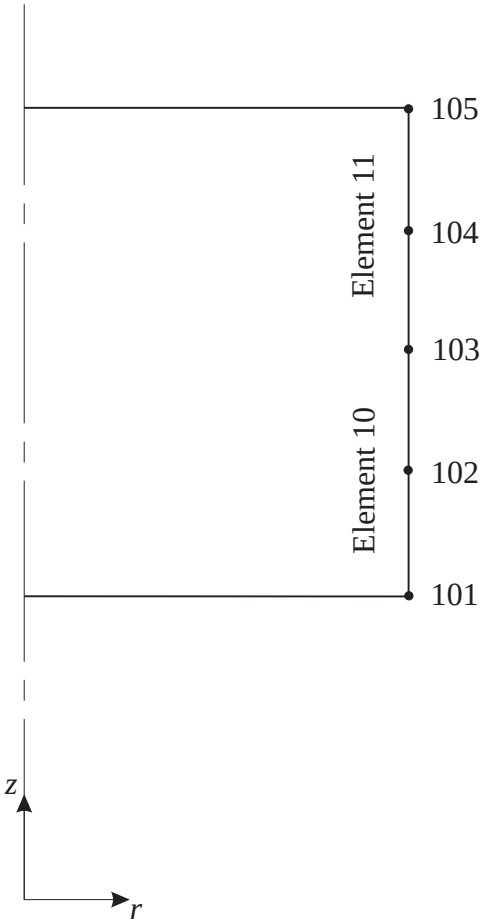


Figure for MOVIng Card: Axisymmetric Model

INTErface

Status - OPTIONAL

Full Keyword - **`INTErface`**

Function - Identifies that the current boundary condition is to be applied to elements contained in an interface definition.

Input Variables - NONE

Additional Information -

This card allows TRACtion-bc, FLUX-bc, TORQue-bc, and LOAD-bc boundary conditions to be applied to elements and nodes that are also contained in an interface definition. Other types of boundary conditions (DISPlacement-bc, TEMPeratures-bc, SPRIng, RESIstance, etc.) are not allowed.

Both time-independent and time-dependent boundary conditions are supported with this INTErface card.

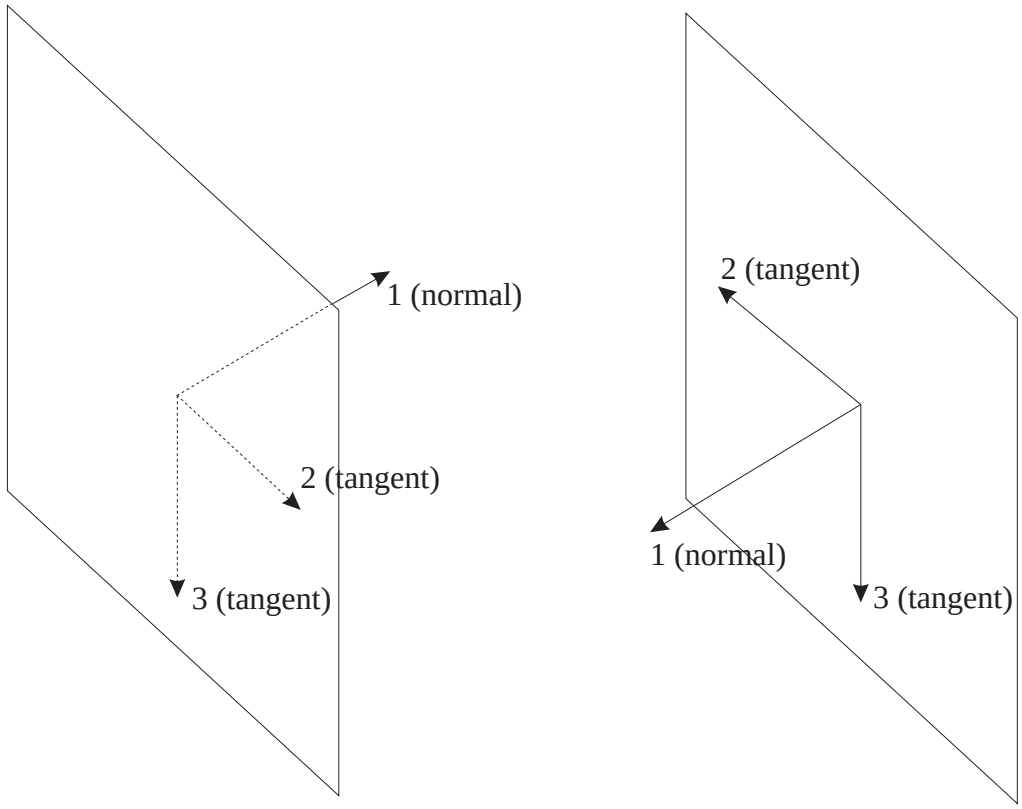
TRACtion-bc and LOAD-bc boundary condition sets containing the INTErface card must be defined in a local coordinate system. See the LOCAl definition on the next card. LOCAl should not be specified for TORQue-bc.

If INTErface and ‘‘LOCAl EXPLICIT’’ appear in the same **BCSET, then the first local component direction must be normal to the surface. It is recommended that ‘‘LOCAl NORMAL’’ be used whenever possible.

If a boundary condition is to be applied to elements on both sides of the interface (e.g., pressure between two regions) then two **BCSETs are required, i.e., one for each region. If the LOCAl card is used as required in the case of TRACtion-bc or LOAD-bc boundary conditions on an interface, then the LOCAl coordinate system used in the two **BCSETs must have the same orientation. However, the corresponding local components may point in opposite directions (see figure below). If ‘‘LOCAl NORMAL’’ is specified without using the DIREction-bc card, then the local coordinates will automatically be oriented in the correct manner. If the user specifies a DIREction-bc card to orient the local tangent directions, then the vector specified in the second **BCSET should be parallel (but may point in opposite directions) to the vector specified on the DIREction-bc card in the first **BCSET. If ‘‘LOCAl EXPLICIT’’ is specified then the local coordinates specified in the two sets should be oriented with the first (outward normal) component parallel and pointing in the opposite direction and the tangential components parallel to their counterpart.

When TRACtion-bc or LOAD-bc boundary conditions are specified for nodes on both sides of an interface, the local coordinate system on each side must have their respective components parallel to one another. However, they may or may not be pointing in similar directions (see figure below). It is important to keep in mind that the direction of the traction or load of a specified component correspond to the direction

of the component of the local coordinate system for the node of the GMR-bc specified in the **BCSET. Specification of the component in the adjacent GMR must be with respect to the local coordinate system specified in the **BCSET for the nodes of that GMR.



Local coordinates for a surface point in Region 1

Local coordinates for a surface point in Region 2

Figure for Boundary Condition INTErface Card

Examples of Use -

1. Apply a pressure of 100.0 psi between two surfaces contained in a frictional contact interface definition.

```
**INTERFACE
  GMR-1 GMR1
  SURFace-gmr-1 SURF1
  ELEMents-gmr-1  101  102  103
  GMR-2 GMR2
  SURFace-gmr-2 SURF2
  ELEMents-gmr-2  201  202  203
  CONTact
  FRICtion 0.3

**BCSET
  ID-Bcset INTTRAC1
  VALUe
  INTErface
  LOCAl NORMAL
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  101  102  103
    TRACtion-bc 1
    SPLIst ALL
    T-VAlue 1   -100.0

**BCSET
  ID-Bcset INTTRAC2
  VALUe
  INTErface
  LOCAl NORMAL
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc  201  202  203
    TRACtion-bc 1
    SPLIst ALL
    T-VAlue 1   -100.0
```

LOCAl ATYPE

Status - OPTIONAL

Full Keyword - **LOCAl**

Function - Indicates that input for the current boundary condition set will be defined using a local coordinate system.

Input Variables - NONE

ATYPE (Alphanumeric) - OPTIONAL

Allowable variables are NORMAL and EXPLICIT. If neither is specified then NORMAL is assumed by default.

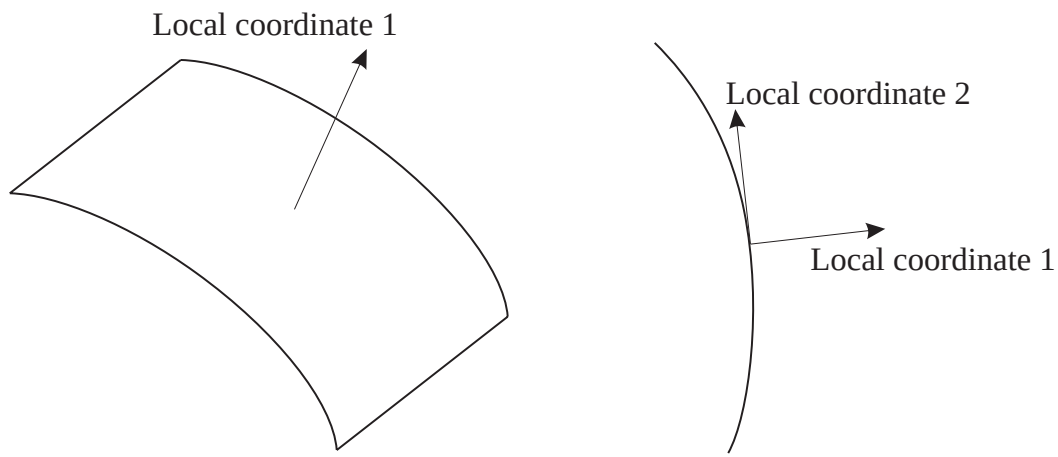
NORMAL -
Indicates that the first local component direction will be in the direction of the outer normal. In 2-D and axisymmetric analysis, the second local component is in the direction defined by the cross product of the out-of-plane vector with the normal vector. In 3-D analysis, the local components 2 and 3 can be assigned using the DIREction-bc card. See the figure on the next page.

EXPLICIT -
Indicates the local component directions will be explicitly defined for each source point in the boundary condition set.

Additional Information -

Local boundary conditions are not applicable to scalar problems (i.e. heat conduction; diffusion).

The ‘‘LOCAl NORMAL’’ card allows for specification of displacement, traction, or springs normal to a (not necessarily flat) surface. The first local component direction will be in the direction of the outer normal to the surface at each node. In 2-D and axisymmetric analysis, the second local component is in the tangent plane defined by the cross product of the out-of-plane vector with the normal vector (see the figure on the next page). In 3-D analysis the local tangent components 2 and 3 can be defined with the use of the DIREction-bc card (see next card). If the DIREction-bc card is not input under the LOCAl NORMAL card in a 3-D analysis, then components 2 and 3 must not be referred to in that boundary condition set.



3-D Surface Element 2-D Boundary Element

Figure for LOCAl Card: Local Coordinate Directions

If ‘‘LOCAl EXPLICIT’’ is input, then NODE cards (see the card following the next card) defining the local component directions must be input for every node in the boundary condition set immediately following this card.

If a node on a particular element has a local definition, then a specification in another direction on the **same node** over the **same element** must also be prescribed in the same local system. This is true even if a separate **BCSET is included. Note, however, another boundary condition on the **same node** but over a **different element** may be specified in either a global or local coordinate system as long as a different **BCSET is used for this purpose. The user should inspect the boundary results to verify that the boundary conditions have been correctly interpreted by **GPBEST** .

Only two distinct local displacement and/or spring boundary condition specifications are allowed at a 3-D Corner (see the Special Notes for Local Boundary Conditions below).

It is recommended that local boundary conditions be used only when a local definition becomes necessary. Excessive use of local boundary conditions is often a source of input data errors. For this reason, **it is recommended that local boundary conditions not be used when a global definition is feasible.** Finally, the user must carefully inspect the boundary results to ascertain that the boundary condition input has been properly interpreted by **GPBEST** .

This card is not intended for use in structural-acoustic analysis. For structural-acoustic analysis, the ‘‘LOCAl-fem FEM’’ card, described in Section 5.5.11, should be used.

Special Notes for Local Boundary Conditions -

Local boundary conditions on smooth or nearly-smooth surfaces:

Since shape functions are used to model surfaces, there are often discontinuities present in the normal vectors across adjacent elements of curved surfaces. The degree of discontinuity is dependent upon the refinement of the mesh. When local boundary conditions are used, these discontinuities, if not correctly accounted for, may have an adverse effect on the results by producing a component of traction tangential to the surface. In **GPBEST** , these adverse effects are avoided by averaging the normals across all adjacent elements specified by the user in the boundary condition set.

When a LOCAl boundary condition is applied to a group of elements representing a surface which is intended to be smooth, the entire group of elements over which the local boundary condition is applied should be contained in a single LOCAl boundary condition set. The normals of the adjacent elements in this **BCSET (and only the elements in this **BCSET) are then averaged, when the boundary condition is applied.

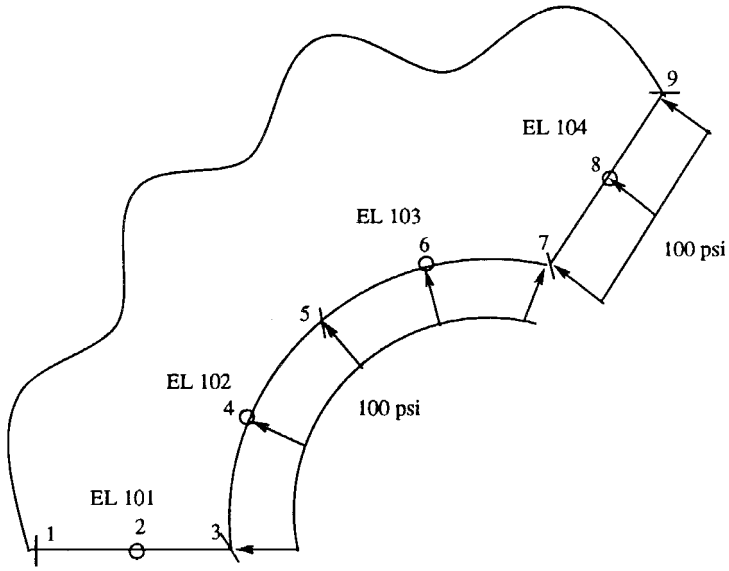
Local boundary conditions at corners:

In many circumstances however, the averaging of normals across adjacent elements is not desirable. For instance, when the actual surface is not smooth, such as near an edge (two surfaces) or corner (three or more surfaces). In such cases, if the two adjacent surfaces at the edge are put into two separate **BCSETs, the normals will not be averaged. This implies that two distinct, local coordinate systems are used at the edge. In 3-D analysis, **GPBEST** limits the number of distinct local displacement and/or local spring boundary condition specifications at the corner to two. Thus at a corner only two distinct local displacement (and/or local spring) coordinate systems may be used. However, a global system can be used to specify a third displacement (and/or spring) constraint at the corner. To avoid over-constraining corners, more than three displacement (and/or spring) constraints should be avoided regardless of the system selected. Furthermore, a local displacement coordinate system is automatically selected by **GPBEST** for elements on a not perfectly bonded interface (i.e., sliding, contact, interference fit, or spring interface). Therefore, if a not perfectly bonded interface intersects a corner of a 3-D GMR, only one additional local displacement (or spring) boundary condition can be specified on adjacent elements at the corner of each GMR. There is no limit to the number of traction boundary conditions that may be prescribed at a corner.

These two cases are illustrated in the diagrams on subsequent pages.

Examples of Use -

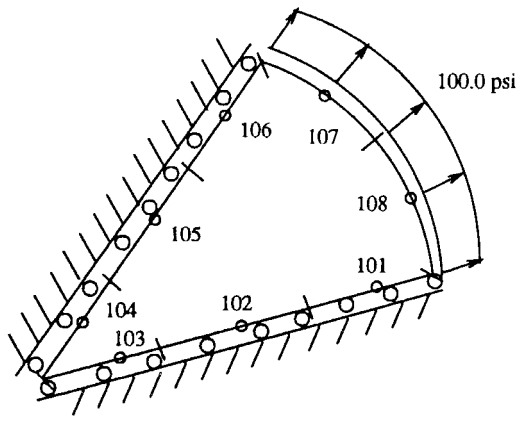
1. Discontinuous traction at Node 7.




```
**BCSET
  ID-Bcset BC001
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 102 103
  TRACtion-bc 1
  SPLIst ALL
  T-VAlue 1 -100.0

**BCSET
  ID-Bcset BC002
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 104
  TRACtion-bc 1
  SPLIst ALL
  T-VAlue 1 -100.0
```

Note: Since elements 102 and 103 are contained in the same boundary condition set, an average value of their normal vectors at node 5 is used in the application of the local boundary condition. This produces a traction with a continuous normal across the two elements. However, at Node 7, there exists an actual corner; so a discontinuous normal traction vector is desirable. This is accomplished by defining the local traction on element 104 in a **BCSET independent of element 103. See the Special Note on the LOCAl card for more information.

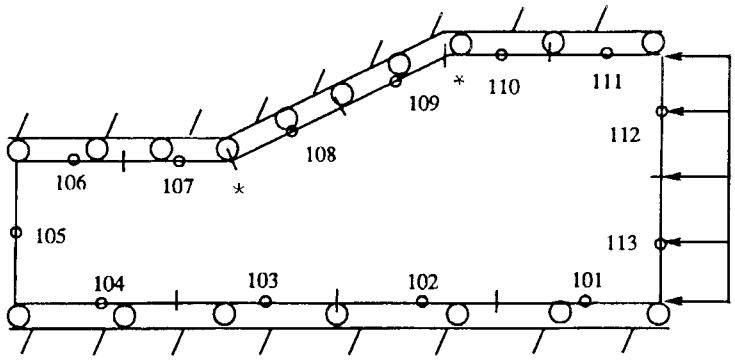
2. Local displacement constraints at corners.




```
**BCSET
  ID-Bcset BC001
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 101 102 103
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0

**BCSET
  ID-Bcset BC002
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 104 105 106
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0

**BCSET
  ID-Bcset BC003
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 107 108
  TRACtion-bc 1
  SPLIst ALL
  T-VAlue 1 100.0
```

Note: This example illustrates the proper technique for modeling local displacement constraints at corners. **GPBEST** will average normals of adjacent elements at the common node when the elements are contained in the same **BCSET. To model a fixed constraint at a corner using local boundary conditions, the adjacent elements at the corner should be put in separate **BCSETs. See the Special Note on the LOCAl card for more information.

3. Another example of local displacement constraints at corners.




```
**BCSET
  ID-Bcset BC032
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 106 107
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0

**BCSET
  ID-Bcset BC041
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 108 109
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0

**BCSET
  ID-Bcset BC015
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 110 111
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0
```

Note: This example also illustrates how the local displacement constraints can be modeled at the corners. However, in this problem the nodes marked with * will be fixed in two local directions. Hence, they will not move at all. If this is not what the user desires, this problem can be eliminated by placing the entire upper surface in the same **BCSET data block, as shown here:

```
**BCSET
  ID-Bcset BC038
  VALUe
  LOCAl
  GMR-bc REG1
  SURFace-bc SURF1
  ELEMents-bc 106 107 108 109 110 111
  DISPlacement-bc 1
  SPLIst ALL
  T-VAlue 1 0.0
```

DIREction-bc V1 V2 V3

Status - OPTIONAL (for use with ‘‘LOCAl NORMAL’’ in 3-D only)

Full Keyword - **`DIREction-bc`**

Function - Used to define the local component direction 2 and 3 tangent to the surface at every node in a ‘‘LOCAl NORMAL’’ boundary condition set.

Input Variables -

V1,V2,V3 (Real) - REQUIRED

Cartesian components of the vector used in defining the tangential directions.

Additional Information -

This card is only used in 3-D analysis when the ‘‘LOCAl NORMAL’’ card has been input. It is not required, however, if it is not included, then local components 2 and 3 must not be referenced to in a ‘‘LOCAl NORMAL’’ boundary condition set in a 3-D analysis.

This card is not applicable to 2-D or axisymmetric analysis. In 2-D and axisymmetric problems, the local tangent component 2 is uniquely defined. (See the LOCAl card.)

This card must immediately follow the ‘‘LOCAl NORMAL’’ card.

Using the DIREction-bc card with ‘‘LOCAl NORMAL,’’ the outer normal direction is the first component direction (component 1). The vector defined on this card is used as an aid in defining the local direction of component 3, tangent to the surface at each node. The normalized cross product of the outer normal at each node with the vector defined on this card defines the direction of component 3 at that node. The direction of component 2 is then the cross product of direction 3 with direction 1. It is important to keep in mind that the direction of the vector must not coincide with the normal direction of any of the nodes in the **BCSET in order for their cross product to be non-zero. A fatal error will result if the cross product at any node is zero.

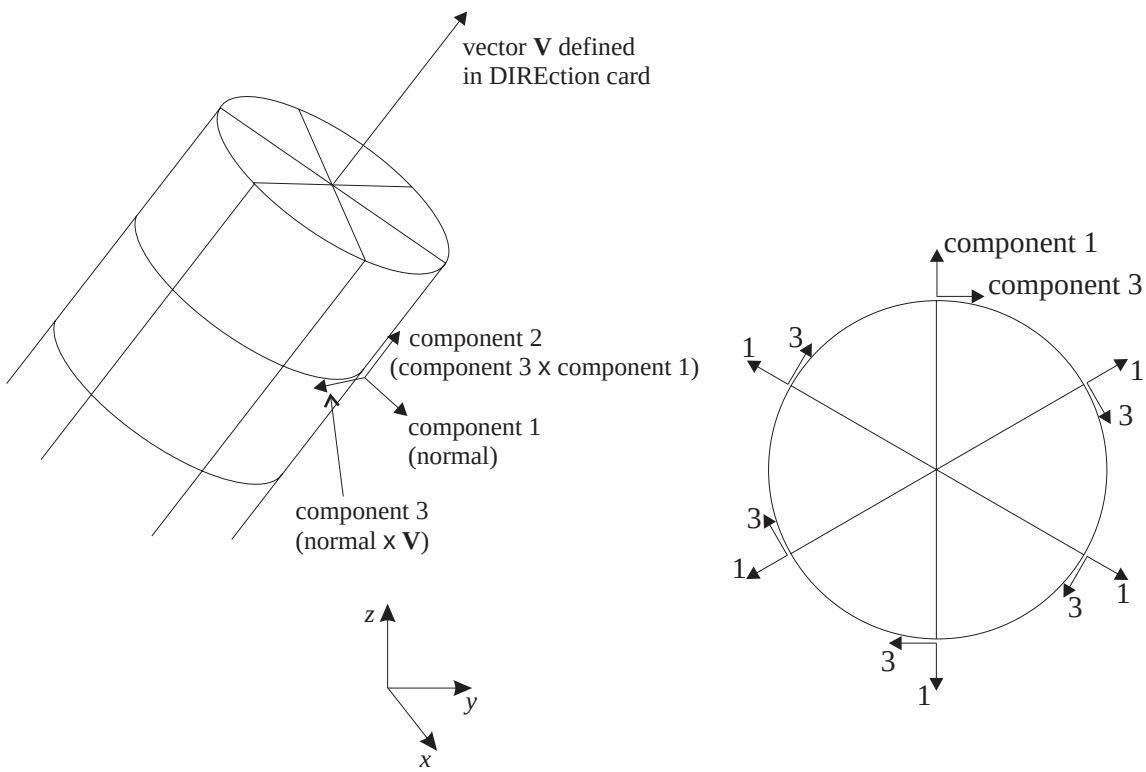
This card should appear only once in a **BCSET.

The vector input on this card does not have to be a unit vector. **GPBEST** will normalize this vector upon input.

Examples of Use -

1. As an example, we examine a shaft with its axis at an oblique angle to the global axis (see figure below). It would be very difficult to define a torque, an axial loading, or a pressure loading without the use of local boundary conditions. However, this can be easily accomplished by using the ‘‘LOCAl NORMAL’’ card along with the DIREction-bc card defining a vector **V** which is aligned with the axis of the shaft. Component 1 would be referenced to apply a pressure loading, component 2 will be referenced to apply a loading in the axial direction, and component 3 could be referenced to apply a torque loading on the side of the shaft. The **BCSET for a torque loading might look like this:

```
**BCSET
  ID-Bcset  TORQUE
  VALUe
  LOCAl NORMAL
  DIREction-bc  0.707  0.707  0.707
  GMR-bc  SHAFT
  SURFace-bc  S1
  ELEMents-bc  101   102   103   104   105   106
  TRACtion-bc  3
  SPLIst ALL
    T-VAlue  1  100.0
```



ISOMETRIC VIEW TOP VIEW

Figure for DIREction-bc Card: Shaft Model

(NONE) NODEID A1 A2 A3 B1 B2 B3

Status - OPTIONAL (REQUIRED if ‘‘LOCAl EXPLICIT’’ is input)

Full Keyword - **NO KEYWORD REQUIRED**

Function - Used to define the direction of the component in a ‘‘LOCAl EXPLICIT’’ boundary condition set.

Input Variables -

NODEID (INTEGER) - REQUIRED

User node number of the node for which the local coordinate system is defined.

A1,A2,A3 (REAL) - REQUIRED (A3 is required for 3-D analysis only)

Cartesian components of a vector whose direction defines the direction for the local component 1.

B1,B2,B3 (REAL) - REQUIRED (for 3-D analysis only)

Cartesian component of a vector. The vector (A1,A2,A3) defining the direction of local component 1 when crossed with this vector defines the direction of local component 3.

Additional Information -

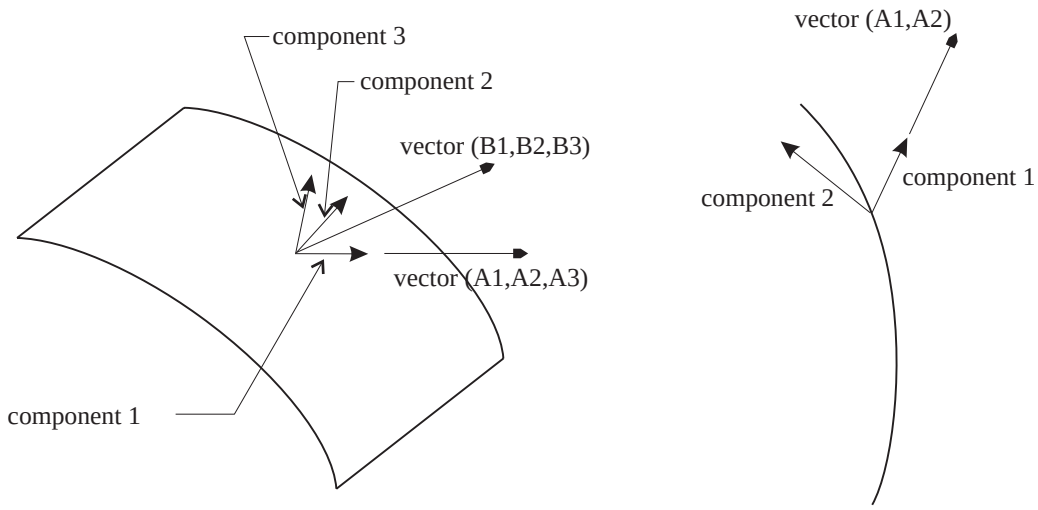
In 3-D analysis the direction of local component 2 is defined as the direction obtained from the cross product of the direction of local component 3 into local component 1. Note, if B1, B2, B3 is perpendicular to A1, A2, A3, then B1, B2, B3 defines the direction of local component 2.

In 2-D and axisymmetric analysis only variables A1 and A2 should be input. Since vector (A1,A2) must be in the modeling plane A3 must be zero, and since the out-of-plane direction can be thought of as direction 3, (B1,B2,B3) is not required. Load component 2 is defined by the direction obtained from the cross product of a vector in out-of-plane direction with the vector defining component 1.

If ‘‘LOCAl EXPLICIT’’ appears in a boundary condition set, then this card must be input for each user generated source point in the **BCSET, and immediately following the ‘‘LOCAl EXPLICIT’’ card.

If all source points in a ‘‘LOCAl EXPLICIT’’ **BCSET are to have the same local component directions, then this card only needs to be input once, and the keyword ‘‘ALL’’ should appear in place of the node number. See example 3 below.

The vectors input on this card do not have to be unit vectors. **GPBEST** will normalize these vectors upon input.



3-D Surface Element 2-D Boundary Element

Figure for NODE Card: Defining the DIREction-bc Vector

Examples of Use -

1. Define a 3-D ‘‘LOCAl EXPLICIT’’ **BCSET.

```
**BCSET
  ID-Bcset  TRAC12
  VALUe
  LOCAl EXPLICIT
    101     1.0   0.5   0.0     0.0   1.0   0.0
    102     1.1   0.5   0.0     0.0   1.0   0.0
    103     1.3   0.5   0.0     0.0   1.0   0.0
    104     1.2   0.5   0.0     0.0   1.0   0.0
     .             .                   .
     .             .                   .
     .             .                   .
    120      .0   1.0   1.0     0.0   0.0   1.0
    121      .0   1.3   1.0     0.0   0.0   1.0
  GMR-bc  TOP
  SURFace-bc  SURF1
  ELEMents-bc  1001   1002   1003   1004
  TRACtion-bc  1
  SPLIst  ALL
    T-VAlue  1  100.0
```

2. Define a 2-D ‘‘LOCAl EXPLICIT’’ **BCSET.

```
**BCSET
  ID-Bcset DISP10
  VALUe
  LOCAl EXPLICIT
    10   0.0   1.0
    11   0.1   0.9
    12   0.3   0.7
    13   0.6   0.4
    15   0.7   0.3
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc  101   102
  DISPlacement-bc 1
  SPLIst ALL
    T-VAlue  1  0.0
```

3. Define a 3-D ‘‘LOCAl EXPLICIT’’ **BCSET with all local components having the same direction.

```
**BCSET
  ID-Bcset DISP15
  VALUe
  LOCAl EXPLICIT
  ALL   0.707   0.707   0.0     -0.707   0.707   0.0
  GMR-bc  RING
  SURFace-bc  SIDE1
  ELEMents-bc  101   102   103   104
  DISPlacement-bc 1
  SPLIst ALL
    T-VAlue  1  0.0
```

5.5.4 DEFINITION OF SURFACE FOR APPLICATION OF BOUNDARY CONDITIONS

In the boundary element method, the primary load variable is traction (or flux), which **acts over a surface area, not nodal forces (or sources)** as in the finite element method. This means that in defining the region of application of a boundary condition in **GPBEST** , it is necessary to specify both the nodal points and the elements involved.

If no boundary condition is specified (for a particular component) at a node, the primary load variable (of that component) is assumed to be zero.

The input lines involved in defining the element and nodes for a particular boundary condition set are described in this section.

GMR-bc IDGMR

Status - REQUIRED

Full Keyword - **GMR-bc**

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-Gmr card in **GMR input).

Additional Information -

A given boundary condition set can involve only a single GMR. If a boundary condition is to be applied to more than one GMR, a separate boundary condition set must be defined for each GMR.

Examples of Use -

1. Identifies the GMR name REG1 in connection with the specification of boundary conditions.

```
**BCSET
  ID-Bcset  TRAC1
  VALUe
  GMR-bc  REG1
  SURFace-bc SURF1
  ELEMents-bc   101   102
  TRACtion-bc  1
  SPLIst  ALL
  T-VAlue  1  100.0
```

SURFace-bc IDSUR

Status - OPTIONAL

Full Keyword - **SURFace-bc**

Function - Identifies the surface within the selected GMR on which the boundary condition is to be defined (NAME on SURFace card in **GMR input).

Input Variables -

IDSUR (Alphanumeric) - REQUIRED

Additional Information -

Either this keyword or the HOLE-line-bc 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 SURFace-bc card is not followed by an ELEMents-bc or POINts-bc card, then **GPBEST** will apply the boundary condition to all of the elements in the surface IDSUR.

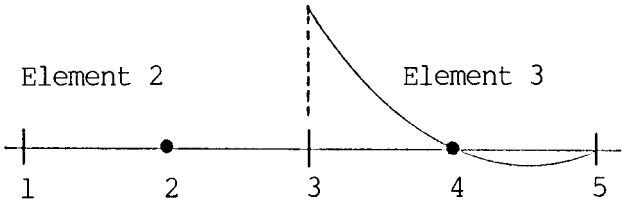
Examples of Use -

1. Identifies the surface name SURF1 relevant to the specification of boundary conditions.

```
**BCSET
  ID-Bcset  TRAC1
  GMR-bc  GMR1
  SURFace-bc SURF1
  ELEMents-bc  101   102   103
  TRACtion-bc  1
  SPLIst  ALL
  T-VAlue  1  100.0
```

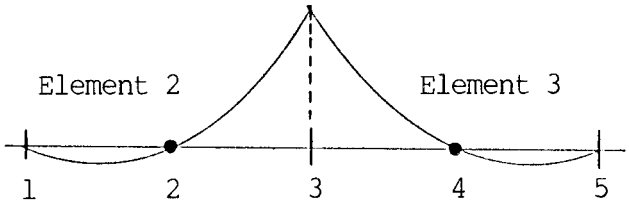
2. Specifies boundary conditions on surface SURF5. Because there are no subsequent ELEMents-bc or POINts-bc specified, the boundary condition is applied to all elements in the surface.

```
**BCSET
  ID-Bcset  DISP3
  GMR-bc  GMR1
  SURFace-bc SURF5
  DISPlacement-bc  3
  SPLIst  ALL
  T-VAlue  1  25.0
```



traction at NODE 3 is applied only over element 3

```
**BCSET
  ID BC1
  VALUE
    GMR GMR1
    SURF SURF1
    ELEMENT  3
    POINT  3
      TRAC  2
      SPLIST ALL
        T 1  100.0
```

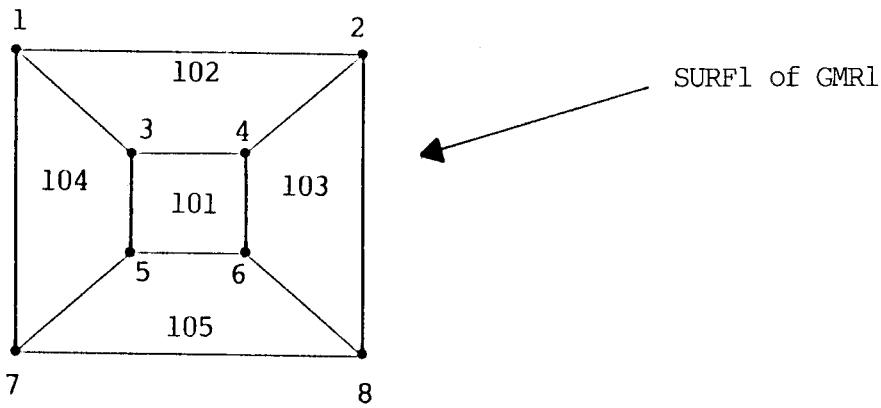


traction at NODE 3 is applied over both elements 2 and 3

```
**BCSET
  ID BC1
  VALUE
    GMR GMR1
    SURF SURF1
    ELEMENT  2  3
    POINT  3
      TRAC  2
      SPLIST ALL
        T 1  100.0
```

Figure for **BCSET: SURF, ELEM, and POIN card

2-D Boundary subset definition



	Result
Case 1 Input GMR1 SURF1	Boundary condition applied at nodes 1 through 8. Over elements 101 through 105.
Case 2 Input GMR1 SURF1 ELEM 101	Boundary condition applied at nodes 3, 4, 5 and 6. Over element 101 only.
Case 3 Input GMR1 SURF1 ELEM 102 103 105 POIN 3 4 5 6	Boundary condition applied at nodes 3, 4, 5 and 6. Over elements 102, 103 and 105.
Case 4 Input GMR1 SURF1 ELEM 101 POIN 3	Boundary condition applied at node 3. Over element 101 only.

Figure for **BCSET: SURF, ELEM and POIN cards

3-D boundary subset definition

ELEMents-bc EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - **ELEMents-bc**

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

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User-assigned numbers of the elements of surface IDSUR which are to be included within the boundary condition set.

Additional Information -

The effect of this card is to restrict the application of the boundary condition to a portion of the surface IDSUR.

This input may be continued on more than one card. Each card must begin with the keyword ELEMents-bc.

If the ELEMents-bc card is not followed by a POINts-bc card, then **GPBEST** will apply the boundary condition to all of the source points in the specified elements.

Examples of Use -

1. Specifies three elements on the surface SURF1 on which traction boundary conditions are given.

```
**BCSET
  ID-Bcset  TRAC2
  SURFace-bc  SURF1
  ELEMents-bc  101   102   103
  TRACtion-bc   2
  SPLIst  ALL
  T-VAlue  1  0.0
```

2. Boundary conditions are specified on 14 elements. The use of multiple ELEMents-bc cards is illustrated.

```
**BCSET
  ID-Bcset  DISP2
  SURFace-bc  SURF1
  ELEMents-bc    11    12    13    14
  ELEMents-bc    21    22    23    24    25    26
  ELEMents-bc   302   305   306   309
  DISPlacement-bc  2
  SPLIst  ALL
  T-VAlue  1  0.0
```

POINts-bc P1 P2 ... PN

Status - OPTIONAL

Full Keyword - **POINts-bc**

Function - Restricts the application of a boundary condition to a subset of the source points lying on the surface IDSUR.

Input Variables -

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

Additional Information -

This card restricts the application of the boundary condition to the source points specified.

This card may be repeated as often as is required. Each card must begin with the keyword.

If the POINts-bc card is specified, then **GPBEST** will apply the boundary condition to and only to the source points specified in this list.

If the POINts-bc card is specified, then the elements containing the points must be specified on the ELEMents-bc card.

Examples of Use -

1. Time-dependent input (in the x-direction) for points 5, 6, 7, and 8 over elements 102 and 103.

```
**BCSET
  ID-Bcset BC1
  VALUe
  GMR-bc REG3
    SURFace-bc BOTTOM
    ELEMents-bc 102 103
    POINts-bc  5  6  7
    TIMEs-bc  2.0  5.0  10.0  25.0
    TRACtion-bc 1
    SPLIst  6     7     5
    T-VAlue 1   100.0 100.0 100.0
    T-VAlue 2   200.0 200.0 300.0
    T-VAlue 3   500.0 600.0 200.0
    T-VAlue 4   800.0 400.0 100.0
```

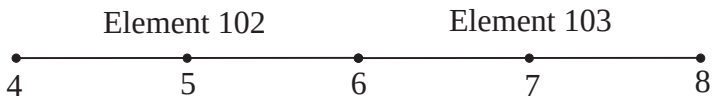


Figure for POINts-bc Card: Point Boundary Condition Specification

2. This example illustrates the use of multiple POINts-bc cards.

```
**BCSET
  ID-Bcset DISP1
  VALUe
  GMR-bc GMR1
    SURFace-bc SURF1
    ELEMents-bc   45   46   47   59   71
    POINts-bc      5    6    9
    POINts-bc     14   16   18   24
    POINts-bc     56   58
    DISPlacement-bc 1
    SPLIst     5    6    9
    SPLIst    14   16   18   24
    SPLIst    56   58
    T-VAlue 1    0.0
```

3. This example illustrates the use of multiple POINts-bc cards and displacements specified in all three global directions.

```
**BCSET
  ID-Bcset BC001
  VALUe
  GMR-bc REG1
  SURFace-bc SURF1
    ELEMents-bc   21    22    23    24    25    26
    ELEMents-bc   27    28    29    30    31    32
    POINts-bc    101   103   104   105   107   113
    POINts-bc    114   102   161   162   163   170
    POINts-bc    171   173   174   147   175   176
  DISPlacement-bc 1
    SPLIst   101   103   104   105   107   113
    SPLIst   114   102   161   162   163   170
    SPLIst   171   173   174   147   175   176
    T-VAlue 1      0.5   1.1   1.2   1.3   1.6   2.0
    T-VAlue 1      1.5   1.0   1.3   1.3   1.3   1.1
    T-VAlue 1      0.3   0.8   0.9   0.2   2.3   2.2
  DISPlacement-bc 2
    SPLIst   101   103   104   105   107   113
    SPLIst   114   102   161   162   163   170
    SPLIst   171   173   174   147   175   176
    T-VAlue 1      0.5   0.5   0.5   0.5   0.5   0.5
    T-VAlue 1      0.5   0.5   0.5   0.5   0.5   0.6
    T-VAlue 1      0.5   0.5   0.6   0.6   0.6   0.6
  DISPlacement-bc 3
    SPLIst   101   103   104   105   107   113
    SPLIst   114   102   161   162   163   170
    SPLIst   171   173   174   147   175   176
    T-VAlue 1      0.3   0.8   1.9   1.2   2.3   2.2
    T-VAlue 1      3.5   3.1   3.2   3.3   3.6   3.0
    T-VAlue 1      2.2   1.0   1.0   1.2   1.3   1.4
```

TIMEs-bc T1 T2 ... TN

Status - OPTIONAL

Full Keyword - **`TIMEs-bc`**

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 TIMEs-bc.

The time values input on this card need not agree with the times at which output was requested in the case control input. Different sets of time points may be used for different boundary conditions in the same analysis.

The time points must be specifed in ascending order.

Boundary condition values at other 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).

This card is not valid for periodic acoustics and structural-acoustics problems.

Examples of Use -

1. Specify a time-varying traction boundary condition.

```
**BCSET
  ID-Bcset TRAC1
  VALUe
    GMR-bc GMR1
    SURFace-bc SURF1
    ELEMents-bc    3
      TIMEs-bc  1.0  2.0  3.0  3.5  4.0  5.0
      TRACtion-bc 1
      SPLIst ALL
      T-VAlue 1  100.0
      T-VAlue 2  200.0
      T-VAlue 3  200.0
      T-VAlue 4  100.0
      T-VAlue 5  100.0
      T-VAlue 6  300.0
```

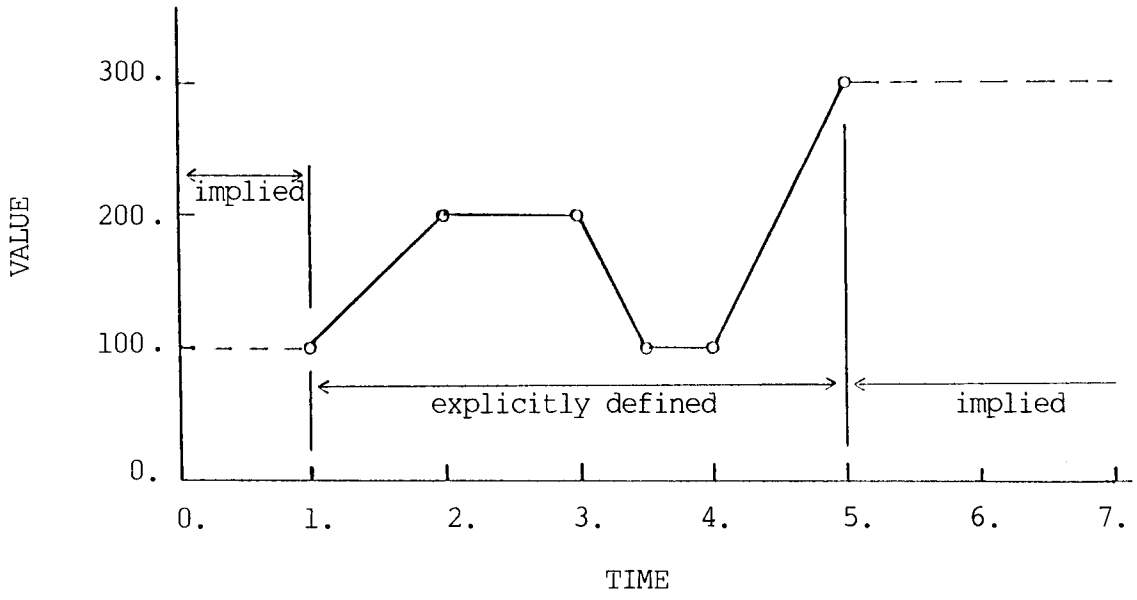


Figure for TIMEs-bc Card: VALUe vs. TIMEs-bc for Boundary Condition Input

2. This is an example of how to run two separate load cases in a single **GPBEST** run. In **CASE, the card TIMEs-output 1.0 2.0 appears. This calls out for a solution at TIME=1.0 and TIME=2.0. For an ELAStic problem, this amounts to having two load cases. The **BCSET block for this problem is shown below. For the first load case, the load magnitude is linearly increasing from node 19 to node 11. In the second load case, the load magnitude is linearly decreasing from node 19 to node 11, and acts in the opposite direction as load 1.

```
**CASE
  Title Two Load Case
  ELASTIC
  TIMEs-output 1.0 2.0
  .
  .
  .
**BCSET
  ID-Bcset LOAD1
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc   6   7   8   9
    POINts-bc    11  19  12  18  17
    POINts-bc    16  15  14  13
    TIMEs-bc   1.0   2.0              $ Load Case 1 and 2
    TRACtion-bc 2
      SPLIst 19  18  17  16  15
      SPLIst 14  13  12  11
      T-VAlue 1      0.0   125.0   250.0   375.0   500.0 $ Load 1
      T-VAlue 1    625.0   750.0   875.0  1000.0
      T-VAlue 2  -1000.0  -875.0  -750.0  -625.0  -500.0 $ Load 2
      T-VAlue 2   -375.0  -250.0  -125.0     0.0
```

NOTE: In the above example, the order of the nodes given on the POINts-bc card does not matter. However, the order of the nodes given on the SPLIst card does matter: it determines the order of values on the subsequent ‘T-VAlue’ cards.

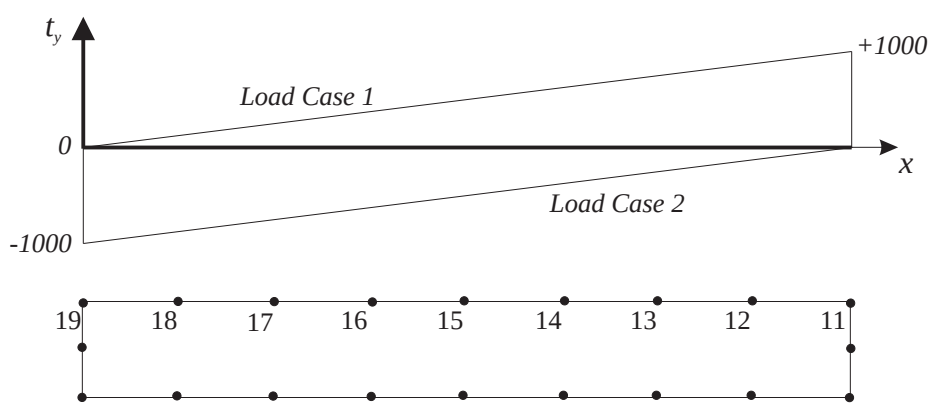


Figure for TIMEs-bc Card: Multiple Load Cases

FREQuency-bc T1 T2 ... TN

Status - OPTIONAL

Full Keyword - **FREQuency-bc**

Function - Specifies the frequnecies at which the variable involved in the boundary condition set will be specified.

Input Variables -

T1 (Frequency) - REQUIRED

First frequency point for boundary condition specification.

T2,...,TN (Frequency) - OPTIONAL

Subsequent frequency 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 FREQuency-bc.

The frequency values input on this card need not agree with the frequencies at which output was requested in the case control input. Different frequency points may be used in different boundary conditions sets in the same analysis.

The frequency points must be specifed in ascending order.

Boundary condition values at other input frequencies are calculated by linear interpolation.

If a frequency card does not appear, the variables involved in the boundary condition are assumed to be frequency-independent. Consequently, only a single frequency point may be specified for the SPACE/FREQUENCY VARIATION (as defined in section 5.5.8).

Examples of Use -

1. Specify a frequency-dependent complex pressure boundary condition.

```
**BCSET
  ID-Bcset PRES1
  VALUe
    GMR-bc GMR1
    SURFace-bc SURF1
    ELEMents-bc    3
      FREQuency-bc  1.0  2.0  3.0  3.5  4.0  5.0
      PRESsure
      SPLIst ALL
      T-VAlue 1  100.0 100.0
      T-VAlue 2  200.0 100.0
      T-VAlue 3  200.0 100.0
      T-VAlue 4  100.0 150.0
      T-VAlue 5  100.0 200.0
      T-VAlue 6  300.0 250.0
```

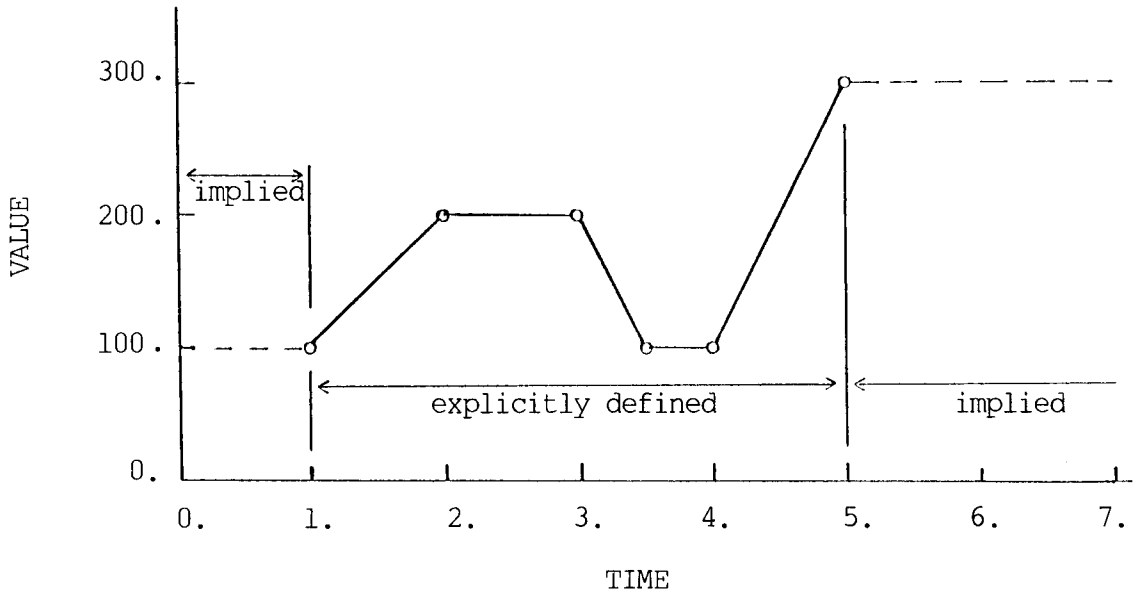


Figure for FREQuency-bc Card: VALUe vs. FREQuency-bc for Boundary Condition Input

5.5.5 VALUE BOUNDARY CONDITIONS FOR SURFACE ELEMENTS

TRACtion-bc IDIR

Status - OPTIONAL

Full Keyword - **TRACtion-bc**

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 axisymmetric analysis:

1 - r direction (or z direction if ‘‘COORdinate-system ZR’’ appears in **CASE)
2 - z direction (or r direction if ‘‘COORdinate-system RZ’’ appears in **CASE)
3 - θ direction (for non-axisymmetric loading only)

For data defined in cylindrical coordinates:

1 - r direction
2 - θ direction
3 - z direction

For local coordinates:

1 - outer normal direction or direction 1 defined under ‘‘LOCAl EXPLICIT’’
2,3 - can only be used if defined under ‘‘LOCAl NORMAL’’ or ‘‘LOCAl EXPLICIT’’

Additional Information -

This card can only be used in a boundary condition data set containing a VALUe or VARIable card.

Up to three sets (two in 2-D) 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-bc input line must be immediately followed by the space/time variation.

The structural part of a structural-acoustics problem cannot have a traction input.

Examples of Use -

1. Defines a traction of magnitude 100.0 units in the positive global x-direction.

```
**BCSET
  ID-Bcset TRAC1
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc    43
  TRACtion-bc 1
  SPLIst     ALL
  T-VAlue   1    100.0
```

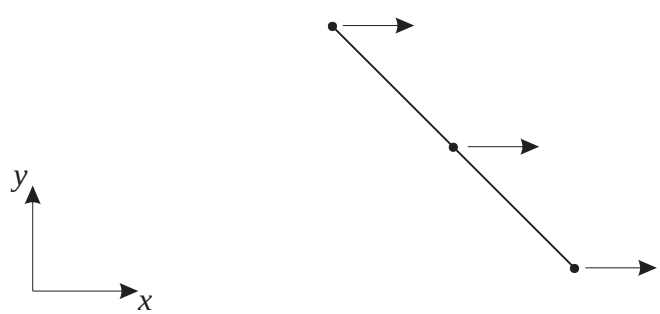


Figure for TRACtion-bc Card: (a) Traction in Global x-Direction

2. Defines a traction of magnitude 100.0 units in the direction of the outward normal.

```
**BCSET
  ID-Bcset TRAC1
  VALUe
  LOCAl
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc    43
  TRACtion-bc 1
  SPLIst     ALL
  T-VAlue   1    100.0
```

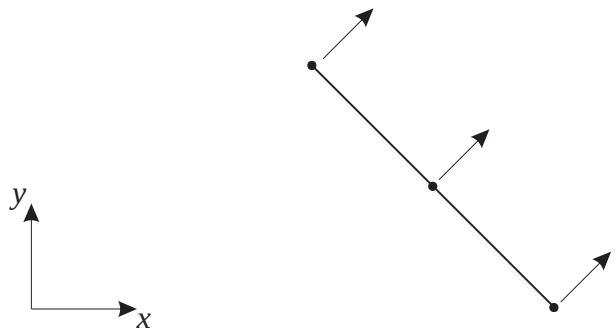


Figure for TRACtion-bc Card: (b) Traction in Normal Direction

3. Specify tractions in two coordinate directions.

```
**BCSET
  ID-Bcset TRAC2
  VALUe
  LOCAl
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc    15   19   24
  TRACtion-bc 1
    SPLIst     ALL
    T-VAlue   1    25.0
  TRACtion-bc 2
    SPLIst     ALL
    T-VAlue   1    12.0
```

4. Frequently a user may desire to specify a concentrated load at a node. In finite element analysis, this is frequently how forces are modeled. However, in boundary element analysis, a concentrated force cannot be specified unless it lies $inside$ the body (see Section 5.6.6). To model a concentrated force on an edge of a body, it must be distributed as a traction over a small area. Consider Figure (c), where a force F=5 (N) is placed near the end of a cantilever beam.

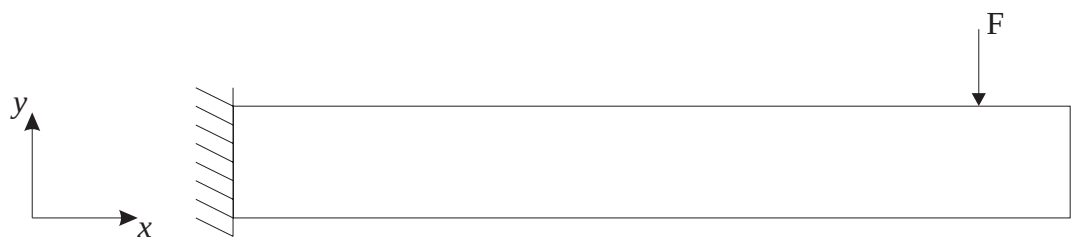


Figure for TRACtion-bc Card: (c) Cantilever Beam With Concentrated Force

Figure (d) shows a close-up of the right end of the beam where the force has been distributed over a boundary element of length .1 (m). Therefore, the magnitude of the traction is 5/.1 = 50 (N/m). This is the proper way to model a concentrated force on the boundary. However, note that the stresses in the immediate region near the traction will not be as accurate as the stresses in regions further away from the traction.

```
**BCSET
  ID-Bcset TRAC1
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc    25
  TRACtion-bc 2
    SPLIst     ALL
    T-VAlue   1    -50.0
```

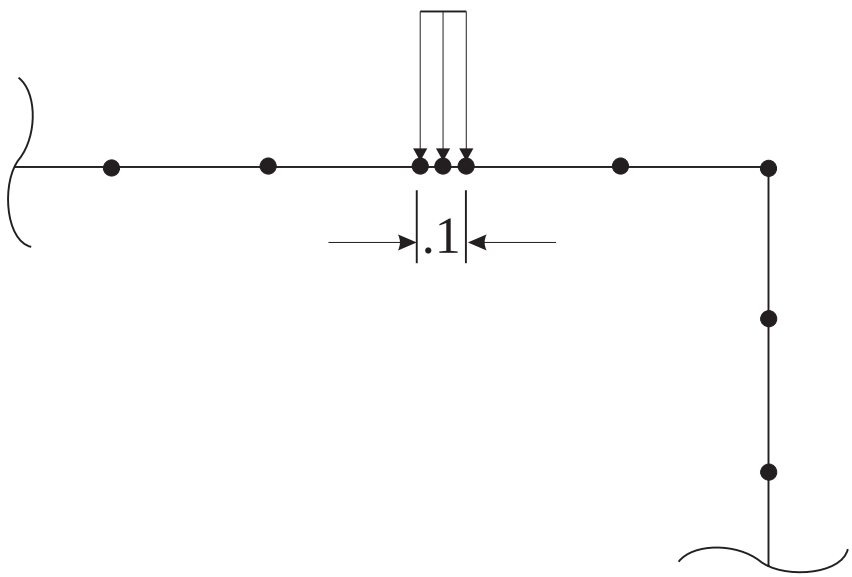


Figure for TRACtion-bc Card: (d) Close-up of Force Distributed Over Element

The boundary element user must be cautioned about using the modeling method in Figure (e). Although a single force can be specified at a single node, **GPBEST** will distribute the force over the adjoining elements (7 and 8) using the element shape functions, as illustrated in the figure in the SURF section of Section 5.5.4. The user would have to calculate the proper force magnitude, taking into account the shape functions. And since the force would be distributed over a fairly large area, users are encouraged not to model forces this way.

```
**BCSET
  ID-Bcset TRAC1
  VALUe
  GMR-bc  REG1
  SURFace-bc  SURF1
  ELEMents-bc    7    8
  POINts-bc     15
  TRACtion-bc 2
    SPLIst     15
    T-VAlue   1    -31.249
```

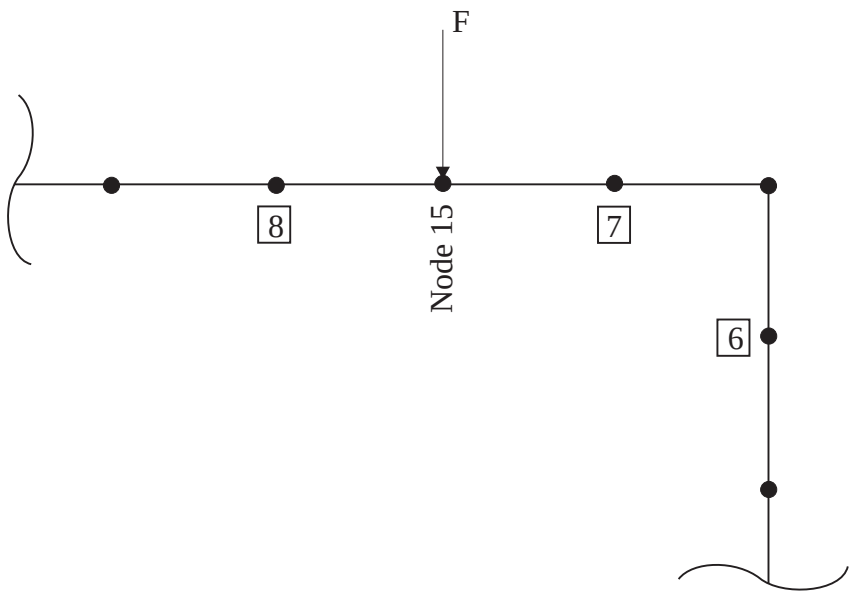


Figure for TRACtion-bc Card: (e) Incorrect Modeling of Concentrated Force

LOAD-bc ADIR

Status - OPTIONAL

Full Keyword - **LOAD-bc**

Function - Indicates that a force or heat rate load will be specified and uniformly distributed (as traction or flux) over the elements (and nodes) specified.

Input Variables -

ADIR (Integer or Alphanumeric) - REQUIRED

Allowable values are ‘‘1’’, ‘‘2’’, ‘‘3’’, or ‘‘HEAT’’.

Defines the component in which the force or heat load is specified.

For cartesian coordinates:

1 - x direction
2 - y direction
3 - z direction
HEAT - heat rate

For axisymmetric analysis:

1 - r direction (or z direction if ‘‘COORdinate-system ZR’’ appears in **CASE)
2 - z direction (or r direction if ‘‘COORdinate-system RZ’’ appears in **CASE)
3 - θ direction (for non-axisymmetric loading only)
HEAT - heat rate

For data defined in cylindrical coordinates:

1 - r direction
2 - θ direction
3 - z direction
HEAT - heat rate

For local coordinates:

1 - outer normal direction or direction 1 defined under ‘‘LOCAl EXPLICIT’’
2,3 - can only be used if defined under ‘‘LOCAl NORMAL’’ or ‘‘LOCAl EXPLICIT’’
HEAT - heat rate

Additional Information -

This type of boundary condition is useful when the total force or rate of heat loading is known and the exact distribution is uniform or not important. Often when an auto-mesher is used the size of the elements are somewhat arbitrary, and the surface area of the elements is not readily known. This presents difficulty in determining the proper traction or flux distribution. When the LOAD-bc boundary condition is

invoked, **GPBEST** calculates the total area of the elements specified and determines the correct traction or flux to be applied. This value of traction or flux is uniformly distributed over the elements specified to produce the total load specified on the ‘‘T’’ card.

The LOAD-bc statement must be immediately followed by the statement ‘‘SPLIst ALL’’ and this card is followed by a time variation (the T-VAlue cards). Since a uniform traction or flux is assumed, nodal variation input is not applicable to the LOAD-bc under **BCSET, and therefore, the argument ALL is required on the SPLIst statement.

This card can only be used in a boundary condition data set containing a VALUe or VARIable card.

Up to three sets (two in 2-D) of force load and one heat rate load specifications may be included in the same boundary condition data set. All must refer to the same boundary condition set. Only one condition may be applied in a given component direction.

The default condition is always to set tractions and flux to zero. After all boundary conditions data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction/flux conditions.

The LOAD-bc boundary condition is not available in structural-acoustic analysis.

Examples of Use -

1. Define a time-dependent LOAD-bc boundary condition in directions 1 and 3 which is distributed over three elements.

```
**BCSET
  ID-Bcset  FORCEBC
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF11
  ELEMents-bc  511   512   637
  TIMEs-bc      0.0   7.5
  LOAD-bc 1
    SPLIst  ALL
    T-VAlue  1  100.0
    T-VAlue  2  900.0
  LOAD-bc 3
    SPLIst  ALL
    T-VAlue  1  200.0
    T-VAlue  2 1800.0
```

2. Define a rate of heat loading boundary condition.

```
**BCSET
  ID-Bcset FLUXBC
  VALUe
  GMR-bc  TOP
  SURFace-bc  FRONT
  ELEMents-bc  101
  LOAD-bc HEAT
    SPLIst   ALL
    T-VAlue   1    1500.0
```

GLOAd-bc NODE COOR-X COOR-Y COOR-Z

Status - OPTIONAL

Full Keyword - **GLOAd-bc**

Function - Indicates that a generalized load b.c. will be specified. The generalized load is a force-moment system which contains three components in 2D (two loads and one moment) and six components in 3D (three loads and three moments) and is assumed to act through a specified point.

Input Variables -

NODE (Integer) - Required

The node ID of the point at which the generalized load is assumed to be applied. This ID is not referenced by GPBEST. It is required only as an aid for identification.

COOR-X, COOR-Y,	(Real)	REQUIRED
COOR-Z,	(Real)	REQUIRED IN 3D

The coordinates of the point which the generalized load is assumed to be applied.,

Additional Information -

The user specifies force and moment vectors and a point at which they are assumed to act. In general, this point is independent of the discretized geometry and is given to uniquely define the force-moment system to be applied. The user also specifies elements on the boundary of the model for which a force-moment system, which is equivalent to the one specified, will be applied. GPBEST will automatically generate the appropriate traction, which will be applied over the specified elements to exactly represent the force-moment system specified.

The GLOAd-bc statement must be immediately followed by the "SPLIst ALL" statement. This card must then be followed by a time variation (the T-VAlue cards). Three (and only three) values must be input on the T-VAlue card in 2D analysis and 6 (and only six) values must be input on the T-Value card in 3D. In 2D, the three values (in the order they are input) are forced in the x-direction, the force in the y-direction, and the moment in the out-of-plane direction. In 3D, six values (in the order they are input) are the force in the x, y, and z-directions followed by the moment in the x, y, and z-direction.

No other mechanical boundary condition (displacement, tractions, spring, etc.) can be applied to the elements that are specified in a GLOAd-bc **BCSET. In a thermoelastic analysis, thermal boundary conditions (temperature, flux, convection, heat load) may be applied to elements that are also contained in the GLOAd-bc **BCSET.

The GLOAd-bc boundary condition is not allowed in axisymmetric or generalized axisymmetric analysis.

Definition of Boundary Conditions

The GLOAd-bc boundary condition is not allowed in structural-acoustic analysis.

Examples of Use -

1. In a 2D analysis, apply a time dependent generalized load.

```
**BCSET
  ID   BC101
  VALUe
  GMR-bc   GMR2
  SURFace-bc SHAFT1
  ELEMent-bc     101     102     103     104
  TIMEs-bc        1.0     2.0
  GLOAd-bc    2001    0.500   0.1573
  SPLIst ALL
    T    1    0.0     0.0     100.0
    T    2    100.0   0.0     200.0
```

2. In a 3D analysis, apply a time dependent generalized load.

```
**BCSET
  ID   BC501
  VALUe
  GMR-bc   GMR5
  SURFace-bc SURF11
  ELEMent-bc     1057     1056     1118     1119     1120
  TIMEs-bc        1.0      10.0
  GLOAd-bc    3007     0.5109   11.051   113.17
  SPLIst ALL
    T    1    0.0     0.0     0.0     100.0     0.0     0.0
    T    2    100.0   0.0     0.0     100.0     0.0     500.0
```

TORQue-bc NODE1 NODE2

Status - OPTIONAL

Full Keyword - **`TORQue-bc`**

Function - Indicates that a torque will be specified about the point NODE1 in 2-D, or about the axis defined by NODE1 and NODE2 in 3-D. A uniform traction will be calculated by **GPBEST** and distributed over the elements (and nodes) specified to produce the specified torque.

Input Variables -

NODE1 (Integer) - REQUIRED

In 2-D this node defines the point about which the torque will be applied. In 3-D this will be the first of two points used to define the axis about which the torque will act.

NODE2 (Integer) - REQUIRED in 3-D only

This is the second point used to define the axis about which the torque will act.

Additional Information -

The TORQue-bc boundary condition is a convenient way to apply a torque to a modeled part. The user defines the value of torque, the axis about which it will act, and the elements (and nodes, if desired) which will carry the tractions which will generate the torque. **GPBEST** will automatically calculate the magnitude and direction of the tractions required to produce the desired torque in the specified direction. The tractions are uniformly distributed over the specified elements in the direction formed by the cross product of the torque axis and the radius from the axis to the point of application.

The direction of torque is defined with the use of the right-hand rule, i.e., the torque will act in a counter-clockwise direction when the axis of torque is pointing at your eyes. In 2-D, the axis of torque is defined as a vector perpendicular to the x-y plane which passes through NODE1. In 3-D, the axis is a vector from NODE1 pointing to NODE2.

The coordinates of NODE1 (and NODE2 in 3-D) must be defined in the current GMR (specified on the GMR-bc card of the current **BCSET).

The TORQue-bc statement must be immediately followed by the ‘‘SPLIst ALL’’ statement. This card must then be followed by a time variation (the T-VAlue cards). The value on the T-VAlue card is the total value of torque. **GPBEST** automatically creates the appropriate nodal variation of traction for the specified value of torque. For this reason, nodal variation input is not applicable to the TORQue-bc **BCSET, and therefore, the argument ALL is required on the SPLIst statement.

No other mechanical boundary conditions (displacement, tractions, springs, etc.) can be applied to the elements that are specified in a TORQue-bc **BCSET. In a

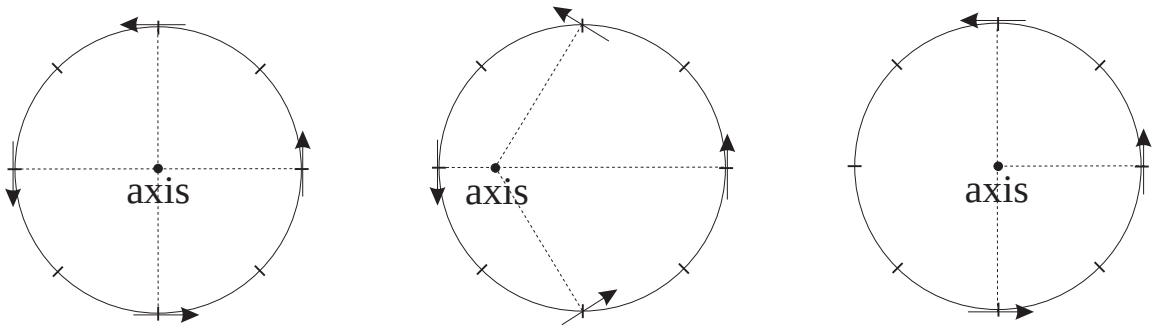
thermoelastic analysis, thermal boundary conditions (temperature, flux, convection, heat load) may be applied to elements that are also contained in a TORQue-bc **BCSET.

The TORQue-bc boundary condition is not allowed in axisymmetric or generalized axisymmetric analysis.

The TORQue-bc boundary conditions are not allowed in structural-acoustic analysis.

The application of the TORQue-bc boundary condition may produce a resultant force when the elements specified to carry the tractions are not symmetrical about the axis of rotation. For instance, if a torque is applied uniformly to a shaft and the axis of torque coincides with the axis of the shaft (see the Figure below), the resultant force will be zero. However, if the axis is off center or elements are specified only on one side of the shaft, then a resultant force will be generated. The resultant force vector that is generated as a by-product of the applied torque is reported in the **GPBEST** result file for each TORQue-bc **BCSET, at every time step. It is the user's responsibility to be aware of this.

NOTE: The GLOAd-bc boundary condition does not produce a residual force other than the force specified. For this reason, the GLOAd-bc boundary condition is a better choice for applying a torque over a surface that is contained in a single GMR. If the surface is split among two or more GMR's, the TORQue-bc boundary condition is more appropriate. In this case the value of torque should be split and applied over the individual surfaces. The sum of all individual values should add up to the total torque desired.



elements specified symmetric about axis: resultant force is zero

axis is off center: resultant upward force is generated

elements specified only on right side (unsymmetric): resultant upward force is generated

Figure for TORQue-bc Card: Tractions generated by **GPBEST** to model the torque are uniform and perpendicular to the moment arm

Examples of Use -

1. Define a time-dependent TORQue-bc boundary condition set for 2-D analysis.

```
**BCSET
  ID-Bcset  TORQUE2D
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF11
  ELEMents-bc  101   102   103   104   105   106   107   108
  TIMEs-bc      0.0   1.0   5.0
  TORQue-bc    1101
    SPLIst  ALL
    T-VAlue  1  1000.0
    T-VAlue  2  2000.0
    T-VAlue  3 10000.0
```

2. Define a TORQue-bc boundary condition set for 3-D analysis.

```
**BCSET
  ID-Bcset TORQUE3D
  VALUe
  GMR-bc  SHAFT
  SURFace-bc  CYLINDER
  ELEMents-bc  101   102   103   104   105   106   107   108
  TORQue-bc    2001  2002
    SPLIst   ALL
    T-VAlue   1    5000.0
```

DISPlacement-bc IDIR

Status - OPTIONAL

Full Keyword - **`DISPlacement-bc`**

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 cartesian coordinates:

1 - x direction

2 - y direction

3 - z direction

For axisymmetric analysis:

1 - r direction (or z direction if ''COORdinate-system ZR'' appears in **CASE)

2 - z direction (or r direction if ''COORdinate-system RZ'' appears in **CASE)

3 - θ direction (for non-axisymmetric loading only)

For data defined in cylindrical coordinates:

1 - r direction

2 - θ direction

3 - z direction

For local coordinates:

1 - outer normal direction or direction 1 defined under ''LOCAl EXPLICIT''

2,3 - can only be used if defined under ''LOCAl NORMAL'' or ''LOCAl EXPLICIT''.

Additional Information -

This card can only be used in a boundary condition data set containing a VALUe or VARIable card.

Up to three sets (two in 2-D) 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-bc input line must be immediately followed by space/time variation.

For displacement constraints over the structural part of structural-acoustics problems, see Section 5.5.12.

Examples of Use -

1. Defines complete fixity of nodes (i.e., displacements in x and y directions are zero) of three elements for a two-dimensional elasticity problem.

```
**BCSET
  ID-Bcset  DISP1
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF1
  ELEMents-bc  104  105  106
  DISPlacement-bc  1
    SPLIst  ALL
    T-VAlue  1  0.0
  DISPlacement-bc 2
    SPLIst  ALL
    T-VAlue  1  0.0
```

2. Constrain the normal movement of an element (i.e., roller boundary condition).

```
**BCSET
  ID-Bcset DISP1
  VALUe
  LOCAl
  GMR-bc  GMR1
  SURFace-bc  SURF1
  ELEMents-bc    53
  DISPlacement-bc 1
    SPLIst   ALL
    T-VAlue   1    0.0
```

FLUX-bc

Status - OPTIONAL

Full Keyword - **FLUX-bc**

Function - Indicates that the flux will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

FLUX-bc is only valid for HEAT, DIFFusion, CTHErmal, or CONSolidation analysis.

This card can only be used in a boundary condition data set containing a VALUe or VARIable 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-bc input line must be immediately followed by the space/time variation.

Note that a positive value of FLUX-bc indicates heat is leaving the body. A negative flux indicates heat is flowing into the body.

Examples of Use -

1. Defines unit flux flowing out of a region across three elements.

```
**BCSET
  ID-Bcset  ENTER1
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF1
  ELEMents-bc  22  23  24
  FLUX-bc
  SPLIst  ALL
  T-VAlue  1  1.0
```

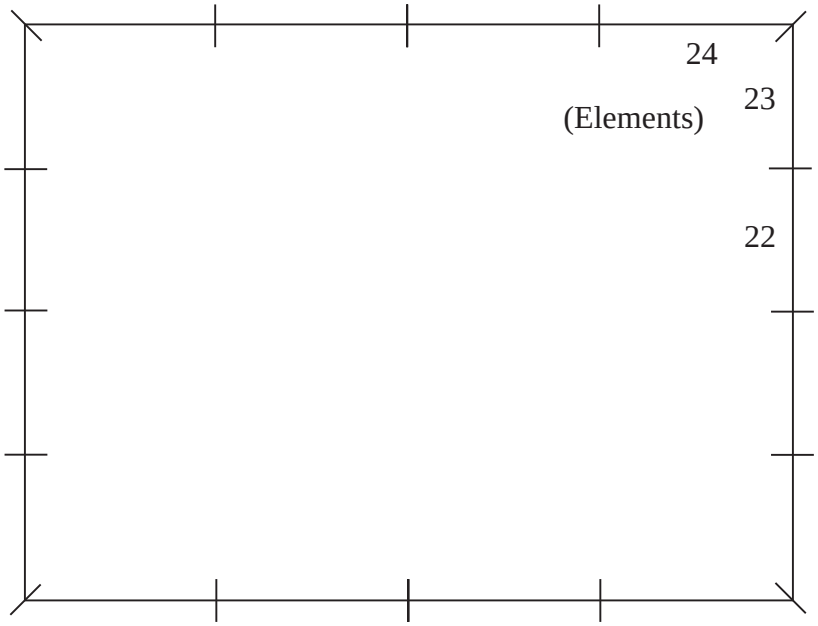


Figure for FLUX-bc Card: Two-Dimensional Problem

TEMPeratures-bc

Status - OPTIONAL

Full Keyword - **`TEMPeratures-bc`**

Function - Indicates that the temperature will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

TEMPeratures-bc is only valid for HEAT, DIFFusion or CTHErmal analysis.

This card can only be used in a boundary condition data set containing a VALUe or VARIable 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 TEMPeratures-bc 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-Bcset  TOP
  VALUe
  GMR-bc  GMR2
  SURFace-bc  SURF2
  ELEMents-bc  218  219
  DISPlacement-bc  1
  SPLIst  ALL
  T-VAlue  1  0.0
  TEMPeratures-bc
  SPLIst ALL
  T-VAlue  1  0.0
```

PRESsure-bc

Status - OPTIONAL

Full Keyword - **PRESsure-bc**

Function - Indicates that the pressure will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

PRESsure-bc is only valid for ACOUstic, SACOustic (structural- acoustics) and CONSolidation analyses. If the user wants to specify a pressure boundary condition for another analysis (e.g., elasticity), use the TRACtion-bc card.

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

The specification of flux or pressure may be included with up to three sets of traction and/or displacement specifications in the same boundary condition data set for consolidation analysis. All must refer to the same boundary condition set.

When applicable, the default condition is to set pressure to zero.

The PRESsure-bc input line must be immediately followed by the space/time variation.

For structural-acoustics problems, a structural pressure loading (complex value) may be applied in an identical manner. The GMR name would identify that the pressure loading is applied over elements of the structural region.

Examples of Use -

1. Indicates that zero pressure will be specified on the concerned elements.

```
**BCSET
  ID-Bcset  PRES
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF1
  ELEMents-bc  4  5
  PRESsure-bc
  SPLIst  ALL
  T-VAlue  1  0.0
```

2. Specify a non-zero pressure in an acoustic analysis. Pressure is a complex quantity. However, in this example pressure only contains a real component.

```
**BCSET
  ID-Bcset PRES16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
  PRESsure-bc
    SPLIst ALL
    $       Real    Imaginary
    T-VAlue 1     3.2         0.
```

3. Specify a pressure boundary condition along with a traction boundary condition.

```
**BCSET
  ID-Bcset TRAC16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
  TRACtion-bc 2
    SPLIst ALL
    T-VAlue 1  -100.0
  PRESsure-bc
    SPLIst ALL
    T-VAlue 1     0.0
```

VELOcity-bc

Status - OPTIONAL

Full Keyword - **`VELOcity-bc`**

Function - Indicates that the velocity will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

VELOcity-bc is only valid for ACOUstic analysis and acoustic regions of structural-acoustics problems.

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

When applicable, the default condition is to set velocity to zero.

Examples of Use -

1. Defines unit velocity conditions across three elements. Velocity is a complex quantity. However, in this example velocity has only a real component.

```
**BCSET
  ID-Bcset  ENTER1
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SURF1
  ELEMents-bc  22  23  24
  VELOcity-bc
  SPLIst  ALL
  $     Real    Imaginary
  T-VAlue  1  1.0        0.
```

5.5.6 DEFINITION OF HOLE/COOLING LINE FOR APPLICATION OF BOUNDARY CONDITIONS

GMR-line-bc IDGMR

Status - REQUIRED

Full Keyword - **GMR-line-bc**

Function - Identifies the GMR on the surface of 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-Gmr card in **GMR input).

Additional Information -

A given boundary condition set can involve only one 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.

This section is not valid for acoustic and structural-acoustics problems.

Examples of Use -

1. Define a linear increasing flux inside a hole (elements 203, 204) for heat transfer analysis.

```
**BCSET
  GMR-line-bc  REG1
  HOLE-line-bc
  ELEMents-bc  203  204
  TIMEs-line-bc  0.0  10.0
    FLUX-line-bc
    SPLIst  ALL
    T-VAlue  1  0.0
    T-VAlue  2  100.0
```

HOLE-line-bc
CLINe-line-bc

Status - OPTIONAL

Full Keyword - **`HOLE-line-bc`**

Full Keyword - **`CLINe-line-bc`**

Function - Identifies this as a boundary condition set for hole/cooling line elements within the selected GMR.

Input Variables - NONE

Additional Information -

Either this keyword or the SURFace-bc keyword must be input for each boundary condition set.

If the HOLE-line-bc or CLINe-line-bc keyword line is not followed by an ELEMents-bc line, then **GPBEST** will apply the boundary condition to all of the hole/cline elements in the current GMR.

Examples of Use -

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

```
**BCSET
  GMR-line-bc  GMR5
    HOLE-line-bc
    ELEMents-bc  509
    POINts-bc  5025  5026  5027
      TEMPeratures-bc
      SPLIst  5025   5025   5027
      T-VAlue  1    100.0  150.0  200.0
```

ELEMents-hole-cline EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - **ELEMents-bc**

Function - Specifies the hole/cline elements to which a boundary condition is to be applied.

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User element numbers of the hole/cline 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/cline elements within the previously specified GMR.

This input may be continued on more than one card. Each card must begin with the keyword ELEMents-bc.

If the ELEMents-bc card is not followed by a POINts-bc card, the **GPBEST** will apply the boundary condition to all of the source points in the specified hole/cline element.

Examples of Use -

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

```
**BCSET
  GMR-line-bc REG5
    HOLE-line-bc
    ELEMents-bc  207  208  209  311
      FLUX-line-bc
      SPLIst ALL
      T-VAlue 1 100.0
```

POINts-line-bc P1 P2 ... PN

Status - OPTIONAL

Full Keyword - **POINts-line-bc**

Function - Restricts the application of boundary conditions to a subset of the source points associated with hole/cline elements specified above.

Input Variables -

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

Additional Information -

This card restricts the application of the boundary conditions to the nodal hole/cline points specified.

This card may be repeated as often as is required. Each card must begin with the keyword.

If the POINts-line-bc card is specified, then GPBEST will apply the boundary condition to all of the hole/cline points specified in this list. Also, if the POINts-line-bc 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 flux (in time) that varies from point to point in an element.

```
**BCSET
  GMR-line-bc REG9
    HOLE-line-bc
    ELEMents-bc 207
    POINts-line-bc  2005  2007  2011
    TIMEs-line-bc    0.0   1.0
      FLUX-line-bc      SPLIst  2005  2007  2011
        T-VAlue  1   0.0   0.0   0.0
        T-VAlue  2   10.0  15.0  20.0
```

TIMEs-line-bc T1 T2 ... TN

Status - OPTIONAL

Full Keyword - **`TIMEs-line-bc`**

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_hole_cline.

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 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
  GMR-line-bc REG10
    HOLE-line-bc
    ELEMents-bc  907
    POINts-line-bc   9003  9004
    TIMEs-line-bc     0.0   5.0   10.0   20.0
      TEMPeratures-bc
      SPLIst  9003  9004
      T-VAlue   1   0.0   0.0
      T-VAlue   2   10.0  20.0
      T-VAlue   3   20.0  50.0
      T-VAlue   4   30.0  60.0
```

5.5.7 VALUE BOUNDARY CONDITIONS FOR HOLE/COOLING LINE ELEMENTS

FLUX-line-bc

Status - OPTIONAL

Full Keyword - **FLUX-line-bc**

Function - Indicates that the flux will be specified for all hole/cline nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

FLUX-line-bc specification for holes/clines is only valid for HEAT analysis.

This card can only be used in a boundary condition data set containing a VALUe or VARIable card.

When applicable, the default condition is to set flux to zero.

The FLUX-line-bc input line must be immediately followed by the space/time variation.

A positive value for flux implies that heat is leaving the body. A negative value of flux indicates that heat is entering the body.

Examples of Use -

1. Define a flux (for all times) at points in element 509.

```
**BCSET
  GMR-line-bc REG3
    HOLE-line-bc
    ELEMents-bc  509
    POINts-line-bc    5003  5004
      FLUX-line-bc
      SPLIst 5003  5004
      T-VAlue  1   5.0   6.0
```

TEMPeratures-line-bc

Status - OPTIONAL

Full Keyword - **TEMPeratures-line-bc**

Function - Indicates that the temperature will be specified for all hole/cline nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

TEMPeratures-line-bc specification for holes/clines is only valid for HEAT analysis.

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

When applicable, the default condition is to set flux to zero.

The TEMPeratures-line-bc **BCSET 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-line-bc EXCHANGER
    HOLE-line-bc
    ELEMents-bc 201
    TIMEs-line-bc    0.0  5.0  10.0  30.0
      TEMPeratures-line-bc
      SPLIst ALL
      T-VAlue 1 0.0
      T-VAlue 2 20.0
      T-VAlue 3 50.0
      T-VAlue 4 100.0
```

5.5.8 DEFINITION SPACE/TIME VARIATION

SPLIst 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 on the T-VAlue card (see next page) .

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 -

The SPLIst card must not be confused with the POINts-bc or POINts-line-bc cards (Sections 5.5.4 and 5.5.6). The POINts-bc and POINts-line-bc cards restrict the application of the boundary condition set to specific nodes of the elements listed. The SPLIst card defines the order in which nodal values will be input on the T-VAlue card. The POINts-bc and POINts-line-bc cards are optional whereas the SPLIst card is required for all VALUe input.

If N1 = ALL, then **GPBEST** 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 or 5.5.6).

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 SPLIst must equal the number of nodes defined by the SURFace-bc, ELEMents-bc, POINts-bc and POINts-line-bc cards (section 5.5.4 or 5.5.6).**

This input may be continued on more than one card if required. Each card must begin with the keyword SPLIst.

This card is not valid for nodal boundary conditions in the structural portion of a structural-acoustics problem. Refer to Section 5.5.12 for more information.

Examples of Use -

1. Explicitly specify nodal points.

```
**BCSET
  ID-Bcset DISP1
  VALUe
  GMR-bc GMR1
    SURFace-bc SEG1
    ELEMents-bc   21
    POINts-bc     23  34  45  57
    DISPlacement-bc 1
      SPLIst   34  57  23  45
      T-VAlue 1     0.0 1.0 2.0 3.0
```

2. Use of the ALL modifier.

```
**BCSET
  ID-Bcset DISP1
  VALUe
  GMR-bc GMR1
    SURFace-bc SEG1
    ELEMents-bc   21
    POINts-bc     23  34  45  57
    DISPlacement-bc 1
      SPLIst   ALL
      T-VAlue 1 0.0
```

3. Use of the SAME modifier.

```
**BCSET
  ID-Bcset DISP2
  VALUe
  GMR-bc GMR1
    SURFace-bc SEG1
    ELEMents-bc  211
    POINts-bc     34  37
    DISPlacement-bc 1
      SPLIst   34  37
      T-VAlue1 0.0
    DISPlacement-bc 2
      SPLIst SAME
      T-VAlue1 0.0
```

T-VAlue IT V1 V2 ... VN

Status - REQUIRED

Full Keyword - **T-VAlue**

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 TIMEs-bcset card in the definition of the surface for application of boundary condition (section 5.5.4 and 5.5.6). 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 SPLIst card.

Additional Information -

This input may continue for as many cards as required. Each additional card must begin with T-VAlue and the time point IT. The input for each new time point must begin on a new card.

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

If the card ‘‘AXISymmetry GENE’’ is specified in case input, then the variation of boundary conditions along the circumferential direction (in $22.5°$ intervals) must be input for every nodal point defined in the SPLIst card. The nodal input for the first time point takes the form of

```
T 1 V1A V1B V1C . . . . . . V1P
T 1 V2A V2B V2C . . . . . . V2P
. . . . . . . . . . . . . . . .
. . . . . . . . . . . . . . . .
. . . . . . . . . . . . . . . .
T 1 VNA VNB VNC . . . . . . VNP
```

where the number after the letter V corresponds to the node and the following letter corresponds to the angle A = $0°$, B = $22.5°$, C = $45°$, D = $67.5°$, E = $90°$, . . . , P = $180°$. (Repeat for next time point). If N1 = ALL on the SPLI card, then only a single circumferential variation of the variables 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.

Examples of Use -

Definition of Boundary Conditions

1. This example illustrates the ‘‘SPLIst ALL’’ option. Since this is a time-independant boundary condition, only one value needs to be specified on the ‘‘T’’ card.

```
**BCSET
  ID-Bcset U1FIX
  VALUe
  LOCAl
  GMR-bc REG2
  SURFace-bc SIDE1
  ELEMents-bc 45  78
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0
```

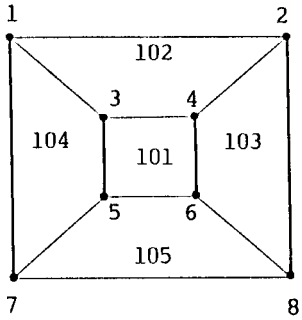
2. This example illustrates two **BCSET data blocks for a generalized axisymmetry problem (AXISymmetry-gmr GENE). The first data block is for a single node, whereas the second data block is for all nodes connected to element 2. Nine values on the ‘‘T’’ cards are needed to specify each 22.5° interval.

```
**BCSET
  ID-Bcset DISP1
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 1
    POINts-bc   12
    DISPlacement-bc 3
      SPLIst 12
      T-VAlue 1  0. 5.74025 1.06066E+1 1.38582E+1
      T-VAlue 1  1.5E+1 1.38582E+1 1.06066E+1 5.74025 0.

**BCSET
  ID-Bcset  TRAC3
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  2
    TRACtion-bc 3
      SPLIst ALL
      T-VAlue  1   .00000E+00 -.92808E+00 -.13125E+01 -.92808E+00  0.0
      T-VAlue  1   .92808E+00  .13125E+01  .92808E+00  0.0
```

3. Defines a complex impedance at a boundary for an acoustic analysis. Note that the LOAD COMPLEX card will appear in **CASE, and real and imaginary components must appear on the ‘‘T’’ card.

```
**BCSET
  ID-Bcset IMPBC
  RELAtion
  GMR-bc GMR1
  SURFace-bc  SURF1
  ELEMents-bc 12  13  15
  IMPEdance
    SPLIst ALL
    T-VAlue  1   25.4  67.8        $ Real and Imaginary values
```

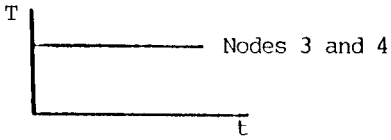
The results of various uses of the SPLIst and T-VAlue cards are shown in the following figure:



Note differences in traction input of nodes 3 and 4 over elements 101 and 102

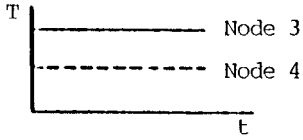
Case 1 Input

```
TIME   0.0
TRAC 1
SPLIST ALL
T 1  100.0
```



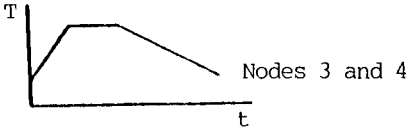
Case 2 Input

```
TIME   0.0
TRAC 1
SPLIST 3  4
T 1  100.0  50.0
```



Case 3 Input

```
TIME   0.0  1.0  2.0  4.0
TRAC 1
SPLIST ALL
T 1  50.0
T 2  150.0
T 3  150.0
T 4  50.0
```



Case 4 Input

```
TIME   0.0  1.0  2.0  4.0
TRAC   1
SPLIST 3  4
T 1  75.0   50.0
T 2  100.0  25.0
T 3  75.0   50.0
T 4  75.0   25.0
```

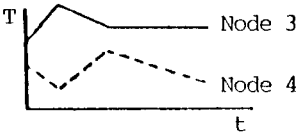


Figure for BCSET: TIME and SPLIST cards

TIME-SPACE variation

5.5.9 RELATION BOUNDARY CONDITION

SPRIng IDIR CONST DWALL

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 axisymmetric analysis:

1 - r direction (or z direction if ‘‘COORdinate-system ZR’’ appears in **CASE)

2 - z direction (or r direction if ‘‘COORdinate-system RZ’’ appears in **CASE)

3 - θ direction (for non-axisymmetric loading only)

For data defined in cylindrical coordinates:

1 - r direction

2 - θ direction

3 - z direction

For local coordinates:

1 - outer normal direction or direction 1 defined under ‘‘LOCAl EXPLICIT’’

2,3 - can only be used if defined under ‘‘LOCAl NORMAL’’ or ‘‘LOCAl EXPLICIT’’.

CONST (Real) - REQUIRED

Spring constant. (If complex, both real and imaginary parts of the spring constant)

DWALL (Real) - OPTIONAL

Displacement of wall support to which the spring is connected. (Not available in periodic analysis.)

Additional Information -

The SPRIng boundary condition utilizes the relationship:

$$t_i = K(d_i^{\mathrm{wall}} - d_i)$$

where:

K is the spring constant

d_i^{wall} is the displacement of the wall support to which the spring is connected

d_i is the surface displacement of the GMR in the ith direction

t_i is the traction in the ith direction

The SPRIng card can only be used if the RELAtion 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 modeled in **GPBEST** through their stiffnesses (i.e. spring constants).

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.

Note that the dimension of the spring constant K in **GPBEST** is traction/length or force/length^3 (NOT force/length as on a physical one-dimensional spring).

This card is not available for acoustics and structural-acoustics problems.

If the wall displacement is zero, then a value for DWALL does not need to be input. If **GPBEST** finds no input for DWALL, then a value of zero is assumed.

The spring coefficient must be time-independent

If a TIME-bcset card is **not** included in the current boundary condition set, then the wall displacement is time independent and DWALL (if non-zero) must be specified on the SPRIng card.

If a TIME-bcset card is included in the current **BCSET, then T-VAlue card(s) must follow the SPRIng card to define the time variation of DWALL.

No spatial variation of spring coefficient nor wall displacement is permitted within an individual **BCSET. Thus, a POINts-bc and POINts-line-bc cards are not applicable.

Examples of Use -

1. A spring boundary condition in a plane stress analysis is specified using the RELAtion card. Figure (a) illustrates the geometry. The user wants to impose a spring-type boundary condition on the free end of the cantilever beam. The desired spring stiffness is $K = 590 N/m$ where K relates force to the deflection of a physical 1-D spring. Since the beam height is $.5m$ and a unit width of $1m$, the spring force is distributed as a traction so a spring constant of $1180 N/m^3$ is used which relates the traction to the displacement of the element at each node. A close-up view of the resulting boundary condition is depicted in Figure (b).

```
**BCSET
  ID-Bcset SPRING1
  RELAtion
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc   5
    SPRIng  2  1180     $ Y-DIRECTION      K=1180 N/m3
```

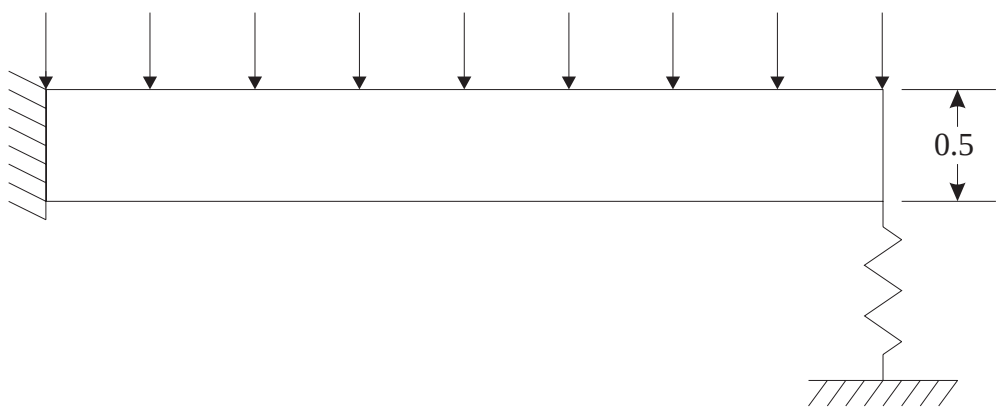


Figure for SPRIng Card: (a) Plane Stress Example

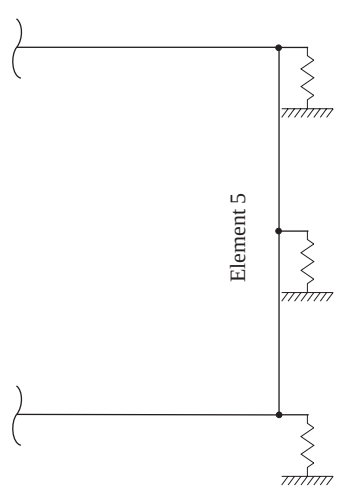


Figure for SPRIng Card: (b) Close-Up View of Spring Boundary Condition

2. A spring boundary condition is specified in which the wall support to which the spring is connected has a time-independent displacement of 0.1. If the spring constant is $1000N/m^3$, then the initial displacement is equivalent to a spring that is prestressed to a level of $100N/m^3$.

```
**BCSET
  ID-Bcset SPRING2
  RELAtion
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc   101     102     103
    SPRIng 3  1000.0  0.1
```

3. A local spring boundary condition is specified in which the wall support in the normal direction has a time-dependent displacement. Note, the spring constant is $1000N/m^3$ and the wall displacement will be 0.1 at time 1.0.

```
**BCSET
  ID-Bcset SPRING3
  RELAtion
  LOCAl
  GMR-bc GMR3
  SURFace-bc SURF3
    ELEMents-bc   301     302     303     304
    TIME-bcset      0.0     10.0
    SPRIng 1  1000.0
    T-VAlue  1        0.0
    T-VAlue  2        1.0
```

CONVection FCOEFF TAMBT
RESIstance GCOEFF TAMBT

Status - OPTIONAL

Full Keyword - **`CONVection`**

Full Keyword - **`RESIstance`**

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)

GCOEFF (Real) - REQUIRED

Resistance coefficient $(\frac{1}{h})$

TAMBT (Real) - OPTIONAL

Ambient temperature of convective fluid (T_a)

Additional Information -

FCOEFF is the reciprocal of GCOEF.

CONVection and RESIstance is only valid for HEAT, DIFFusion or CTHErmal analysis.

The CONVection and RESIstance cards can only be used if the RELAtion card has been input for the current boundary condition data set.

The CONVection and RESIstance option utilizes the relationship:

$$Q = -h * (T_a - T)$$

The film coefficient must be time-independent.

Q is positive if heat is leaving the body. Q is negative if heat is entering the body. Q is not specified per se, but its value is determined from h, T_a and the temperature at the surface, T.

If a TIME-bcset card is **not** included in the current boundary condition set, then the ambient temperature is time independent and TAMBT must be specified in the CONVection card.

If a TIME-bcset card was included in the current **BCSET, then T card(s) must follow the CONVection card to define the time variation of ambient temperature.

No spatial variation of film coefficient nor ambient temperature is permitted within an individual **BCSET. Thus, a POINts-bc and POINts-line-bc cards are not applicable.

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 fluid ambient temperature of 100 units. The figure illustrates the geometry.

```
**BCSET
  ID-Bcset  BCS1
  RELAtion
  GMR-bc  GMR1
  SURFace-bc SURF1
    ELEMents-bc  2
    CONVection      1.26  +100.0           $ h=1.26    Ta=100
```

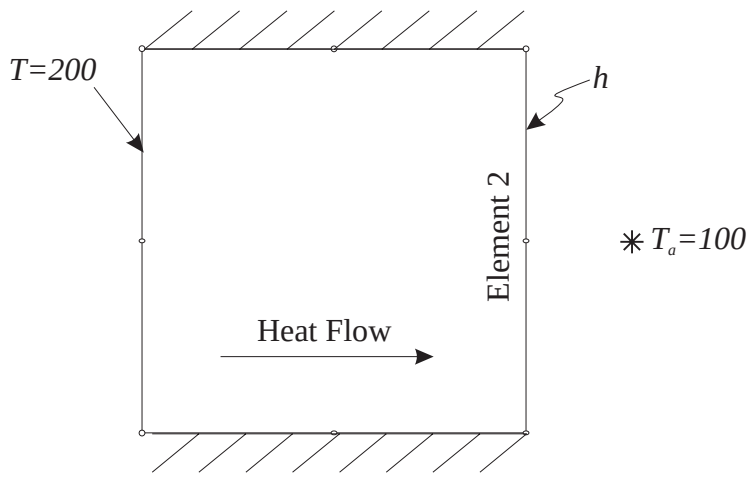


Figure for CONVection Card

2. Define a time-dependent convection boundary condition set, with a film coefficient of 10.43 units. The ambient temperature increases with time.

```
**BCSET
  ID-Bcset  CONV1
   RELAtion
   GMR-bc  GMR1
   SURFace-bc SURF1
     ELEMents-bc  1  2  14
     TIME-bcset 0.0 4.0 13.0 25.0
     CONVection      10.43                   $ H=10.43
     T-VAlue   1      0.0
     T-VAlue   2    100.0
     T-VAlue   3    200.0
     T-VAlue   4    300.0
```

IMPEdance R1

Status - OPTIONAL

Full Keyword - **`IMPEdance`**

Function - Identifies a boundary with an impedance condition applied. The velocity across this boundary is linearly related to the acoustic pressure.

Input Variables -

R1 - REQUIRED

Impedance coefficient (Z)

Additional Information -

The IMPEdance option utilizes the relationship:

$$v = \frac{1}{Z}p$$

where:

p	local acoustic pressure
v	local normal velocity
Z	impedance coefficient

The user is responsible for providing Z in the units consistent with the specification of material properties, geometry and boundary conditions.

If the impedance value is input as less than 1.0E-10, then it is set to 1.0E-10. If it is input as greater than 1.0E+10, then it is set to 1.0E+10.

This card is only relevant to periodic ACOUstic analysis.

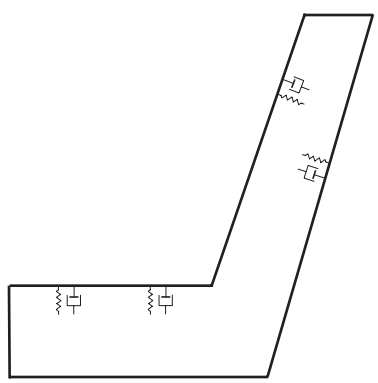


Figure of a car seat

An impedance boundary candition can be used to model the elastic and damping interaction between a structural part and an acoustic fluid. Take for instance the car seat shown above.

The impedance is defined by pressure over the normal velocity

$$Z(f) = \frac{p(f)}{v_n(f)} = \frac{F(f)}{A \cdot v_n(f)}$$

The frequency dependent force $F(f)$ can be expressed by the stiffness $c(f)$ and the damping coefficient $d(f)$ as

$$F(f) = c(f) \cdot u_n + d(f) \cdot v_n = \left(\frac{c(f)}{i2\pi f} + d(f) \right) v_n$$

in which the normal displacement is related to the normal velocity by $v_n = i2\pi u_n$, where f is the frequency in Hertz. This yeilds the acoustic impedance

$$Z(f) = \frac{d(f)}{A} - \frac{ic(f)}{2\pi A f}$$

Therefore, the real part of the impedance represents the damping of the surface and the imaginary part represents the elastic properties.

Examples of Use -

1. Defines a real value impedance at a boundary. Note that the LOAD COMPLEX card will not appear in **CASE.

```
**BCSET
  ID-Bcset IMPBC
  RELAtion
  GMR-bc GMR1
  SURFace-bc  SURF1
    ELEMents-bc 12  13  15
    IMPEdance 53.2             $ Real value only
```

2. Defines a complex impedance at a boundary. Note that the LOAD COMPLEX card will appear in **CASE.

```
**BCSET
  ID-Bcset IMPBC
  RELAtion
  GMR-bc GMR1
  SURFace-bc  SURF1
    ELEMents-bc 12  13  15
    IMPEdance  25.4  67.8      $ Real and Imaginary values
```

ADMIttance R1

Status - OPTIONAL

Full Keyword - **ADMIttance**

Function - Identifies a boundary with an admittance condition applied. The velocity across this boundary is linearly related to the acoustic pressure.

Input Variables -

R1 - REQUIRED

Admittance coefficient (A)

Additional Information -

The ADMIttance option utilizes the relationships:

$$v = Ap$$

$$A = \frac{1}{Z}$$

(A and Z are complex)

where:

p	local acoustic pressure
v	local normal velocity
A	admittance coefficient
Z	impedance coefficient

The user is responsible for providing A in the units consistent with the specification of material properties, geometry and boundary conditions.

If the admittance value is input as greater than 1.0E+10, then it is set to 1.0E+10. If it is input as less than 1.0E-10, then it is set to 1.0E-10.

This card is only relevant to periodic ACOUstic analysis.

Examples of Use -

1. Defines a real value admittance at a boundary. Note that the LOAD COMPLEX card will not appear in **CASE.

```
**BCSET
  ID-Bcset ADMIBC
  RELAtion
  GMR-bc GMR1
  SURFace-bc  SURF1
    ELEMents-bc 12  13  15
    ADMIttance    0.015         $ Real value only
```

5.5.10 MOVING BOUNDARY CONDITION

SOURce

Status - OPTIONAL

Full Keyword - **SOURce**

Function - Identifies a moving heat source boundary condition

Input Variables - NONE

Additional Information -

The SOURce card can only be used if the MOVIng card has been input for the current boundary condition set.

Currently this is only available for axisymmetric HEAT and CTHErmal analysis.

Information regarding the timing and magnitude of the heat source must be specified on the subsequent TIME-bcset and ENERgy-rate cards.

A positive value of the heat source energy-rate implies that heat is entering the region at the given point.

Examples of Use -

1. Specify a heat source which passes uniformly across element 5 during the first second of the analysis. The source is at node 9 at time 0.0, node 10 at time 0.5 and node 11 at time 1.0.

```
**BCSET
  ID-Bcset  TOOL01
  MOVIng
  GMR-bc  GMR1
    SURFace-bc  SRF1
    ELEMents-bc 5
    POINts-bc  9  10  11
  SOURce
    TIMEs-moving    0.0  0.5  1.0
    ENERgy-rate    1.0  1.0  1.0
```

TIMEs-moving T1 T2... TN

Status - REQUIRED

Full Keyword - **`TIMEs-moving`**

Function - Specifies the time at which the heat source passes each source point of the element.

Input Variables -

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

Time at which the heat source passes the points P1, P2, ... ,PN specified on the POINts-bc card included in the current **BCSET.

Additional Information -

The time must be specified in ascending value. That is, T1<T2<T3N...<TN. Thus, the **BCSET establishes the direction in which the heat source crosses the element. (The heat source may not pass through a portion of the element and then reverse direction.)

A value must be included for each source point specified on the POINts-bc card.

Examples of Use -

1. Define a constant heat source which moves non-uniformly along a three-noded quadratic element. The source reaches node 23 at time 5.0, node 22 at time 5.15, and node 21 at time 5.20.

```
**BCSET
  ID-Bcset  NONUNIF
  MOVIng
  GMR-bc  WORKPC
    SURFace-bc   OUTER
    ELEMents-bc   10
    POINts-bc    23  22  21
  SOURce
    TIMEs-moving   5.0  5.15  5.20
    ENERgy-rate   1.0  1.0   1.0
```

ENERgy-rate E1 E2 ... EN

Status - REQUIRED

Full Keyword - **`ENERgy-rate`**

Function - Specifies the energy-rate of the heat source as it passes each source point of the element.

Input Variables -

E1, E2, ... ,EN (Real) REQUIRED

Energy-rate of heat source as it passes the points P1, P2, ... ,PN specified on the POINts-bc card included in the current BCSET.

Additional Information -

A positive value of energy-rate implies that heat is entering the region at the given point.

A value must be included for each source point specified on the POINts-bc card.

Examples of Use -

1. Specify a heat source with increasing intensity as it moves along a quadratic element (number 5). Since the energy-rate is positive, heat is entering the region at the source point.

```
**BCSET
  ID-Bcset  TOOL01
  MOVIng
  GMR-bc  GMR2
    SURFace-bc  SURF2
    ELEMents-bc 5
    POINts-bc   9  10  11
  SOURce
    TIMEs-moving  0.0  0.1  0.2
    ENERgy-rate  1.0  2.0  3.0
```

2. Specify a moving heat source condition of constant strength but varying position. Notice how the heat source moves from one element to the next. Since the energy-rate is positive, heat is entering the region at the source point.

```
**BCSET
  ID-Bcset FLUX101
  MOVIng
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  10
    POINts-bc    101  102  103
  SOURce
    TIMEs-moving    0.00  2.50  5.00
    ENERgy-rate    6.2832   6.2832   6.2832

**BCSET
  ID-Bcset FLUX102
  MOVIng
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  11
    POINts-bc    103  104  105
  SOURce
    TIMEs-moving    5.00  7.50  10.00
    ENERgy-rate    6.2832   6.2832   6.2832
```

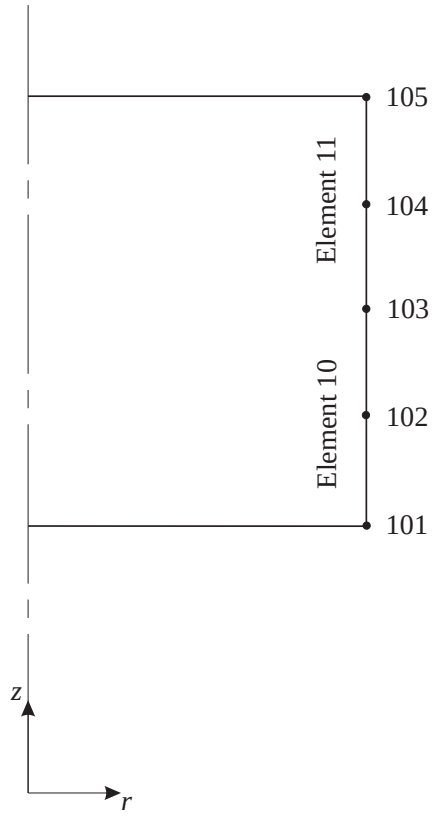


Figure for ENERgy-rate Card: Axisymmetric Model

5.5.11 NODE BASED BOUNDARY CONDITIONS

LOCAl-fem FEM ATYP
N1 V31 V32 V33 V11 V12 V13 V21 V22 V23

Status - OPTIONAL (applies to FE GMR only)

Full Keyword - **`LOCAl-fem FEM`**

Function - To define the direction cosines of the local axes at certain nodes for defining displacement or force along the local axes.

Input Variables -

ATYP (Alphanumeric) - REQUIRED

Allowable values are: DISP, SPRI or FORC

DISP - Nodal displacement constraints are described along the local axes

SPRI - Nodal springs are specified along the local axis.

FORC - Nodal forces act along the local axes

N1 (Integer) - REQUIRED

The node number where the direction cosines are being defined

V31, V32, V33 (Real) - REQUIRED

The global components of the z' axis (local z)

V11, V12, V13 (Real) - REQUIRED

The global components of the x' axis (local x)

V21, V22, V23 (Real) - REQUIRED

The global components of the y' axis (local y)

Additional Information -

This card is applicable only for a the structural portion of a structural-acoustics problem where the structural loads and constraints may be defined along local axes.

Nodal displacement boundary constraints for the structural part of a fully coupled structural-acoustics problem must be defined using LOCAl-fem boundary conditions. It is important that the local z-axis defines the true (or nearly true) normal at the node. The condensation of the structural system matrices to the normal translation depends entirely on the direction of the aforementioned nodal normal definition at the locations where the boundary conditions are defined. The nodal load vector, on the other hand, could be defined in either a GLOBAL or LOCAL coordinate system.

Examples of Use -

1. Define direction cosines for the local axes at nodes 2, 20 and 40.

```
**BCSET
  ID-Bcset  bc1  NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  LOCAl-fem FEM DISP
    2    0.0   0.0   1.0   1.0   0.0   0.0   0.0   1.0   0.0
   20    0.0  -0.707 0.707 1.0   0.0   0.0   0.0   0.707 0.707
   40    0.0  -1.0   0.0   1.0   0.0   0.0   0.0   0.0   1.0
```

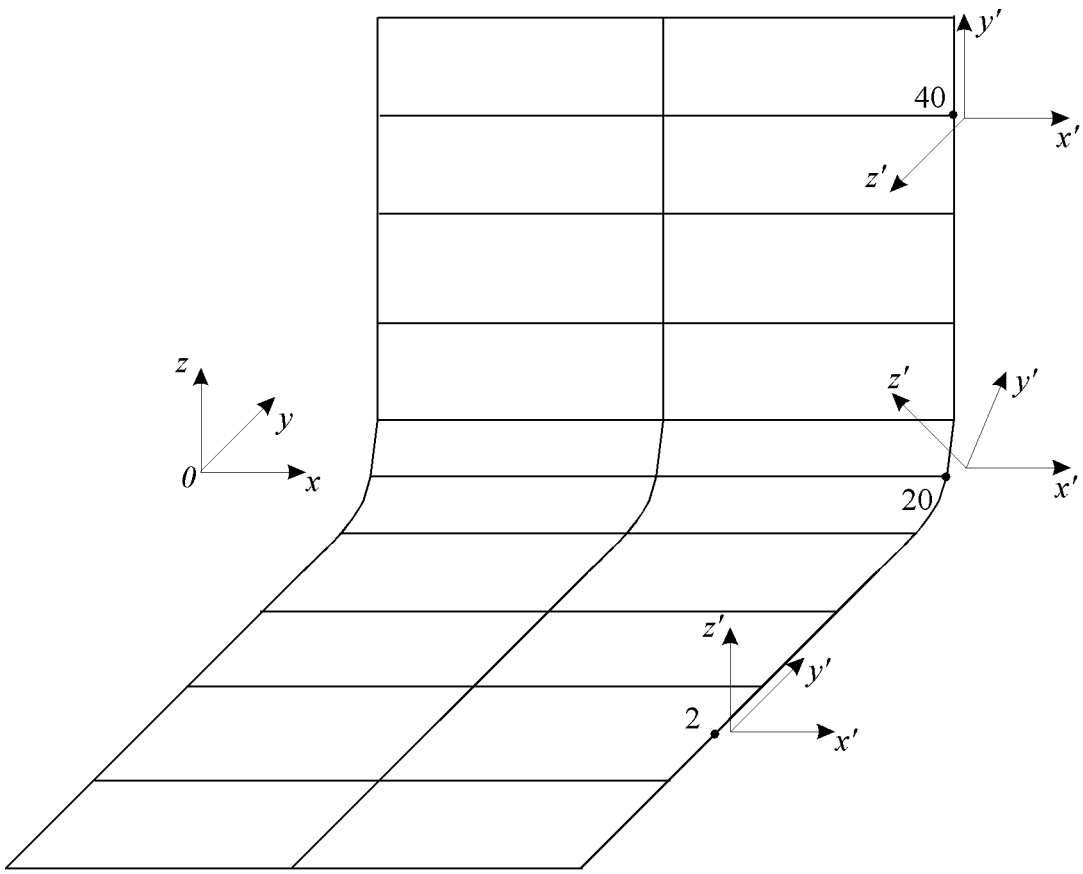


Figure for the LOCAl-fem FEM Card: Displacement BC

POINts-fem N1 N2 . . . NN

Status - OPTIONAL (applies to FE GMR only)

Full Keyword - **POINts-fem**

Function - Defines a list of nodes over which displacements and forces are defined.

Input Variables -

N1, N2,...,NN (Integer) REQUIRED

List of nodes over which the boundary conditions apply.

Additional Information -

This card is applicable only for a the structural portion of a structural-acoustics problem.

Examples of Input -

```
**BCSET
  ID-Bcset  bc1  NODAL
  GMR-bc   GMR1
  SURFace-bc  GMR1
  POINts-fem   2   20
```

DISPlacement-fem IDIR
T 1 value 1 [value 2] [value 3 value 4 . . .]

Status - OPTIONAL (applies to FE GMR only)

Full Keyword - **`DISPlacement-fem`**

Function - Defines the direction along which a displacement (translation or rotation) would be prescribed and their corresponding values.

Input Variables -

IDIR (Integer) - REQUIRED

Allowable values are 1 through 6

IDIR:

1 implies translation along x (or x') is prescribed
2 implies translation along y (or y') is prescribed
3 implies translation along z (or z') is prescribed
4 implies rotation along x (or x') is prescribed
5 implies rotation along y (or y') is prescribed
6 implies rotation along z (or z') is prescribed

where x, y, z are the global axes and x', y', z' are the local axes (if LOCAl-fem FEM DISP card is used).

value 1 [value 2] [value 3 value 4 . . .] (Real) - REQUIRED

The prescribed value is a complex quantity. Therefore, if only one node is declared in the POINts-fem card, two values are required.

value 1 corresponding to the real value
value 2 corresponding to the imaginary value

If value 2 is omitted, it is taken as zero.

If more than one node is declared in the POINts-fem card, then two sets of values would correspond to prescribed quantities at each node. If no value is given, then the value will be assumed to be that of the last-declared pair.

Additional Information -

This card is applicable only for the structural portion of a structural-acoustics problem where the nodal constraints have to be declared over the structure in the local or global direction.

Nodal displacement boundary constraints for the structural part of a fully coupled structural-acoustics problem must be defined using LOCAl-fem boundary conditions. It is important that the local z-axis defines the true (or nearly true) normal at the node. The condensation of the structural system matrices to the normal translation depends entirely on the direction of the aforementioned nodal normal definition at the locations

where the boundary conditions are defined. The nodal load vector, on the other hand, could be defined in either a GLOBAL or LOCAL coordinate system.

Examples of Use -

1. Define global fixity for nodes 2 and 10.

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  POINts-fem   2   10
  DISPlacement-fem    1
    T-VAlue     1   0.0
  DISPlacement-fem    2
    T-VAlue     1   0.0
  DISPlacement-fem    3
    T-VAlue     1   0.0
  DISPlacement-fem    4
    T-VAlue     1   0.0
  DISPlacement-fem    5
    T-VAlue     1   0.0
  DISPlacement-fem    6
    T-VAlue     1   0.0
```

2. Define a prescribed value of (0.2, 0.0) at node 2 and (0.2, 0.0) at node 10 for translation along the local x-axis.

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  LOCAl-fem FEM DISP
    2   0.0   0.0   1.0   0.71   0.71   0.0   -0.71   0.71   0.0
    10 -0.58 -0.06 -0.81  0.07   0.99   0.0   -0.81  -0.06   0.58
  POINts-fem   2   10
  DISPlacement-fem    1
    T-VAlue     1   0.2   0.0   0.2
```

3. The same definition given in Example 2 may be written as:

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  LOCAl-fem FEM DISP
    2   0.0   0.0   1.0   0.71   0.71   0.0   -0.71   0.71   0.0
    10 -0.58 -0.06 -0.81  0.07   0.99   0.0   -0.81  -0.06   0.58
  POINts-fem   2   10
  DISPlacement-fem    1
    T-VAlue     1   0.2
```

STIFfness-fem IDIR
T1 value 1 [value 2] [value 3 value 4 . . .]

Status - OPTIONAL (applies to FE GMR only)

Full Keyword - **`STIFfness-fem`**

Function - Defines the (local) direction along which a user-specified stiffness (translational and/or rotational) would be prescribed and their corresponding values.

Input Variables -

IDIR (Integer) REQUIRED

Allowable values are 1 through 6.

IDIR:

1 implies translational spring along x' is specified
2 implies translational spring along y' is specified
3 implies translational spring along z' is specified
4 implies rotational spring around x' is specified
5 implies rotational spring around y' is specified
6 implies rotational spring around z' is specified

where x', y', z' are the local axes defined through a LOCAl-fem FEM STIF section.

value 1 [value 2][value 3 value 4 . . .] (real) - REQUIRED

The specified spring constant is a complex quantity. Therefore, if only one node is declared in the POINts-fem card, two values are required:

value 1 corresponding to the real value
value 2 corresponding to the imaginary value

If value 2 is omitted, it is taken as zero.

If more than one node is declared in the POINts-fem card, then two sets of values would correspond to specified quantities at each node. If no value is given, then the value will be assumed to be that of the last- declared pair.

Additional Information -

This card is applicable only for the structural portion of a structural-acoustics problem where the nodal spring stiffness must be declared over the structure in the local direction.

Examples of Use -

1. Define a translational spring (stiffness=1.0E+5) at nodes 10 and 20.

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  LOCAl-fem FEM STIFFNESS
  10   0.0   0.0   1.0   1.0   0.0   0.0   0.0   1.0   0.0
  20   0.0   0.0   1.0   1.0   0.0   0.0   0.0   1.0   0.0
  POINts-fem   10   20
  STIFfness-fem   3
    T-VAlue     1   1.0E+5
```

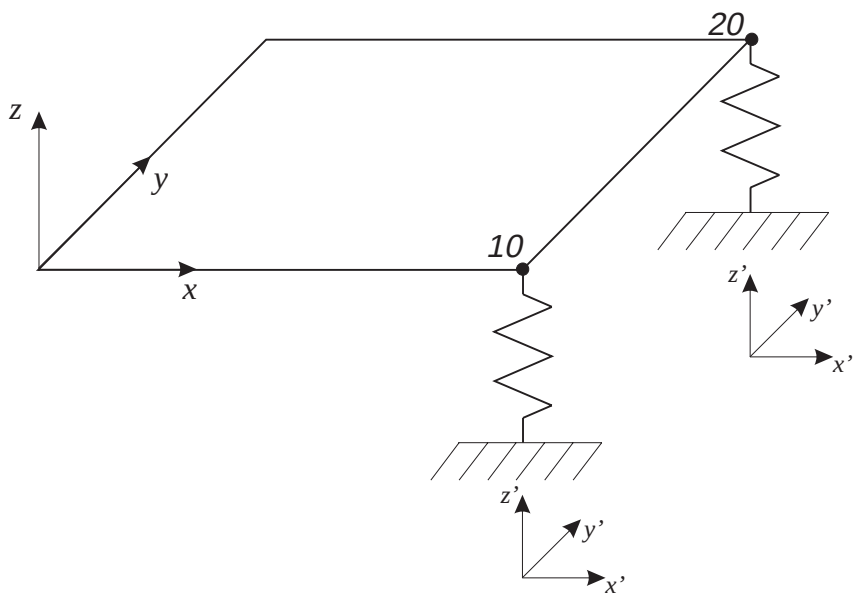


Figure for the STIFfness-fem Card: SPRIng BC

FORCe-fem IDIR
T1 value 1 [value 2] [value 3 value 4 . . .]

Status - OPTIONAL (applies to FE GMR only)

Full Keyword - **`FORCe-fem`**

Function - Defines the direction along which the forces (loads and moments) would be given and their corresponding values.

Input Variables -

IDIR (Integer) REQUIRED

Allowable values are 1 through 6.

IDIR:

1 implies load along x (or x')
2 implies load along y (or y')
3 implies load along z (or z')
4 implies moment along x (or x')
5 implies moment along y (or y')
6 implies moment along z (or z')

where x, y, z are the global axes and x', y', z' are the local axes (if LOCAl-fem FEM FORC card is used).

value 1 [value 2][value 3 value 4 . . .] (real) - REQUIRED

The force is a complex quantity. Therefore, if only one node is declared in the POINts-fem card, two values are required,

value 1 corresponding to the real value
value 2 corresponding to the imaginary value

If value 2 is omitted, it is taken as zero.

If more than one node is declared, then two sets of values would correspond to force quantities at each node. If no value is given, then it is assumed to be that of the last-declared pair.

Additional Information -

This card is applicable only for the structural portion of a structural-acoustics problem where the nodal forces must be declared over the structure in the local or global direction.

Examples of Use -

1. Define global force along z at nodes 2 and 10.

```
**BCSET
  ID-Bcset   bc2   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  POINts-fem   2   10
  FORCe-fem   3
    T-VAlue     1   100.0
```

2. Define a complex force (10.0, 5.0) at node 2 and (-10.0, -5.0) at node 10 in the local x-axis.

```
**BCSET
  ID-Bcset   bc2   NODAL
  GMR-bc  GMR1
  SURFace-bc GMR1
  LOCAl-fem FEM FORCE
    2   0.0   0.0   1.0   0.707 -0.707  0.0    0.707  0.707  0.0
   10   0.0   0.0   1.0   0.707 -0.707  0.0    0.707  0.707  0.0
  POINts-fem   2   10
  FORCe-fem   4
    T-VAlue     1   10.0   5.0   -10.0   -5.0
```

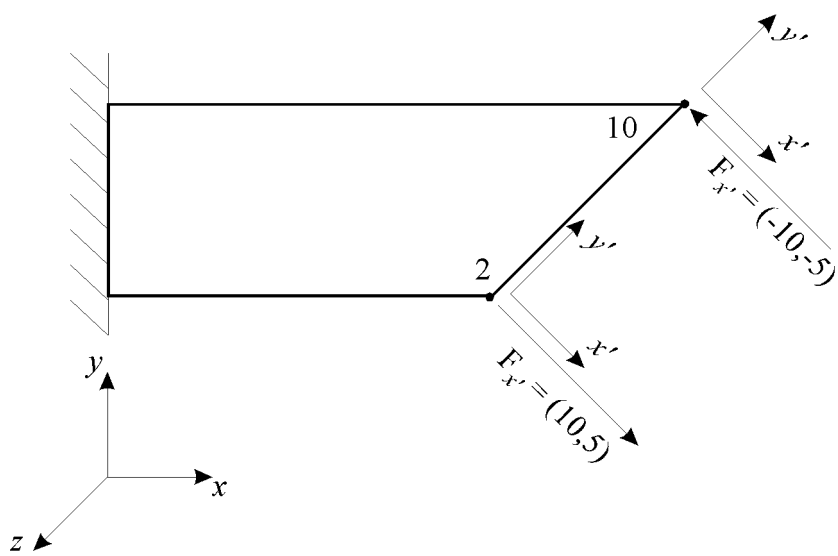


Figure for the FORCe-fem Card: Force BC

DROP-fem
T1 value 1 [value 2] [value 3 value 4 . . .]

Status - OPTIONAL (applies to indirect acoustic analysis only)

Full Keyword - **DROP-fem**

Function - Defines acoustic pressure drop at certain nodes.

Input Variables -

value 1 [value 2] [value 3 value 4 . . .] - (Real) - REQUIRED

The pressure drop is a complex quantity. Therefore, if only one node is declared in the POINts-fem card, two values are required.

value 1 corresponding to the real value
value 2 corresponding to the imaginary value

If value 2 is omitted, it is taken as zero.

If more than one node is declared, then two sets of values would correspond to drop quantities at each node.

For multiple nodes on the POINts-fem card and a single set (two values) of input value, it is assumed that the drop quantities for the other nodes are the same as the drop defined for the first node in the list.

Additional Information -

This card is applicable only for an acoustic GMR being solved by the Indirect boundary element formulation.

Examples of Use -

1. Define an acoustic pressure drop (0.0 0.0) at the nodes 17, 27 and 31.

```
**BCSET
  ID-Bcset   bc1   NODAL
  GMR-bc  g01
  SURFace-bc g01
  POINts-fem   17   27  31
  DROP-fem
    T-VAlue   1   0.0
```

5.6 BODY FORCE DEFINITION

SECTION	KEYWORD	PURPOSE	PAGE
5.6.1	Body Force Input Card		5.6.4
	**BODY	start of body force input	5.6.4
5.6.2	Centrifugal Body Force		5.6.6
	CENTrifugal	centrifugal load input	5.6.6
	POINt-centrifugal	point on the axis of rotation	5.6.7
	DIREction-centrifugal	direction of axis of rotation	5.6.9
	TIMEs-centrifugal	time for input	5.6.10
	SPEEd	speed input	5.6.11
5.6.3	Inertial Body Force		5.6.12
	INERtia-force	inertial body force input	5.6.12
	DIREction-inertia-force	direction of acceleration	5.6.13
	TIMEs-inertia-force	time of input	5.6.14
	ACCEleration-inertia	accleration input	5.6.15
5.6.4	Thermal Body Force		5.6.16
	THERmal	thermal body force input	5.6.16
	TIMEs-thermal	times for input	5.6.18
	GMR-thermal	identifies GMR	5.6.19
	TEMPeratures-thermal	start of temperature input for GMR	5.6.20
	(NONE)	(node temperatures)	5.6.22
	(TERM)	temperature coefficient cards	5.6.23
5.6.5	Generalized Plane Stress/Plane Strain Input		5.6.25
	GENEralized	generalized plane stress/strain input	5.6.25
	TIMEs-generalized	times for input	5.6.26
	GMR-generalized	identifies GMR	5.6.27
	CONStant	constant coefficient	5.6.28
	XVALue	X coefficient	5.6.29
	YVALue	Y coefficient	5.6.30

Body Force Definition

SECTION	KEYWORD	PURPOSE	PAGE
5.6.6	Point Source/Force		5.6.31
	CONCentrated	concentrated (point) source/force	5.6.31
	TIMEs-concentrated	times for input	5.6.33
	FREQuencies-concentrated	frequencies for input	5.6.33
	GMR-concentrated	identifies GMR	5.6.34
	POINts-concentrated	list of points at which values are specified	5.6.35
	SOURce-concentrated	source input	5.6.36
	FORCe-concentrated	force input	5.6.38
	T-COncentrated	values at points	5.6.40
5.6.7	Discontinuity Surface Input		5.6.41
	DISContinuity-surface	discontinuity surface input	5.6.41
	TIMEs-disc-surf	times for input	5.6.50
	FREQuencies-disc-surf	frequencies for input	5.6.51
	GMR-disc-surf	identifies GMR	5.6.52
	DISPlacement-disc-surf	start of displacement discontinuity input	5.6.53
	(NONE)	displacement discontinuity values	5.6.54

Modeling Body Forces

Loading and force input can be specified in **GPBEST** in two places: 1) along an edge or surface through boundary conditions, and 2) inside or throughout a body. The first type was discussed in Section 5.5 (Boundary Conditions). The latter is discussed in this section.

Body forces tend to act throughout a body or inside a body. A typical example of a body force is gravity, which acts throughout a mass. Even though the **GPBEST** model may consist only of surface modeling, the force due to gravity can be accounted for throughout the model. In **GPBEST** , all body forces are specified as functions of time, even for static analyses.

There are basically six kinds of body forces in **GPBEST** which apply to elastic problems:

1) Centrifugal body force. This is the force resulting from spinning a body about an arbitrary axis at a specified rotational speed (RPM). The entire body is affected by a centrifugal force proportional to $m\omega^2$. Since it is assumed that the entire body is spinning, centrifugal body forces act throughout the entire model, not just on specific GMRs.
2) Inertial body force. This is a force due to an acceleration (proportional to ma) in a specified direction. This can also represent gravitational loading on a static body, which is proportional to mg. Like centrifugal loading, inertial body forces are assumed to act throughout the entire model and cannot be specified on a GMR by GMR basis.
3) Thermal body force. For elastic analysis, a body force may arise due to a temperature distribution. These can be specified on selected GMRs.
4) Generalized plane stress and plane strain input. This is how loading can be accounted for in the third (z) direction of a planar elastic analysis. This can also be specified on a GMR by GMR basis.
5) Concentrated point source, or force. A force that occurs inside a body can be specified this way. Although acting at points in space, they are associated with body forces in how they are treated in the boundary element formulation.
6) Discontinuity element motion. For discontinuity elements only, the relative motion between its surfaces is specified using a body force input.

It is extremely important to use units on all body forces that are consistent with the geometry and material properties of the model.

For acoustic and structural-acoustic problems, the only type of body force available is an acoustic source which is a CONCentrated point source.

5.6.1 BODY FORCE INPUT CARD

**BODY

Status - OPTIONAL

Full Keyword - **BODY**

Function - Identifies the beginning of body force input.

Input Variables - NONE

Additional Information -

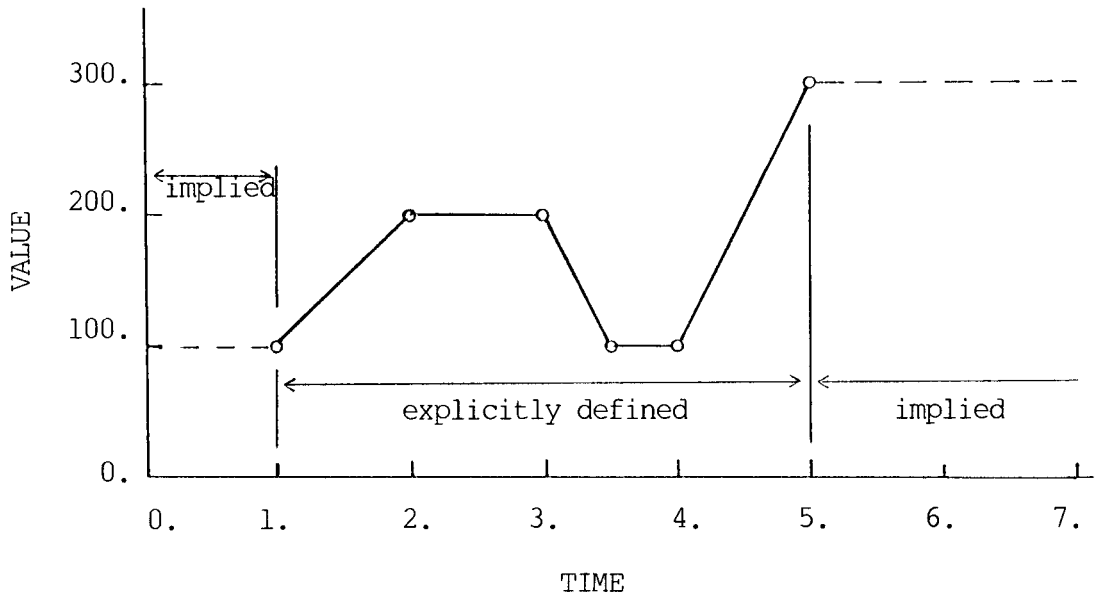
If more than type of body force is present, a separate block starting with **BODY should be defined for each type.

One must not use more than one **BODY card for body forces of one type (centrifugal or thermal or concentrated, etc.).

Examples of Use -

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

```
**BODY
  CENTrifugal
  DIREction-centrifugal  0.0  0.0  1.0
  POINt-centrifugal 0.0  0.0  0.0
  TIMEs-centrifugal  1.  2.  3.  4.
  SPEEd-centrifugal  45.  80.  100.  120.

**BODY
  THERmal
  TIMEs-thermal  0.  5.
  GMR  REG1
  TEMPeratures-thermal
    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

5.6.2 CENTRIFUGAL BODY FORCE

CENTrifugal

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.

In problems with centrifugal loading, the displacement takes on a cubic variation. For this reason, using a quartic functional variation (see TYPE-surface card in **GMR) will produce better results than a quadratic variation in models with crude discretization.

It is important to specify the proper mass density using the DENSity card in **MATERIAL for problems with centrifugal loading. If the DENSity card is not specified or an improper value is assigned, the centrifugal body force will not be correctly applied to the model.

Examples of Use -

1. Request a three-dimensional centrifugal input.

```
**BODY
  CENTrifugal
  DIREction-centrifugal  0.0  0.0  1.0
  POINt-centrifugal  0.0  0.0  0.0
  TIMEs-centrifugal   1.   2.   3.
  SPEEd-centrifugal  50.  150.  200.
```

POINt-centrifugal XP YP ZP

Status - OPTIONAL

Full Keyword - **`POINt-centrifugal`**

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

Input Variables -

XP,YP (Real) - REQUIRED (2-D analysis)

XP,YP,ZP (Real) - REQUIRED (3-D analysis)

Cartesian coordinates of a point on the axis of rotation.

GPBEST also accepts data defined using cylindrical coordinate systems. See the COORdinate-system system card under **CASE input.

Additional Information -

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

In axisymmetry, this reference point must lie on the z-axis (default). Therefore, the point card is not required.

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).

```
**BODY
  CENTrifugal
  POINt-centrifugal  1.  1.  0.
  DIREction-centrifugal   4.  4.  1.
  TIMEs-centrifugal   1.0
  SPEEd-centrifugal  50.0
```

2. Request a two-dimensional centrifugal input for rotation about point (5,3).

```
**BODY
  CENTrifugal
  POINt-centrifugal  5.0  3.0
  TIMEs-centrifugal   1.0
  SPEEd-centrifugal  100.0
$ (Note: DIREction-centrifugal card is not used for 2-D input)
```

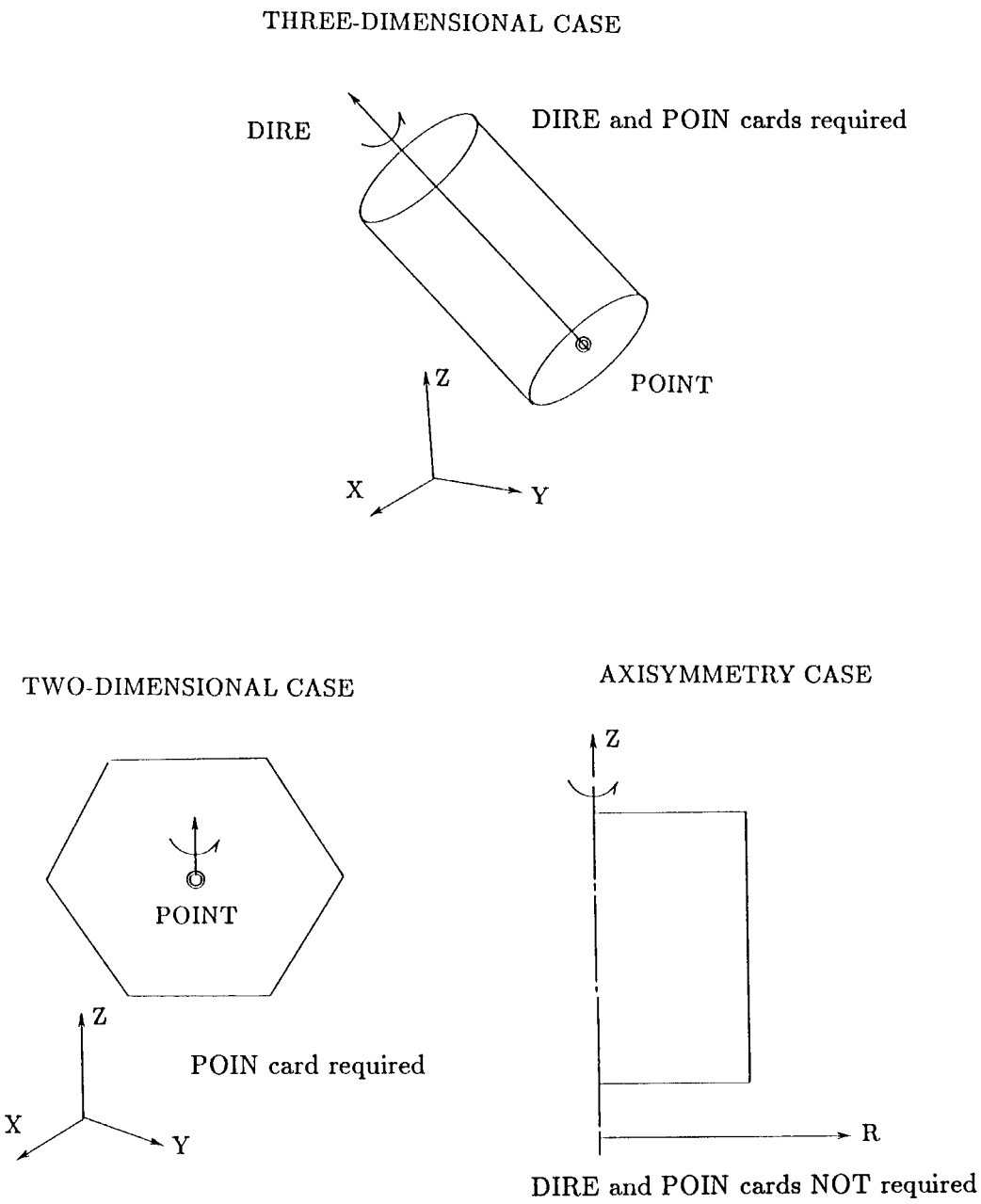


FIGURE FOR CENTRIFUGAL BODY FORCE
CENT, DIRE AND POIN CARDS

DIREction-centrifugal X Y Z

Status - OPTIONAL

Full Keyword - **DIREction-centrifugal**

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.

In 2-D plane stress and plane strain analyses, the direction of rotation must either be in the XY-plane (X,Y,0.0) i.e. Z=0.0 or be perpendicular to the XY-plane (0.0,0.0,1.0). The latter is the default value.

In axisymmetry, the axis of rotation must be parallel to the z axis (default). Therefore, the direction card is not required.

Examples of Use -

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

```
**BODY
    CENTrifugal
    POINt-centrifugal  0.0  0.0  0.0
    DIREction-centrifugal   1.0  0.0  0.0
    TIMEs-centrifugal   1.0
    SPEEd-centrifugal  100.
```

TIMEs-centrifugal T1 T2 ... TN

Status - REQUIRED (if CENTrifugal is input)

Full Keyword - **`TIMEs-centrifugal`**

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 TIMEs-centrifugal.

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
  CENTrifugal
  POINt-centrifugal  0.0  0.0  0.0
  DIREction-centrifugal   0.0  0.0  1.0
  TIMEs-centrifugal   0.0  2.0  4.0  6.0  10.0
  SPEEd-centrifugal  0.0  400.  500.  400.  0.
```

SPEEd-centrifugal OMEGA1 OMEGA2 .... OMEGAN

Status - REQUIRED

Full Keyword - **`SPEEd-centrifugal`**

Function - Defines the speed of rotation of the part.

Input Variables -

OMEGA1,OMEGA2,...,OMEGAN (Real) -REQUIRED

Speed of rotation (RPM) at times specified on TIMEs-centrifugal card.

Additional Information -

This card may be input as often as required. Each card begins with the keyword SPEEd-centrifugal.

The speed of rotation must be input in RPM.

(Note: 1 radian/second = 9.5493 RPM ($\frac{60}{2\pi}$ revolutions per minute)

Examples of Use -

1. Request an input for three-dimensional centrifugal case.

```
**BODY
  CENTrifugal
  POINt-centrifugal  0.0  0.0  0.0
  DIREction-centrifugal   0.0  0.0  1.0
  TIMEs-centrifugal   1.0  2.0  3.0
  SPEEd-centrifugal  100.  200.  400.
```

5.6.3 INERTIAL BODY FORCE

INERtia-force

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.

It is important to specify the proper mass density using the DENSity card in **MATERIAL for problems with inertial loading. If the DENSity card is not specified or an improper value is assigned, the inertial body force will not be correctly applied to the model.

Examples of Use -

1. Request a two-dimensional inertial input.

```
**BODY
  INERtia-force
  DIREction-inertia-force   1.0  0.0
  TIMEs-inertia-force   1.0
  ACCEleration-inertia   300.
```

DIREction-inertia-force X Y Z

Status - OPTIONAL

Full Keyword - **DIREction-inertia-force**

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

Input Variables -

X,Y (Real) - REQUIRED (2-D analysis)

X,Y,Z (Real) - REQUIRED (3-D analysis)

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).

In 2-D, the direction of the inertia force must lie in the x-y plane.

In axisymmetry, the direction of the inertia force must be the z-axis. Hence, this card should be omitted.

This direction vector is normalized inside GPBEST, i.e. it defines the direction of the inertia force only, the magnitude of the vector is not considered. Therefore, the magnitude of the inertia force is defined only by the value input on the ACCEleration card.

Examples of Use -

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

```
**BODY
  INERtia-force
  DIREction-inertia-force  0.  0.  1.
  TIMEs-inertia-force  1.0
  ACCEleration-inertia  10.0
```

TIMEs-inertia-force T1 T2 ... TN

Status - REQUIRED (if INERtia-force is input)

Full Keyword - **`TIMEs-inertia-force`**

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 TIMEs-inertia-force.

A maximum of 20 time values may be specified.

Examples of Use -

1. Specifies accelerations at three times.

```
**BODY
  INERtia-force
  DIREction-inertia-force  0.    0.    1.
  TIMEs-inertia-force  1.0   2.0   3.0
  ACCEleration-inertia  10.0  15.0  20.0
```

ACCEleration-inertia ACC1 ACC2 .... ACCN

Status - OPTIONAL

Full Keyword - **`ACCEleration-inertia`**

Function - Defines the total acceleration of the body.

Input Variables -

ACC1,ACC2,...,ACCN (Real) -REQUIRED

Acceleration at times specified on TIMEs-inertia-force card.

Additional Information -

This card may be input as often as required. Each card begins with the keyword ACCEleration-inertia.

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
  INERtia-force
  DIREction-inertia-force  0.   1.
  TIMEs-inertia-force  1.
  ACCEleration-inertia  100.
```

5.6.4 THERMAL BODY FORCE

THERmal

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 data block, even though more than one GMR may be involved.

This card is intended for use in elastic analysis with a thermal body force loading. It is not relevant to heat transfer (i.e., ‘‘HEAT’’ specified in **CASE input) or uncoupled thermal analysis (i.e., ‘‘CTHErm’’ specified in **CASE input).

The temperature distribution may be defined on a nodal basis or as a continuous function (a polynomial of up to quadratic order for each GMR).

In problems with a quadratic temperature distribution (CONTact or FIT-value on TEMPeratures-thermal card), the displacement takes on a cubic variation. For this reason, using a quartic functional variation (see TYPE card in **GMR) will produce better results than a quadratic variation in models with crude discretization.

It is important to specify the proper coefficient of thermal expansion using the ALPHa-temp-dep card in **MATERIAL for problems with thermal body forces. If the ALPHa-temp-dep card is not specified or an improper value is assigned, the thermal body force will not be correctly applied to the model.

Examples of Use -

1. Thermal input for a two-region problem at three time points. For nodal point input, see TEMPeratures-thermal card.

```
**BODY
  THERmal
  TIMEs-thermal  2.0  4.0  6.0
  GMR-thermal  REG1
  TEMPeratures-thermal  FIT
   1   100.0   200.0   500.0
   6   100.0   200.0   550.0
   5   100.0   200.0   525.0
           .       .       .
           .       .       .
  27   100.0   200.0   475.0
  GMR-thermal  REG2
  TEMPeratures-thermal  FIT
  40   100.0   300.0   500.0
  41   100.0   300.0   475.0
           .       .       .
           .       .       .
```

TIMEs-thermal T1 T2 .... TN

Status - REQUIRED (if THERmal is input)

Full Keyword - **`TIMEs-thermal`**

Function - Defines the times at which the nodal point temperatures or constants will be defined.

Input Variables -

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

Times at which nodal point temperatures or constants will be defined.

Additional Information -

If all times do not fit on one card, TIMEs-thermal may be continued on a second card immediately following the first time card, starting with the keyword TIMEs-thermal. Only one time definition is allowed for thermal input. 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
  THERmal
  TIMEs-thermal    1.0     2.0     3.0
  GMR-thermal  GMR1
  TEMPeratures-thermal  FIT
  1       100.0   200.0   400.0
  2       150.0   300.0   600.0
  3       200.0   400.0   800.0
              .       .       .
              .       .       .
              .       .       .
              .       .       .
```

GMR-thermal IDGMR

Status - REQUIRED (if THERmal is input)

Full Keyword - **`GMR-thermal`**

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-Gmr card in **GMR input).

Additional Information -

The GMR-thermal card is repeated for each GMR. All GMRs for which thermal input is desired should be contained under one **BODY input.

Examples of Use -

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

```
**BODY
  THERmal
  TIMEs-thermal   1.0
  GMR-thermal    REG1
```

TEMPeratures-thermal ITYPE DEPE

Status - REQUIRED (if THERmal is input)

Full Keyword - **`TEMPeratures-thermal`**

Function - Signals the beginning of input of temperatures for this GMR.

Input Variables -

ITYPE (Alphanumeric) - REQUIRED

Allowable values are CONTINUOUS or FIT

CONT - Temperatures are input as a spatial function in each GMR. Volume discretization is **NOT** required.

FIT - Temperatures are input at discrete nodal points and a quadratic curve fit is performed to approximate the temperature field in each GMR. Volume discretization is **NOT** required.

DEPEndent (Alphanumeric) - OPTIONAL (with FIT only)

Specifies a temperature-dependent coefficient of thermal expansion for this region.

Additional Information -

ITYPE must be the same for all GMRs in the problem.

This card is followed by data cards defining nodal point temperatures (if ITYPE = FIT) or functional variation (if ITYPE = CONT).

If DEPE appears after FIT, then a temperature dependent coefficient of thermal expansion is assumed for this region where the variation of alpha with temperature is defined in the material set (**MATERIAL) corresponding to this region. Otherwise, a contant value is used. The constant value is chosen at the integration temperature (TINTegration card in **GMR) of this region.

Examples of Use -

1. Defines constant temperature (50 degrees) distribution in the referenced GMR of an orthotropic medium.

```
**BODY
  THERmal
  TIMEs-thermal  1.0
  GMR-thermal   GMR1
  TEMPeratures-thermal CONTINUOUS
  C0   50.
```

2. Defines time-dependent temperatures at discrete points which will be used to generate two independent, quadratic least-squares-fit temperature distributions for two different GMR's.

```
**BODY
  THERmal
  TIMEs-thermal  1.0 2.0
  GMR-thermal   GMR1
  TEMPeratures-thermal FIT
    100    10.0  50.0
    110    10.0  50.0
    120    20.0  60.0
    160    25.0  90.0
  GMR-thermal   GMR2
  TEMPeratures-thermal FIT
    200    20.0  20.0
    210    20.0  30.0
    230    20.0  50.0
```

(NONE) NNODE TEM1 TEM2 ... TEMN

Status - REQUIRED (if ITYPE = FIT on TEMPeratures-thermal card)

Full Keyword - **NO KEYWORD REQUIRED**

Function - Defines nodal point temperatures at times specified on TIMEs-thermal 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 TIMEs-thermal card.

Additional Information -

This card is input as many times as is 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.

For ITYPE = FIT, Temperatures may be defined at any number of points desired by the user. No volume discretization is required. When this card is used, temperatures should be input at a sufficient number of points to get a ‘reasonable’ fit.

Examples of Use -

1. Defines a linear temperature distribution in GMR1, specified using ITYPE = FIT

```
**BODY
  THERmal
  TIMEs-thermal   1.0
  GMR-thermal    GMR1
  TEMPeratures-thermal FIT
    101  50.0
    110 100.0
```

(TERM)* **T1 T2 ... TN**

Status - OPTIONAL (IF ITYPE = CONT on TEMPeratures-thermal card, then at least one term card must be input)

Full Keyword -

Allowable keywords in 2-D and axisymmetry are C0, C1, C2, C11, C12, C22

Allowable keywords in 3-D are C0, C1, C2, C3, C11, C22, C33, C12, C13, C23

Function - Defines the value of the generic term in a continuous temperature distribution at the times specified on the TIMEs-thermal card. The temperature distribution is a polynomial up to the quadratic order. This can be expressed as
$T = C0 + C1 \cdot x_1 + C2 \cdot x_2 + C3 \cdot x_3 + C11 \cdot x_1 x_1 + C22 \cdot x_2 x_2 + C33 \cdot x_3 x_3 + C12 \cdot x_1 x_2 + C13 \cdot x_1 x_3 + C23 \cdot x_2 x_3$
where keywords are used to represent the values of the coefficients of the polynomial at a particular time.

Input Variables -

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

Values corresponding to the value of the specific term for time1, time2, ... timeN.

time1, time2, ..., timeN represent the times specified on the TIMEs-thermal card under the keyword THERmal.

Additional Information -

Each card is input as many times as is required to define the value for all times.

If a specific keyword (i.e. a specific term) is missing then a zero value is assumed for the corresponding term of the polynomial at all times for that GMR.

Volume discretization is not required for this type of temperature input !

*Note: ‘‘TERM’’ is a generic word (and should not be input) representing the keywords. The generic name ‘‘TERM’’ is only used to eliminate the redundancy of creating duplicate cards for each keyword. See example of use for clarity.

Examples of Use -

1. Defines discrete temperature distribution at previously defined nodes.

```
**BODY
  THERmal
    TIMEs-thermal   0.0   2.0   3.0   5.0
      GMR-thermal    GMR1
        TEMPeratures-thermal CONT
        C0     10.   30.   30.   70.
        C1     0.0   10.   10.   20.
        C22    0.0   0.0    2.    4.
      GMR-thermal    GMR2
        TEMPeratures-thermal CONT
        C0     100.  100.  100.  100.
```

Note: GMR1 has a quadratic temperature distribution in the x-direction which changes in time. GMR2 has a time-independent constant temperature throughout the entire GMR. All terms of the polynomial not specified are assumed zero for all times.

5.6.5 GENERALIZED PLANE STRESS/ PLANE STRAIN INPUT

GENEralized

Status - OPTIONAL (Available only in 2-D plane stress or plane strain analysis)

Full Keyword - **GENEralized** (plane stress or plane strain)

Function - Indicates the start of generalized plane stress or plane strain input.

Input Variables - NONE

Additional Information -

In a generalized plane stress (or plane strain) problem, an applied stress (or strain) of the form f = A+Bx+Cy is applied in the third (z) direction. If PLANe STRESS was specified in **CASE input, then the function represents the linear stress applied in the third direction. If PLANe STRAIN was specified in **CASE input, then the function represents linear strain applied in the third direction.

All generalized plane stress/plane strain input for an analysis should be input in a single **BODY data block, even though more than one GMR may be involved.

Examples of Use -

1. Generalized plane strain input for a two region problem at three time points. Note that the second GMR involves only a uniform strain for all times.

```
**BODY
  GENEralized
  TIMEs-generalized  2.0  4.0  6.0
  GMR-generalized  REG1
  CONStant  0.01   0.05   0.1
  XVALue  0.001  0.004  0.008
  YVALue  0.01   0.02   0.03
  GMR-generalized REG2
  CONStant  0.1  0.1  0.1
```

TIMEs-generalized T1 T2 .... TN

Status - REQUIRED (if GENEralized is input)

Full Keyword - **`TIMEs-generalized`**

Function - Defines the times at which the strains or stresses in the third (z) direction will be defined.

Input Variables -

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

Times at which strains or stresses in the third direction will be defined.

Additional Information -

If all times do not fit on one card, TIMEs-generalized 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 generalized plane stress/plane strain input. Therefore strains (or stresses) in each GMR should be defined according to this one definition.

A maximum of 20 time values may be specified.

Examples of Use -

1. Generalize plane stress for three time points.

```
**BODY
  GENEralized
  TIMEs-generalized  0.5   1.0   1.5
  GMR-generalized   PLATE
  CONSolidation  1000.0   2000.0   5000.0
  XVALue   100.0    100.0    100.0
  YVALue   100.0    200.0    300.0
```

GMR-generalized GMRNAME

Status - REQUIRED (if GENEralized is input)

Full Keyword - **GMR-generalized**

Function - Identifies the GMR in which strains or stresses in the third (z) direction will be defined.

Input Variables -

GMRNAME (Alphanumeric) - REQUIRED

Allowable values for GMRNAME are ID-Gmr or ALL.

ID-Gmr = the identifier of a specific GMR for which generalized plane stress/strain input will be defined (NAME on ID-Gmr card in **GMR input).

ALL = indicates the generalized plane stress/strain input of all the GMRs in the problem are identical and will be defined under one definition.

Additional Information -

If ALL is used as the argument of this card, then the GMR-generalized card is input only once.

If the generalized plane stress/strain input differs in a different region, then the GMR-generalized card (with ID-Gmr as the argument) must be repeated for every region with non-zero applied stress or strain in the third (z) direction.

All GMRs with non-zero generalized plane stress/plane strain input must be contained under a single **BODY input.

Examples of Use -

1. The following two examples yield the same result for a two-region problem.

```
**BODY
  GENEralized
  TIMEs-generalized  1.0
  GMR-generalized   REG1
    CONStant  100.0
    XVALue   50.0
    YVALue   50.0
  GMR-generalized   REG2
    CONStant  100.0
    XVALue   50.0
    YVALue   50.0

**BODY
  GENEralized
  TIMEs-generalized    1.0
  GMR-generalized     ALL
    CONStant  100.0
    XVALue   50.0
    YVALue   50.0
```

CONStant A1 A2 ... AN

Status - OPTIONAL

Full Keyword - **CONStant**

Function - Defines the constant coefficient A in the linear function $f = A + Bx + Cy$ defining an applied stress (for plane stress) or strain (for plane strain) in the third (z) direction (for the specified GMR) at the times specified on the TIME card.

Input Variables -

A1, A2,...AN (Real) - REQUIRED

Values corresponding to the constant term of the linear function of the third direction stress or strain at times specified on the TIME card.

Additional Information -

This card is input as many times as is required to define the value for all times.

If this card is missing for a specific GMR, then a zero value is assumed for this term at all times for that GMR.

If the card is input, then the number of values must be equal to the number of times specified on the TIMEs-generalized card.

Examples of Use -

```
**BODY
  GENEralized
  TIMEs-generalized  0.0  2.0
  GMR-generalized   ALL
    CONStant  100.0  200.0
    XVALue    0.0  100.0
    YVALue    0.0  200.0
```

XVALue B1 B2 ... BN

Status - OPTIONAL

Full Keyword - **XVALue**

Function - Defines the linear (x) coefficient B in the linear function $f = A + Bx + Cy$ defining an applied stress (for plane stress) or strain (for plane strain) in the third (z) direction for the specified GMR at the times specified on the TIME card.

Input Variables -

B1, B2,...BN (Real) - REQUIRED

Values corresponding to the linear (x) term of the linear function of the third direction stress or strain at times specified on the TIME card.

Additional Information -

This card is input as many times as is required to define the value for all times.

If this card is missing for a specific GMR, then a zero value is assumed for this term at all times for that GMR.

If the card is input, then the number of values must be exactly equal to the number of times specified on the TIMEs-generalized card.

Examples of Use -

```
**BODY
  GENEralized
  TIMEs-generalized  0.0  2.0
  GMR-generalized   ALL
    CONStant  100.0  200.0
    XVALue    0.0  100.0
    YVALue    0.0  200.0
```

YVALue C1 C2 ... CN

Status - OPTIONAL

Full Keyword - **YVALue**

Function - Defines the linear (y) coefficient C in the linear function $f = A + Bx + Cy$ defining an applied stress (for plane stress) or strain (for plane strain) in the third (z) direction for the specified GMR at the times specified on the TIME card.

Input Variables -

C1, C2,...CN (Real) - REQUIRED

Values corresponding to the linear (y) term of the linear function of the third direction stress or strain at times specified on the TIME card.

Additional Information -

This card is input as many times as is required to define the value for all times.

If this card is missing for a specific GMR, then a zero value is assumed for this term at all times for that GMR.

If the card is input, then the number of values must be exactly equal to the number of times specified on the TIMEs-generalized card.

Examples of Use -

```
**BODY
  GENEralized
  TIMEs-generalized  0.0  2.0
  GMR-generalized   ALL
    CONStant  100.0  200.0
    XVALue    0.0  100.0
    YVALue    0.0  200.0
```

5.6.6 POINT SOURCE/FORCE BODY FORCES

CONCentrated

Status - OPTIONAL

Full Keyword - **CONCentrated**

Function - Indicates the start of point source or force.

Input Variables - NONE

Additional Information -

All point source/force input for an analysis should be input in a single **BODY data block, even if more than one GMR may be involved.

For three-dimensional problems, this truly represents a point force or a source. For axisymmetric or two-dimensional problems, this represents a line load-per-unit length in the direction perpendicular to the two-dimensional cross-section.

Examples of Use -

1. Point source input for a two-region problem at two frequencies. For nodal point input, see SOURce-concentrated card.

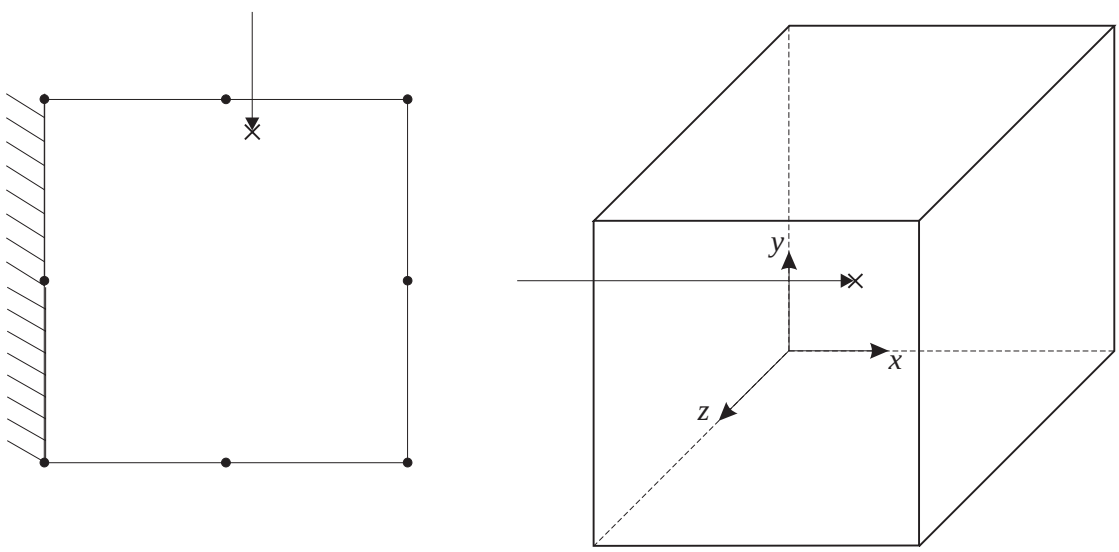
```
**BODY
  CONCentrated
  FREQuencies-concentrated  2.0  4.0
  GMR-concentrated  REG1
  POINts-concentrated 101 102 103
  SOURce-concentrated
   T-COncentrated  1   100.0   200.0   500.0
   T-COncentrated  2   100.0   200.0   550.0
  GMR-concentrated  REG2
  POINts-concentrated 104 106 107
  SOURce-concentrated
   T-COncentrated  1   100.0   300.0   500.0
   T-COncentrated  2   100.0   300.0   475.0
```

2. Point force input for a 2-D problem. Figure (a) illustrates the force acting in the -y direction inside the region.

```
**BODY
  CONCentrated
  TIMEs-concentrated 1.
  GMR-concentrated  REG1
  POINts-concentrated 101 $ Defined in **GMR block
  FORCe-concentrated 2
   T-COncentrated  1   -125.0
```

3. Point force input for a 3-D problem. Figure (b) illustrates the force acting in the x direction at the center of the cube.

```
**BODY
  CONCentrated
  TIMEs-concentrated 1.
  GMR-concentrated  REG1
  POINts-concentrated 201   $ Defined in **GMR block
  FORCe-concentrated 1
   T-COncentrated  1   35.0
```



(a) (b)

Figures for CONCentrated Card: (a) 2-Dimensional Model, (b) 3-Dimensional Model

TIMEs-concentrated T1 T2 .... TN
FREQuencies-concentrated T1 T2 .... TN

Status - REQUIRED

Full Keyword - **`TIMEs-concentrated`**

Full Keyword - **`FREQuencies-concentrated`**

Function - Defines the times or frequecies at which the nodal point source/force will be defined.

Input Variables -

Times or frequencies at which nodal point sources/forces will be defined.

Additional Information -

If all times and frequencies do not fit on one card, TIMEs-concentrated and FREQuencies-concentrated may be continued on a second card immediately following the first card, starting with the keyword TIMEs-concentrated or FREQuencies-concentrated.

Examples of Use -

1. Specifies three times at which point sources are input.

```
**BODY
  CONCentrated
  TIMEs-concentrated    1.0     2.0     3.0
  GMR-concentrated  GMR1
  POINts-concentrated 101 102
  SOURce-concentrated
  T-COncentrated  1       100.0   200.0
  T-COncentrated  2       150.0   300.0
  T-COncentrated  3       200.0   400.0
```

2. Specifies three frequencies at which point sources are input.

```
**BODY
  CONCentrated
  FREQuencies-concentrated    1.0     2.0     4.0
  GMR-concentrated  GMR1
  POINts-concentrated 101 102
  SOURce-concentrated
  T-COncentrated  1   0.0  1.0
  T-COncentrated  2   1.0  4.0
  T-COncentrated  3   1.0  3.0
```

GMR-concentrated IDGMR

Status - REQUIRED (if CONCentrated is input)

Full Keyword - **`GMR-concentrated`**

Function - Identifies the GMR in which point source/force will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

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

Additional Information -

The GMR-concentrated card is repeated for each GMR. All GMRs for which point source / force input is desired should be contained under one **BODY input.

Examples of Use -

1. Identifies GMR name at which point sources/forces are to be specified.

```
**BODY
  CONCentrated
  TIMEs-concentrated   1.0
  GMR-concentrated    REG1
```

POINts-concentrated P1 P2.....PN

Status - REQUIRED

Full Keyword - **POINts-concentrated**

Function - Defines the nodes at which point sources/forces will be defined.

Input Variables -

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

Nodes at which sources/forces will be defined.

Additional Information -

If all nodes do not fit on one card, POINts-concentrated may be continued on a second card immediately following the first card, starting with the keyword POINts-concentrated.

The coordinates of these must be defined in the POINts-concentrated block under **GMR input (not under SAMPling-points). These nodes should not coincide with boundary or sampling nodes.

Examples of Use -

1. Specifies three nodal values at a given time.

```
**BODY
  CONCentrated
  TIMEs-concentrated   1.0
  GMR-concentrated    REG1
  POINts-concentrated 101 102 103
  T-COncentrated  1 0.2 0.8 1.4
```

SOURce-concentrated

Status - REQUIRED (for scalar body forces)

Full Keyword - **SOURce-concentrated**

Function - Signals the beginning of input of sources for this GMR.

Input Variables - NONE

Additional Information -

For three-dimensional problems, this truly represents a point source or a force. For axisymmetric or two-dimensional problems, this represents a line load-per-unit length in the direction perpendicular to the two-dimensional cross-section.

In heat transfer problems, a positive source implies heat leaving the body.

Examples of Use -

1. Defines sources at four nodal points for two time steps.

```
**BODY
  CONCentrated
  TIMEs-concentrated  1.0 2.0
  GMR-concentrated   GMR1
  POINts-concentrated 101 102 103 104
  SOURce-concentrated
  T-COncentrated  1  10.0 50.0 60.0 80.0
  T-COncentrated  2  10.0 50.0 70.0 40.0
```

2. Defines a complex acoustic source at reference point 50001.

```
**BODY
  CONCentrated
  TIMEs-concentrated  1.0
  GMR-concentrated   GMR1
  POINts-concentrated 50001
  SOURce-concentrated
  $       Real    Imaginary
  T-COncentrated  1    2.5         0.
```

3. A 3-D steady heat source allows heat to enter the body at node 35.

```
**BODY
  CONCentrated
  STEADY
  GMR-concentrated   GMR1
  POINts-concentrated 35
  SOURce-concentrated
  T-COncentrated 1 -25.3 $ Negative value:heat entering the body
```

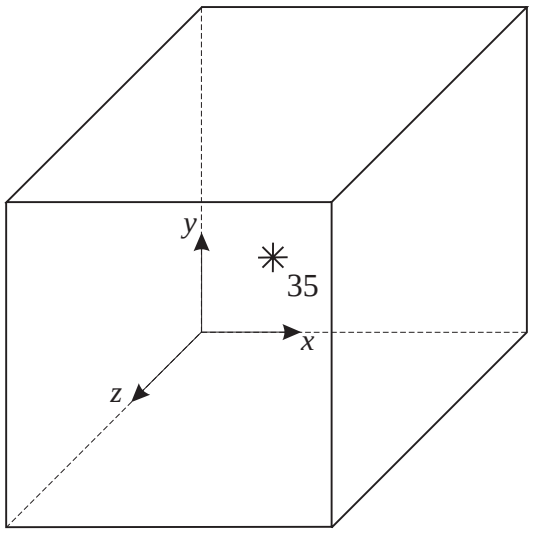


Figure for SOURce-concentrated Card: 3-D Heat Source

FORCe-concentrated ITYPE

Status - REQUIRED (for vector body forces)

Full Keyword - **`FORCe-concentrated`**

Function - Signals the beginning of input of sources for this GMR.

Input Variables -

ITYPE (Integer) -REQUIRED

ITYPE =1 if x-direction

ITYPE =2 if y-direction

ITYPE =3 if z-direction

For Axisymmetry

ITYPE =1 if r-direction (or z-direction if COORdinate-system ZR appears in **CASE)

ITYPE =2 if z-direction (or r-direction if COORdinate-system ZR appears in **CASE)

Additional Information -

For three-dimensional problems, this truly represents a point source or a force. For axisymmetric or two-dimensional problems, this represents a line load-per-unit length in the direction perpendicular to the two-dimensional cross-section.

Point forces have not been implemented for data sets with cylindrical input. However, point forces can be specified if the direction 1 is assumed to be the direction aligned with the $\theta = 0$ direction, and direction 3 corresponds to the direction aligned with the z axis. Direction 2 is defined by the cross-product of direction 3 with direction 1.

Examples of Use -

1. Defines force in x and y directions at three nodal points for two time steps.

```
**BODY
  CONCentrated
  TIMEs-concentrated  1.0 2.0
  GMR-concentrated   GMR1
  POINts-concentrated 101 102 103
  FORCe-concentrated 1
  T-COncentrated  1  0.0 2.0 3.0
  T-COncentrated  2  0.1 2.1 3.1
  FORCe-concentrated 2
  T-COncentrated  1  0.0 0.0 0.3
  T-COncentrated  2  2.0 3.0 1.0
```

2. Point force input for a 3-D problem. The force does not vary with time. Figure (a) illustrates the force acting in the x direction at the center of the cube.

```
**BODY
  CONCentrated
  TIMEs-concentrated 1.
  GMR-concentrated  REG1
  POINts-concentrated 201          $ Defined in **GMR block
  FORCe-concentrated 1
   T-COncentrated  1   35.0
```

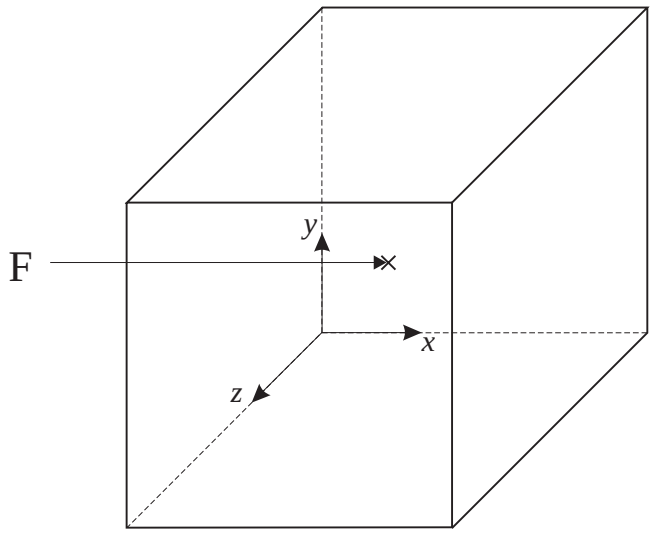


Figure for FORCe-concentrated Card: (a) Force Acting in x-direction

T-COncentrated IT V1 V2 ... VN

Status - REQUIRED

Full Keyword - **`T-COncentrated`**

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 and 5.5.6). 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 POINts card.

Additional Information -

This input may continue for as many cards as required. Each additional card must begin with T-COncentrated and the time point IT. The input for each new time point must begin on a new card.

Examples of Use -

1. Defines force in x and y directions at three nodal points for two time steps.

```
**BODY
  CONCentrated
  TIMEs-concentrated  1.0  2.0
  GMR-concentrated   GMR1
  POINts-concentrated 101 102 103
  FORCe-concentrated 1
  T-COncentrated  1  0.0 2.0
  T-COncentrated  1  3.0
  T-COncentrated  2  0.1 2.1 3.1
  FORCe-concentrated 2
  T-COncentrated  1  0.0 0.0 0.3
  T-COncentrated  2  2.0
  T-COncentrated  2  3.0
  T-COncentrated  2  1.0
```

5.6.7 DISCONTINUITY SURFACE INPUT

DISContinuity-surface

Status - OPTIONAL (Available only for elastostatic and dynamic analyses)

Full Keyword - **`DISContinuity-surface`**

Function - Identifies the start of displacement discontinuity input.

Input Variables - NONE

Additional Information -

Discontinuity elements must be defined in the geometry (**GMR) section. It is advisable to refer to Sections 2.12 and 5.3.7 to gain a fuller understanding of discontinuity elements.

The DISContinuity-surface option in the **BODY data block is used to specify the relative motion between the two resulting discontinuity surfaces. Since this motion is specified as a body force, there is no need to use a fine discretization. In many cases, a single discontinuity element can be used to model a straight crack.

This option is only available for 2-D and 3-D elastostatic analysis and dynamic analysis in frequency domain (periodic).

Discontinuity surfaces are not permitted in axisymmetric analysis.

Orientation of Discontinuity Elements -

The direction of relative motion for discontinuity elements is specified in the global X,Y,Z coordinate directions. However, it is also important to specify the directions in a positive (+) or negative (-) sense. For instance, the user may want to open up a crack instead of closing it. In this case, it is important to know how to specify the direction of motion.

The direction of relative motion for discontinuity elements depends on two factors: 1) the orientation of the element, as specified in the **GMR data block, and 2) the +/- sign on the cards following DISPlacement-disc-surf in **BODY.

A simple approach can be used when the motion is being specified on a single element in only one coordinate direction. In this case, the user can simply specify a + value on the cards following DISPlacement-disc-surf in **BODY and run the problem in **GPBEST** . The resulting deformation will indicate whether the direction is actually + or -. If it is incorrect, switch the sign and rerun the problem. If the model contains multiple discontinuity elements, then merely run the problem with one of them active. This will determine the correct direction for all other elements.

However, the user may wish to understand how the orientation of the discontinuity element affects the direction of motion. Figure 5.6.7.1 illustrates a two-dimensional

boundary element model with an interior discontinuity element. The positive normal direction for the outer boundary (S_1) is taken to be outward, as shown in the figure. This results in an integration path that is counter-clockwise on the outer boundary. (Section 5.3.5 discusses the positive normal direction in **GPBEST**). Viewing the discontinuity element as an interior ‘‘hole’’ in the region, its positive normal direction must be inward, and the direction of integration is clockwise.

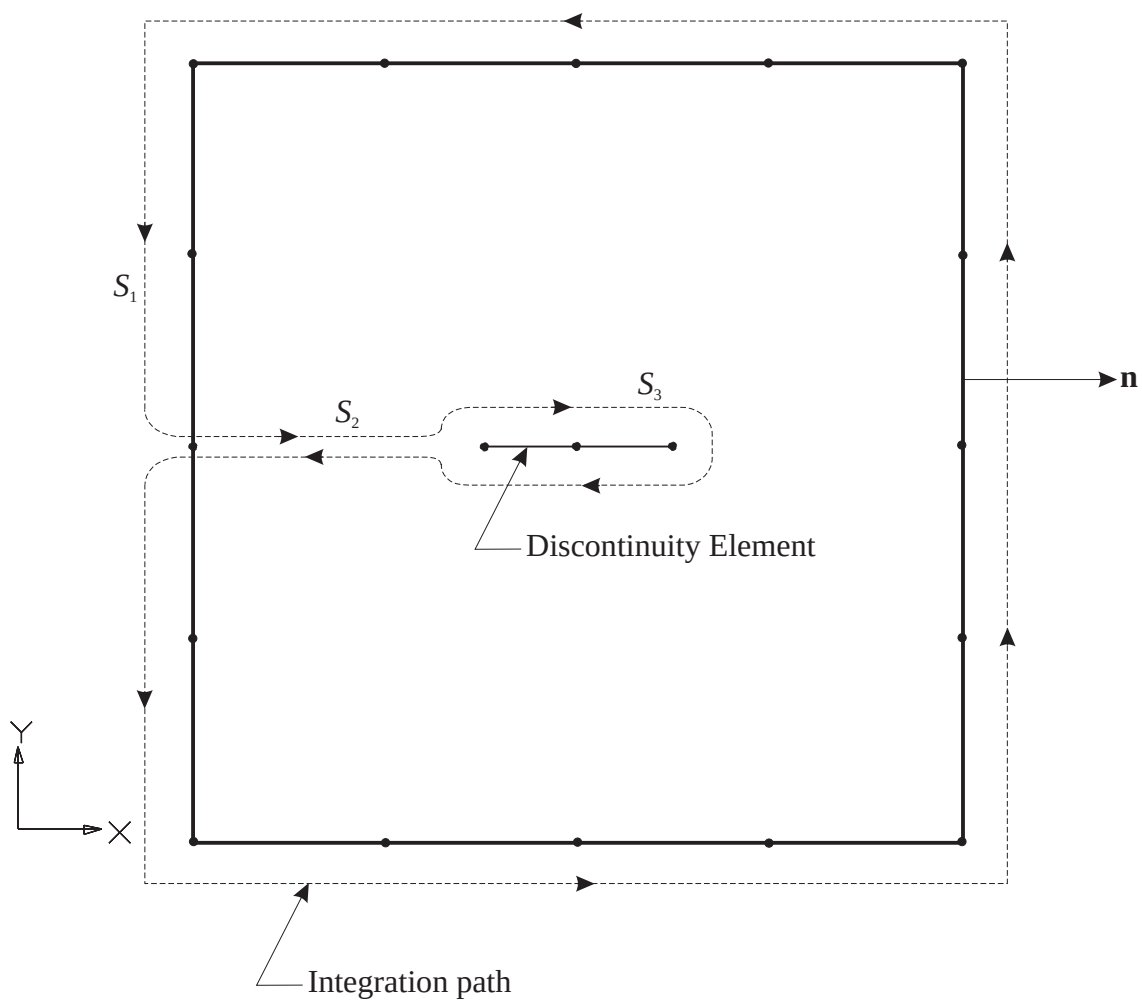


Figure 5.6.7.1 Two-Dimensional Model With a Discontinuity Element

A close-up of the discontinuity element region is depicted in Figure 5.6.7.2. What was specified initially to be a single element actually turns out to have two surfaces, $\mathcal{A}$ and $\mathcal{B}$. The relative motion between the two surfaces is specified on the cards after the DISPlacement-disc-surf card in **BODY, and is given by:

$$u_i^{\mathcal{B}} - u_i^{\mathcal{A}} = \Delta_i$$

where u_i is the displacement of surfaces $\mathcal{A}$ and $\mathcal{B}$, and Δ_i is the specified relative displacement. This relationship is specified at each set of corresponding nodes. For a three-dimensional analysis with one discontinuity element, there will be eight such relationships since there are eight nodes.

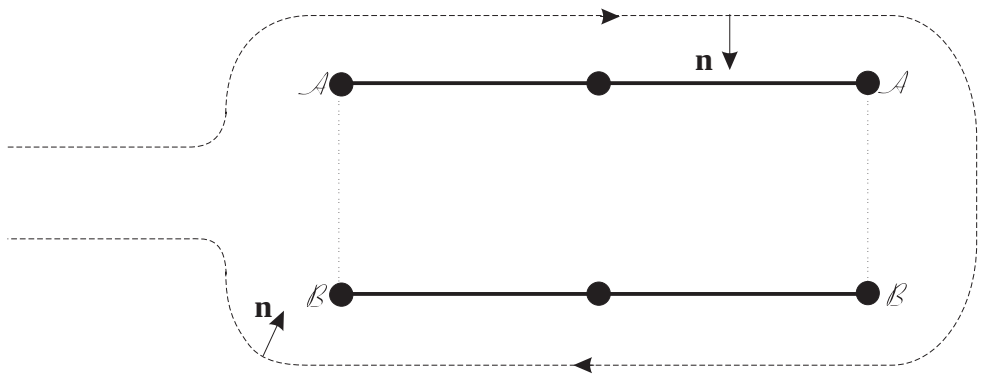


Figure 5.6.7.2 Close-up of Discontinuity Element

To illustrate, suppose the user wanted to specify a relative displacement of .1 mm in the x direction between the two surfaces of a discontinuity element. The relationship would be $u_1^{\mathcal{B}} - u_1^{\mathcal{A}} = 0.1$.

The only question remaining is how to determine which surface is $\mathcal{A}$ and which surface is $\mathcal{B}$. The rule is that the surface that results in a positive normal direction (or positive integration path) for the specified discontinuity element orientation is surface $\mathcal{A}$. The other surface is surface $\mathcal{B}$. The examples on the following pages should help clarify this rule.

A three-dimensional cube with an interior discontinuity element is displayed in Figure 5.6.7.3. In specifying the connectivity of the element, the user is also specifying an orientation. Figure 5.6.7.4 shows a close-up view of the discontinuity element region. Again, the two surfaces are illustrated as being separated by Δ_i. This figure illustrates the proper element orientations to result in an $inward$ normal direction. Whichever direction lines up with the user's specified orientation for the discontinuity element becomes surface $\mathcal{A}$ and the other one becomes surface $\mathcal{B}$. Again, the examples below should help the user determine the correct orientation.

Once it is determined which surface is $\mathcal{A}$ and which is $\mathcal{B}$, the desired sign on Δ in $u_i^{\mathcal{B}} - u_i^{\mathcal{A}} = \Delta_i$ is known.

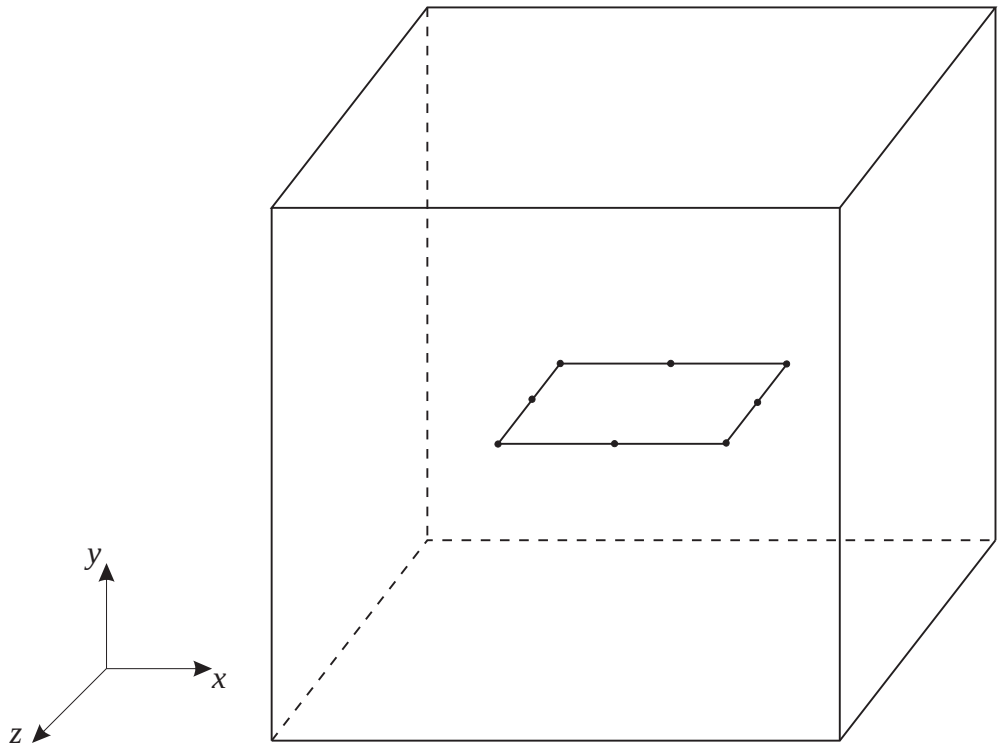


Figure 5.6.7.3 Three-Dimensional Model With a Discontinuity Element

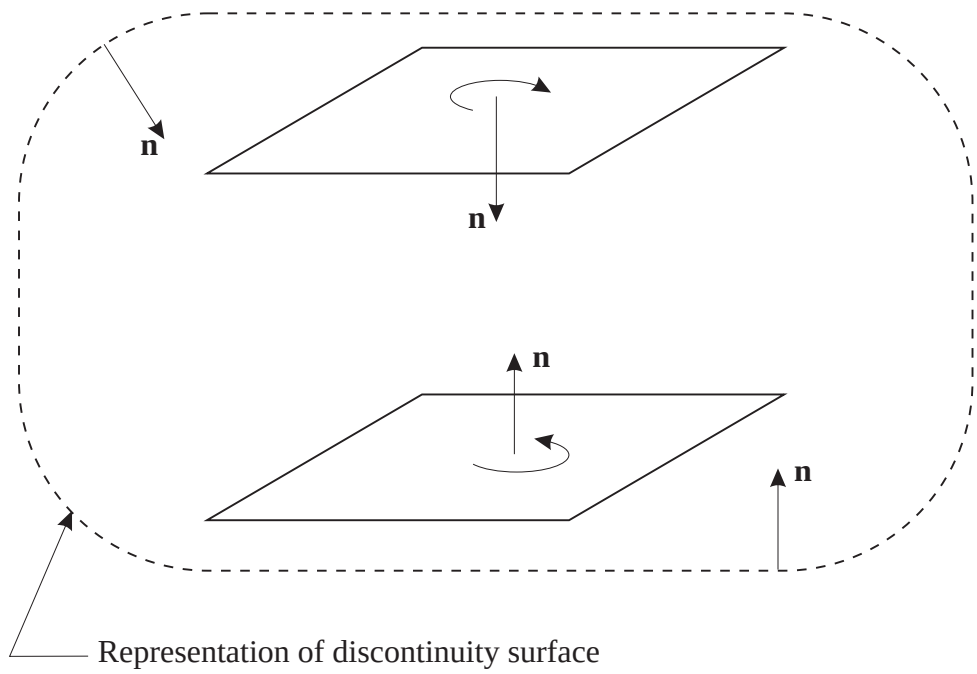


Figure 5.6.7.4 Close-up of Discontinuity Element

Examples of Use -

1. In order to illustrate how to specify a discontinuity surface in a two-dimensional (plane strain) problem, four variations on a single problem are presented here. In all cases, the connectivity is given in the following **GMR data block, where the location of nodes 51, 52 and 53 determine the direction of the resulting deformation. The relative displacements of the discontinuity surfaces are given by the following **BODY data block:

```
**GMR
  .
  .
  DISCcontinuity-elements   $3-NODE DISCONTINUITY ELEMENT
  30      51   52   53

**BODY
  DISContinuity-surface
  TIMEs-disc-surf1.0
  GMR-disc-surf GMR1
  DISPlacement-disc-surf    $  X      Y
   51       0.1    0.0
   52       0.1    0.0
   53       0.1    0.0
```

In Figure (a), the crack is shown parallel to the X-axis. The deformed shape plot shows the lower surface displacing in the positive X-direction, while the upper surface displaces in the opposite direction. The net displacement between the two surfaces is given by $u_i^{\mathcal{B}} - u_i^{\mathcal{A}} = \Delta_i$. Recall from the above discussion that the $\mathcal{A}$ surface is the one that makes the original element have a positive normal definition. Since a positive normal on an interior hole is directed inward (see Section 5.3.5), the integration path will be clockwise. Hence, the defined element here is surface $\mathcal{A}$, while the lower surface is $\mathcal{B}$.

This illustrates the common movement of subsurface cracks in earthquake and other seismic disturbance problems. Although the figure shows the two surfaces displacing in the Y-direction, this is just for illustration purposes. In fact, they do not have any relative displacement in the Y-direction, as specified in the **BODY (DISPlacement-disc-surf) data block.

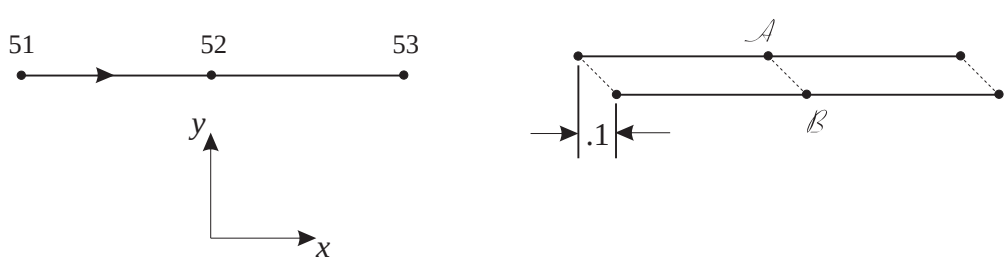


Figure for **BODY (DISPlacement-disc-surf) data block: (a) Parallel to X-axis

Figure (b) shows the crack at a 45 degree angle. Note that the relative motion between the two surfaces of the crack is still .1 in the X-direction.

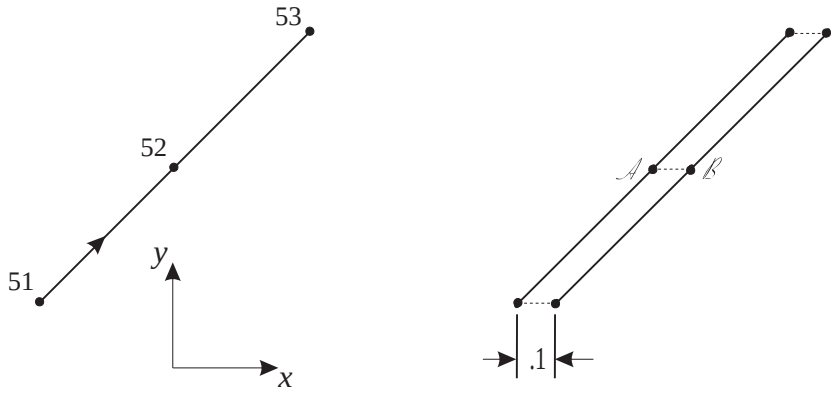


Figure for **BODY (DISPlacement-disc-surf) data block: (b) Diagonal Element

The discontinuity surface (crack) is shown along the Y-axis in Figure (c). Since the DISPlacement-disc-surf data block specified a relative movement of .1 in the X-direction, these element surfaces do not slide past each other, but instead, move away from each other.

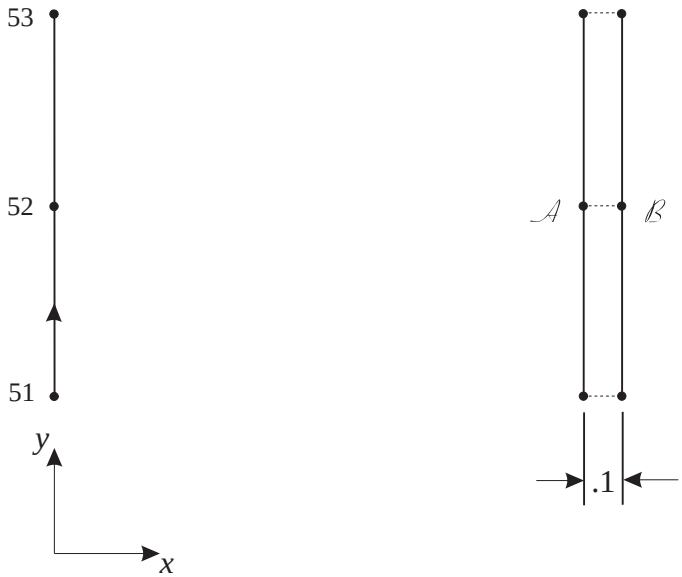


Figure for **BODY (DISPlacement-disc-surf) data block: (c) Parallel to Y-axis

Figure (d) is similar to Figure (a), except their direction of orientation are opposite to each other. Since the specified DISPlacement-disc-surf is the same, the result is that the lower surface in Figure (d) now moves in the -x-direction.

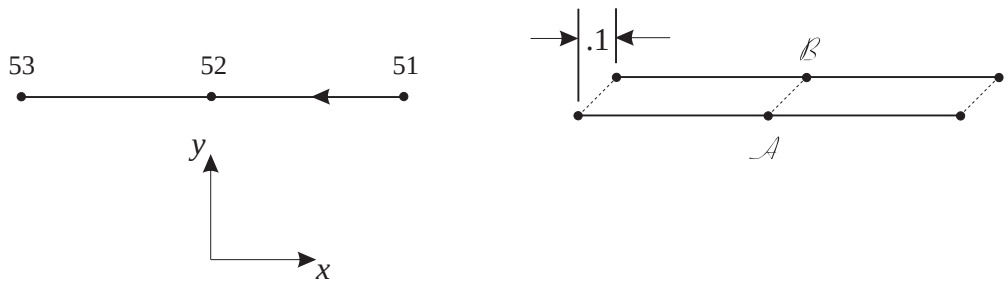


Figure for **BODY (DISPlacement-disc-surf) data block: (d) Parallel to -X-axis

These four examples illustrate several important points regarding discontinuity elements. First, the specified relative displacements are in the global X,Y and Z directions. Second, the direction of the relative displacement between the surfaces of the discontinuity element is determined by two factors: 1) The orientation of the discontinuity element, and 2) the direction specified in the **BODY (DISPlacement-disc-surf) data block.

2. This example illustrates a discontinuity element inside a three-dimensional cube. This is the 3-dimensional analog of the 2-dimensional problem in Figure (a). The geometry is depicted in Figure (e). Note in particular that the orientation of the element results in a normal vector that faces downward (in the -Z-direction). Since the normal direction inside an interior cavity (crack) should point inward, the given element must be the $\mathcal{A}$ (top) surface, while the lower surface is $\mathcal{B}$. Figure (f) illustrates a close-up view of the discontinuity element region, along with the deformed shape.

   The relevant **GMR block and **BODY data blocks are given here:

```
**GMR
  .
  .
  DISCcontinuity-elements      $ (ELEMENT 30 HAS 8 NODES)
    30    81   88   87   86   85   84   83   82

**BODY
  DISContinuity-surface
  TIMEs-disc-surf1.0
  GMR-disc-surf GMR1
  DISPlacement-disc-surf  $  X        Y        Z
   81     0.1      0.0      0.0
   82     0.1      0.0      0.0
   83     0.1      0.0      0.0
   84     0.1      0.0      0.0
   85     0.1      0.0      0.0
   86     0.1      0.0      0.0
   87     0.1      0.0      0.0
   88     0.1      0.0      0.0
```

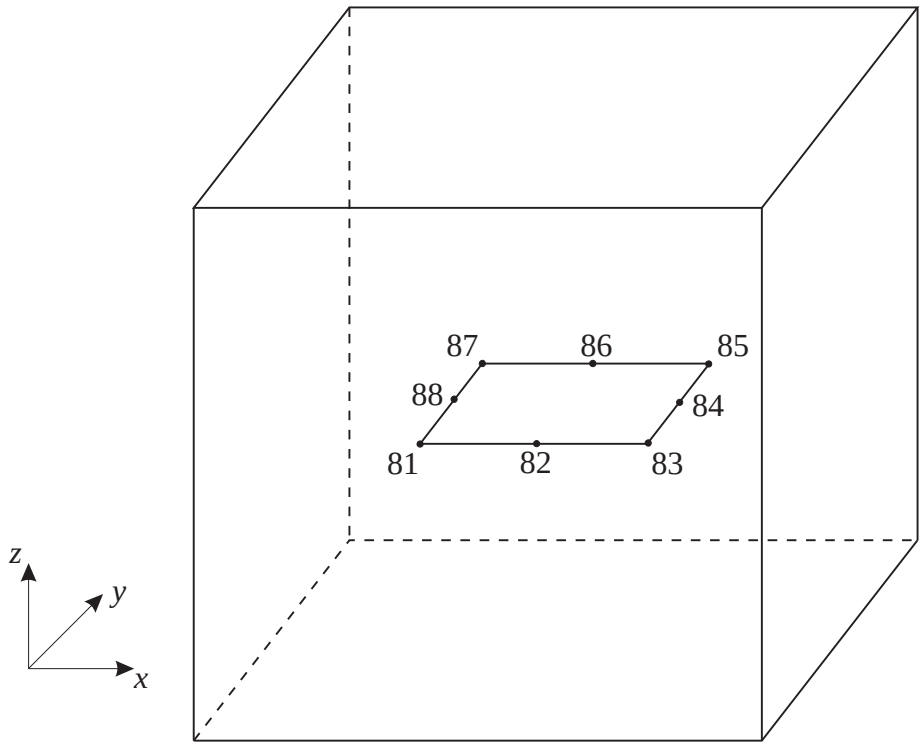


Figure for **BODY (DISPlacement-disc-surf) data block: (e) Cube With DISContinuity-surface

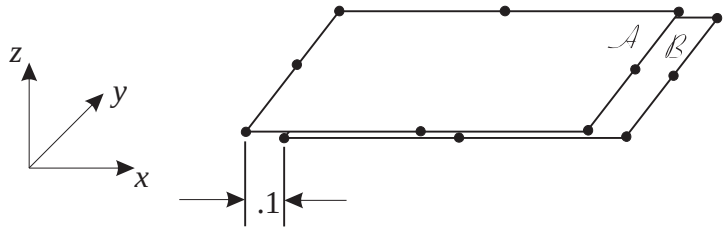


Figure for **BODY (DISPlacement-disc-surf) data block: (f) Close-up of DISContinuity-surface

TIMEs-disc-surf T1 T2 ... TN

Status - OPTIONAL (if DISContinuity-surface is specified)

Full Keyword - **`TIMEs-disc-surf`**

Function - Defines the times at which displacement discontinuities will be specified.

Input Variables -

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

Times at which displacement discontinuities will be specified.

Additional Information -

If all values do not fit on one card, input may be continued on the next line by starting with the keyword TIMEs-disc-surf.

Only one time definition is allowed for all discontinuity input for the entire problem. Therefore, displacement discontinuities in each GMR-disc-surf must be specified in accordance with this definition.

A maximum of 20 time values may be specified.

If a DISContinuity-surface body force is present, then either a TIMEs-disc-surf or FREQuencies card must appear.

Examples of Use -

1. Specify a discontinuity surface time history in a two-dimensional elastostatic problem.

```
**BODY
  DISContinuity-surface
  TIMEs-disc-surf   0.0    0.001  0.002
  TIMEs-disc-surf   0.005
  GMR-disc-surf GMR2 $--T1---  ---T2---  ---T3---  ---T4---
  DISPlacement-disc-surf $ X   Y   X   Y   X   Y   X   Y
    45     0.1  0.0  0.2  0.0  0.1  0.0  0.0  0.0
    46     0.15 0.0  0.3  0.0  0.15 0.0  0.0  0.0
    47     0.2  0.0  0.4  0.0  0.2  0.0  0.0  0.0
```

FREQuencies-disc-surf F1 F2 ... FN

Status - OPTIONAL (if DISContinuity-surface is specified)

Full Keyword - **FREQuencies-disc-surf**

Function - Defines the frequencies at which displacement discontinuities will be specified in a periodic analysis.

Input Variables -

F1, F2, ..., FN (Real) - REQUIRED

Frequencies at which displacement discontinuities will be specified.

Additional Information -

If all values do not fit on one card, input may be continued on the next line by starting with the keyword FREQuencies-disc-surf.

Only one frequency definition is allowed for all discontinuity input for the entire problem. Therefore, displacement discontinuities in each GMR-disc-surf must be specified in accordance with this definition.

A maximum of 20 frequency values may be specified.

The interpolation/extrapolation rules for frequency are identical to those used for time as shown in the figure for **BODY.

If a DISContinuity-surface body force is present, then either a FREQuencies-disc-surf or TIMEs-disc-surf card must appear.

The units on frequency is (rad/time).

Examples of Use -

1. Define the displacement discontinuity at a frequency of 8.5.

```
**BODY
  DISContinuity-surface
  FREQuencies-disc-surf 8.5
  GMR-disc-surf GMR1
  DISPlacement-disc-surf    $    X      Y      Z
    1001     0.000  1.000  0.000
    1002     0.000  1.000  0.000
    1003     0.000  1.000  0.000
```

GMR-disc-surf IDGMR

Status - REQUIRED (if DISContinuity-surface is input)

Full Keyword - **GMR-disc-surf**

Function - Identifies the GMR in which displacement discontinuities will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

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

Additional Information -

The GMR-disc-surf card is required for each GMR. All GMRs for which displacement discontinuity input is desired should be contained under a single **BODY input.

Examples of Use -

1. Identify multiple GMRs for displacement discontinuity data.

```
**BODY
  DISContinuity-surface
  FREQuencies-disc-surf 4.5
  GMR-disc-surf GMR1
    DISPlacement-disc-surf
      .
      .
      .
      .
  GMR-disc-surf GMR2
    DISPlacement-disc-surf
      .
      .
      .
      .
```

DISPlacement-disc-surf

Status - REQUIRED (if DISContinuity-surface is input)

Full Keyword - **`DISPlacement-disc-surf`**

Function - Signals the beginning of displacement discontinuities for this GMR

Input Variables - NONE

Additional Information -

This card must be followed immediately by data cards defining nodal point displacement discontinuities.

Examples of Use -

1. Include a surface of discontinuity in a three-dimensional dynamic analysis.

```
**BODY
  DISContinuity-surface
  FREQuencies-disc-surf  2.3   3.1
  GMR-disc-surf GMR1 $ -----FREQ 1-----   ---FREQ 2----
  DISPlacement-disc-surf $ X   Y   Z     X   Y   Z
    1001     0.0000  0.0010  0.0020   0.0000  0.0030  0.0010
    1002     0.0000  0.0010  0.0020   0.0000  0.0030  0.0010
    1003     0.0000  0.0010  0.0020   0.0000  0.0030  0.0010
```

(NONE) NNODE D1 D2 ... DN

Status - REQUIRED (if DISContinuity-surface is specified)

Full Keyword - **NO KEYWORD REQUIRED**

Function - Defines nodal point displacement discontinuities at times or frequencies specified on TIMEs-disc-surf or FREQuencies-disc-surf card, respectively.

Input Variables -

NNODE (Integer) - REQUIRED

User nodal point number for which displacement discontinuities are specified.

D1, D2,...,DN (Real) - REQUIRED

The displacement discontinuity at NNODE at the times or frequencies specified on the TIMEs-disc-surf or FREQuencies-disc-surf card.

Additional Information -

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

For each elastic analysis, the specification must include a value for each degree of freedom at a node. In a 3-D periodic dynamic problem, the first three values would correspond to the X, Y and Z displacement discontinuity at the first frequency. This would be followed by three values for the second frequency, and so on.

The relative motions are specified in the *global* X, Y and Z coordinate directions.

Discontinuity elements are not allowed in a data set that is defined with cylindrical coordinates.

For periodic analysis, complex displacement discontinuities may be input by including the LOAD COMPLEX card in the **CASE section. If that card is included, then complex body force nodal values must be supplied. Thus, for each degree of freedom at each frequency point, two values are required which would represent the real and imaginary components.

Examples of Use -

1. Specify an elastodynamic discontinuity surface at two frequencies.

```
**BODY
  DISContinuity-surface
  FREQuencies-disc-surf    9.32  6.45
  GMR-disc-surf GMR2 $ ---FREQ1---  ---FREQ2---
  DISPlacement-disc-surf $       X    Y       X    Y
    45        0.10 0.00    0.20 0.00
    46        0.15 0.00    0.30 0.00
```

5.7 DEFINITION OF BOUNDARY CONDITION CHANGES

GPBEST allows for a change of one boundary condition set to another boundary condition set 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 neccessary. This is useful, for instance, in a heat transfer analysis when a body is initially insulated on a side and then changed later in the run to allow a temperature flow across that same face. Interface conditions may also be modified.

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 interface and boundary conditions (with changes in the specified knowns and unknowns) are input, preceded by the **BCCHANGE and TIMEs-bcch card, after the initial definition.

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

This section does not apply for structural-acoustics problems.

SECTION	KEYWORD	PURPOSE	PAGE
5.7.1	Changes in Boundary Condition	Knowns and Unknowns	5.7.2
	**BCCHANGE	start of the new B.C. definition	5.7.2
	TIMEs-bcch	defines the time the change will occur	5.7.4

5.7.1 BOUNDARY CONDITION CHANGE CARD

**BCCHANGE

Status - OPTIONAL (Available only for transient heat conduction)

Full Keyword - **BCCHANGE**

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

Input Variables - NONE

Additional Information -

A TIMEs-bcch card must immediately follow the **BCCHANGE card.

An entire new set of interface and boundary conditions must then be defined following the TIMEs-bcch card. If certain **INTERFACE sets and **BCSETs do not change, these should simply be repeated after each **BCCHANGE card.

All interface and boundary condition sets preceding the **BCCHANGE card are forgotten once the TIMEs-bcch of solution has passed over to the next set of boundary conditions after the **BCCHANGE card.

Boundary condition changes are only available for transient heat conduction.

Examples of Use -

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

```
**BCSET
  ID-Bcset U1FIX
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE1
    TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset FLUX2
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE2
    FLUX-bc
    SPLIst ALL
    T-VAlue 1 1.0
```

```
**BCCHANGE
  TIMEs-bcch  10.0
$ CHANGE FROM TEMP TO FLUX SPECIFIED:
**BCSET
  ID-Bcset U1FIX
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE1
    FLUX-bc
    SPLIst ALL
    T-VAlue 1 0.0
$ UNCHANGED BCSET:
**BCSET
  ID-Bcset FLUX2
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE2
    FLUX-bc
    SPLIst ALL
    T-VAlue 1 1.0

**BCCHANGE
  TIMEs-bcch  20.0
$ CHANGE FROM FLUX TO TEMP SPECIFIED:
**BCSET
  ID-Bcset U1FIX
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE1
    TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1 0.0
$ UNCHANGED BCSET:
**BCSET
  ID-Bcset FLUX2
  VALUe
  GMR-bc GMR1
    SURFace-bc SIDE2
    FLUX-bc
    SPLIst ALL
    T-VAlue 1 1.0
```

TIMEs-bcch VALUE

Status - REQUIRED (if **BCCHANGE is input)

Full Keyword - **`TIMEs-bcch`**

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 TIMEs-bcch card.

The change occurs for the first solution time (specified in the case input) greater than the VALUE specified on the TIMEs-bcch card of the **BCCHANGE and less than the VALUE specified on the TIMEs-bcch cards of the subsequent **BCCHANGE changes.

If more than one **BCCHANGE card is present, the VALUE of time on the subsequent cards should be in ascending order.

The times specified on the TIMEs-bc card within a **BCSET may unneccessarily extend over the range allowed by the TIMEs-bcch card of the **BCCHANGE change. In any case, the TIMEs-bcch card for **BCCHANGE has priority.

Examples of Use -

1. Specifies the time at which the boundary condition change will occur.

```
**BCCHANGE
  TIMEs-bcch  10.0

**BCSET
  ID-Bcset  U1FIX
  VALUe
     .
     .
     .
```

5.8 GPBEST-DCE CONTROL

This input section provides **GPBEST-DCE** with the information to control the Distributed Computing Environment execution. In order to make **GPBEST-DCE** user-friendly, most of the features are completely automatic so the user need only input a minimal amount of information. **GPBEST-DCE** automatically checks if the correct internet host name has been entered and if the respective host is alive and functioning. The current machine loadings of the host pool (available hosts input by user) is automatically calculated and appropriate working hosts are determined. All data and application file handling are done with unique file names to eliminate simultaneous user errors. **GPBEST-DCE** will startup and shutdown the DCE system and handle application errors gracefully. A multiple GMR **GPBEST-DCE** problem is dynamically load balanced among the working hosts, and speedup calculations which show improved performance over serial mode execution are reported for each region. User-defined individual GMR **GPBEST-DCE** log files contain documented **GPBEST** output and warning/error reports. A total summary log of **GPBEST-DCE** execution and results is contained in the user-defined output log file.

The domain of application of **GPBEST-DCE** may be seen in Table 8.2.

A list of keywords recognized in the **GPBEST-DCE** input are given below and a detailed description follows.

NOTE :

A **GPBEST-DCE** LICENSE IS REQUIRED FOR GPBEST TO OPERATE IN A DISTRIBUTED COMPUTING ENVIRONMENT

ALL INPUT TO GPBEST-DCE IS CASE SENSITIVE

SECTION	KEYWORD	PURPOSE	PAGE
5.8.1	Definition of **GPBEST-DCE** Input Card		5.8.3
	**DCE	start of GPBEST-DCE configuration	5.8.3
	HOST	defines required card for host-specific data	5.8.4
	MFLOp	defines required card for host-specific mega flop rating	5.8.5
	SCRAtch	defines required card for host-specific scratch working directory	5.8.6
	COURtesy	defines optional card for user's courtesy factor	5.8.7
	LIMIt	defines optional card for number of working hosts	5.8.9

SECTION	KEYWORD	PURPOSE	PAGE
5.8.1	Definition of **GPBEST-DCE** Input Card		
	MAINdir	defines optional card for main working directory	5.8.11
	MAXTask	defines optional card for maximum number of allowable processes for the DCE configuration	5.8.12
	MEMOry-dce	defines optional card for host-specific dynamic memory allocation value	5.8.14
	MULTiple	defines optional card for host-specific number of processes	5.8.15
	PASSword	defines optional card for the **GPBEST-DCE** password file path and name	5.8.16
	PRIOrity	defines optional card for host-specific priority value	5.8.17
	RESUlt	defines optional card for result directory	5.8.18

5.8.1 GPBEST-DCE Input Card

**DCE

Status - OPTIONAL (Required for best-dce)

Full Keyword - ****DCE**

Function - Identifies the beginning of Distributed Computing Environment input.

Input Variables - NONE

Additional Information -

All relevant **GPBEST-DCE** information must be included under one **DCE input card.

The **DCE card group can be located at the beginning, end, or between other (**) input card groups within the **GPBEST** data file.

Examples of Use -

1. Analysis using two of three available hosts (hal, isaac, and ice) with default result and main directories.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29

  HOST ice.bestview.com
  SCRAtch /ice/mlgreen/ICE
  MFLOp 21

  LIMIt 2
  COURtesy 0
```

HOST HOSTNAME

Status - REQUIRED

Full Keyword - **HOST**

Function - Available hostname for distributed computing environment and identifies the specific hostname to associate the MFLOp, SCRAtch, MEMOry-dce, MULTiple, and PRIOrity input cards with.

Input Variables -

HOSTNAME (Character) - REQUIRED

The full internet hostname of the available host or unique local hostname of the available host

Additional Information -

The full internet hostname can be input, short names and alias names are allowed.

The user must have an account and permission to execute jobs on the specified available host.

A valid **GPBEST** license is required on each host which is to be part of the **GPBEST-DCE** execution.

A host must have an internet or local network connection to be a valid available host.

The number of available hosts can not exceed 80.

Examples of Use -

1. Specifies hosts (hal.gpbest.com and bugs.bestview.com), scratch working directories (/hal/mlgreen/HAL and bugs/mlgreen/BUGS), and mega flop ratings (42, 12).

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12
    .
    .
    .
```

MFLOp MFLOPRATE

Status - REQUIRED

Full Keyword - **MFLOp**

Function - The approximate mega flop rating for a specific available host.

Input Variables -

MFLOPRATE (Integer) - REQUIRED

The mega flop rate for a specific available host

Additional Information -

This card must follow a HOST card that specifies the HOSTNAME to associate the mega flop rate with.

See your computer User Manual for information on MFLOPRATE or see Table 8.3 for approximate MFLOPRATE of popular systems.

An accurate MFLOPRATE value is required to achieve maximum efficiency from **GPBEST-DCE** .

Examples of Use -

1. For hostnames (hal.gpbest.com and bugs.bestview.com) scratch working directories (/hal/mlgreen/trial2 and /bugs/mlgreen/trial2) and mega flop ratings (42 and 12) respectively.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/trial2
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/trial2
  MFLOp 12
```

SCRAtch SCRATCHD

Status - REQUIRED

Full Keyword - **SCRAtch**

Function - The scratch working directory is used for temporary storage of GMR scratch files that contain the decomposed GMR matrices and GMR data files. This input is case sensitive.

Input Variables -

SCRATCHD (Character) - REQUIRED

The full pathname for host-specific scratch working directory

Additional Information -

This card must follow a HOST card that specifies the HOSTNAME to associate the scratch working directory with.

Identifies the scratch working directory for host-specific GMR files.

Each host-specific scratch working directory must have network connection to the main working directory.

The character length of this directory must not exceed 240.

Examples of Use -

1. For hostnames (hal.gpbest.com and bugs.bestview.com) scratch working directories (/hal/mlgreen/trial2 and /bugs/mlgreen/trial2) and mega flop ratings (42 and 12) respectively.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/trial2
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/trial2
  MFLOp 12
```

COURtesy FACTOR

Status - OPTIONAL

Full Keyword - **`COURtesy`**

Function - Sets the courtesy weighting factor for load balancing.

Input Variables -

FACTOR (Integer) - REQUIRED

The courtesy factor for this **GPBEST-DCE** configuration

Additional Information -

$$\text{Load Rating} = \frac{\text{MFLOP Rating}}{(0.1 * \text{FACTOR} * \#\ \text{USERS}) + (1 + \text{CURRENT 5 MIN LOAD})}$$

Load Rating - GPBEST-DCE adjusted rating of machines MFLOp rate.

MFLOp Rating - unloaded machine MFLOp rateinput by the user on HOST card.

FACTOR - user input from the ‘‘COURtesy’’ card.

USERS - number of users logged into machine determined from GPBEST-DCE load sensing routine.

CURRENT 5 MIN LOAD - 5 minute load on machine determined from GPBEST-DCE load sensing routine.

The typical courtesy factor should range from 0 to 10.

A courtesy factor of 0 -- indicates load balancing gives zero weight to the number of users working on a given host in determining the load rating.

A courtesy factor of 10 -- indicates load balancing gives high weight to number of users working on a given host in determining the load rating.

The default value for this is set to 0.

Examples of Use -

1. Specifies a medium courtesy factor for **GPBEST-DCE** execution.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12

  HOST bills.bestview.com
  SCRAtch /bills/mlgreen/BILLS
  MFLOp 19

  MAINdir /run/mlgreen/MAIN

  RESUlt /hal/mlgreen/RESULT.elasbrac

  LIMIt 2

  COURtesy 5
```

LIMIt NHOST

Status - OPTIONAL

Full Keyword - **`LIMIt`**

Function - Identifies the number of working hosts requested.

Input Variables -

NHOST (Integer) - REQUIRED

The number of requested hosts

Additional Information -

Using this card, the user may limit the number of hosts that are actually used for the execution of **GPBEST-DCE** to a subset of the total number of hosts that are available for execution. The names of the available hosts are input on the HOST card. If the number of available hosts input exceeds the number specified on this card, then **GPBEST-DCE** will automatically choose the best ‘‘NHOST’’ machines to carry out the execution according to the load rating factor formula given on the ‘‘COURtesy’’ card.

The default value for this is set to equal the total number of available hosts.

Examples of Use -

1. Specifies limit of two available hosts for **GPBEST-DCE** execution.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 200

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12

  HOST bills.bestview.com
  SCRAtch /bills/mlgreen/BILLS
  MFLOp 19
  MEMOry-dce 100

  HOST gomez.gpbest.com
  SCRAtch /gomez1/mlgreen/GOMEZ
  MFLOp 12

  MAINdir /run/mlgreen/MAIN

  RESUlt /hal/mlgreen/RESULT.elasbrac

  LIMIt 2
```

2. Specifies limit of three available hosts for **GPBEST-DCE** execution.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 200

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12
  MEMOry-dce 16

  HOST bills.bestview.com
  SCRAtch /bills/mlgreen/BILLS
  MFLOp 19

  HOST moriarty.gpbest.com
  SCRAtch /run/mlgreen/MORIARTY
  MFLOp 25
  MEMOry-dce 256

  MAINdir /run/mlgreen/MAIN

  RESUlt /hal/mlgreen/RESULT.elasbrac

  LIMIt 3
```

MAINdir MAIND

Status - OPTIONAL

Full Keyword - **MAINdir**

Function - The main working directory is used for storage of GMR scratch files that contain the decomposed solution matrix. This input is case sensitive.

Input Variables -

MAIND (Character) - REQUIRED

The full pathname for main working directory

Additional Information -

Identifies the main working directory for all **GPBEST-DCE** region files.

This directory will generally require a majority of the **GPBEST-DCE** disk storage.

The main working directory must have network connection to all host node scratch working directories.

The character length of this directory must not exceed 240.

The default value for this is set to the current working directory upon command line execution.

Examples of Use -

1. Specifies main working directory for all region files.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12

  MAINdir /run/mlgreen/elastic/PROBLEM1/trial1
```

MAXTask MAXTASK

Status - OPTIONAL

Full Keyword - **`MAXTask`**

Function - Identifies the maximum number of concurrent GMR decomposition processes allowable in the **GPBEST-DCE** configuration.

Input Variables -

MAXTASK (Integer) - REQUIRED

The maximum number of concurrent GMR decomposition processes

Additional Information -

Using this card, the user may limit the number of GMR decomposition processes that are actually used for the execution of **GPBEST-DCE** to a subset of the total number of allowable processes that are available for execution. The number of allowable processes in a **GPBEST-DCE** configuration can be calculated by summing the input MULTiple card values for all hosts (the default value of MULTiple is set equal to one for all hosts). If the number of available processes input exceeds the number specified on this card, then **GPBEST-DCE** will automatically choose the best ‘‘MAXTask’’ processes to carry out the execution according to the load rating factor.

The default value for this card is set equal to the total number of available processes.

Examples of Use -

1. Specifies limit of three available hosts for **GPBEST-DCE** execution and also limit the maximum number of processes executed by **GPBEST-DCE** to four.

```
**DCE
  HOST byte.gpbest.com
  SCRAtch /byte/mlgreen/BYTE
  MFLOp 57
  MEMOry-dce 300
  MULTiple 3
  PRIOrity 5

  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 200
  MULTiple 2
  PRIOrity 10

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12

  HOST bills.bestview.com
  SCRAtch /bills/mlgreen/BILLS
  MFLOp 19
  MEMOry-dce 100
```

```
HOST gomez.gpbest.com
SCRAtch /gomez1/mlgreen/GOMEZ
MFLOp 12

MAINdir /run/mlgreen/MAIN

RESUlt /hal/mlgreen/RESULT.elasbrac

LIMIt 3

MAXTask 4
```

MEMOry-dce MEMORY

Status - OPTIONAL

Full Keyword - **MEMOry-dce**

Function - The amount of dynamically allocated memory to use for GMR and residual solver decompostion on a specific host.

Input Variables -

MEMORY (Integer) - REQUIRED

The amount of solver decomposition memory available for use

Additional Information -

This card must follow a HOST card that specifies the HOSTNAME to associate the memory value to.

The memory value is an upper limit of the allowable memory to use on a specific host.

If the solver decomposition routines do not require the memory value input -- only the necessary memory required will be allocated for decomposition.

The default value for the MEMOry-dce card is 24 megabytes.

Examples of Use -

1. For hostnames (hal.gpbest.com and bugs.bestview.com) scratch working directories (/hal/mlgreen/trial2 and /bugs/mlgreen/trial2), mega flop ratings (42 and 12), and memory (200 and 50) respectively.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/trial2
  MFLOp 42
  MEMOry-dce 200

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/trial2
  MFLOp 12
  MEMOry-dce 50
```

MULTiple MULTIPLE

Status - OPTIONAL

Full Keyword - **`MULTiple`**

Function - The maximum number of GMR decomposition processes to execute on a specific host.

Input Variables -

MULTIPLE (Integer) - REQUIRED

The number of GMR decomposition processes to execute

Additional Information -

This card must follow a HOST card that specifies the HOSTNAME to associate the multiple process value to.

The multiple process value is an upper limit of the allowable number of processes to use on a specific host.

If **GPBEST-DCE** configuration routine does not require the maximum number of processes -- only the necessary processes required will be allocated for GMR decompositions.

The default value for the MULTiple card is one process.

Examples of Use -

1. For hostnames (byte.gpbest.com and bugs.bestview.com) scratch working directories (/byte/mlgreen/trial2 and /bugs/mlgreen/trial2), mega flop ratings (57 and 12), and memory (200 and 50) respectively. The MULTiple card is used to allow (byte.gpbest.com) to execute two processes if necessary.

```
**DCE
  HOST byte.gpbest.com
  SCRAtch /byte/mlgreen/trial2
  MFLOp 42
  MEMOry-dce 200
  MULTiple 2

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/trial2
  MFLOp 12
  MEMOry-dce 50
```

PASSword PASSWORD

Status - OPTIONAL

Full Keyword - **PASSword**

Function - Identifies the name and full path location of the **GPBEST-DCE** password file.

Input Variables -

PASSWORD (Character) - REQUIRED

The name and full path location of **GPBEST-DCE** password file.

Additional Information -

This card is used for **GPBEST-DCE** slave host executions. This card can be used in conjuction with the GPBEST_PASSWORD_FILE environment variable to ensure proper **GPBEST-DCE** slave host execution.

The default value for this card is obtained from the **best-dce** master executing host environment variable GPBEST_PASSWORD_FILE if set.

Examples of Use -

1. The following example specifies the **GPBEST-DCE** password file with path and name of **/byte/mlgreen/.gpbpassword** for all available hosts.

```
**DCE
  HOST byte
  SCRAtch /byte/mlgreen/BYTE
  MFLOp 57
  MEMOry-dce 300
  MULTiple 3
  PRIOrity 5

  HOST hal
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 200
  MULTiple 2
  PRIOrity 10

  MAINdir /run/mlgreen/MAIN

  RESUlt /hal/mlgreen/RESULT.elasbrac

  MAXTask 3

  PASSword /byte/mlgreen/.gpbpassword
```

PRIOrity PRIORITY

Status - OPTIONAL

Full Keyword - **PRIOrity**

Function - The system level priority value for a specific host used on GMR decomposition processes (equivalent to the shell command nice).

Input Variables -

PRIORITY (Integer) - REQUIRED

The system level priority value

Additional Information -

This card must follow a HOST card that specifies the HOSTNAME to associate the priority value to.

The priority value defines the scheduling order of cpu access on a specific host.

The priority value can range from a high value of 0 and a low value of 19. Where a value of 0 will schedule cpu access higher than 19.

The default value for the PRIOrity card is zero.

Examples of Use -

1. For hostnames (byte.gpbest.com and bugs.bestview.com) scratch working directories (/byte/mlgreen/trial2 and /bugs/mlgreen/trial2), mega flop ratings (57 and 12), and memory (200 and 50) respectively. The MULTiple card is used to allow (byte.gpbest.com) to execute two processes if necessary. Since (byte.gpbest.com) is allowed to execute two GMR decomposition processes the PRIOrity card is used to lower the cpu access.

```
**DCE
  HOST byte.gpbest.com
  SCRAtch /byte/mlgreen/trial2
  MFLOp 42
  MEMOry-dce 200
  MULTiple 2
  PRIOrity 19

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/trial2
  MFLOp 12
  MEMOry-dce 50
```

RESUlt RESULTD

Status - OPTIONAL

Full Keyword - **RESUlt**

Function - Identifies the result directory for **GPBEST-DCE** log and result files. This input is case sensitive.

Input Variables -

RESULTD (Character) - REQUIRED

The full pathname for result directory

Additional Information -

The result directory must have network connection to the main working directory.

The character length of this directory must not exceed 240.

The default value for this is set to the current working directory upon command line execution.

Examples of Use -

1. Specifies result directory for **GPBEST-DCE** log and result files.

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /hal/mlgreen/HAL
  MFLOp 42

  HOST bugs.bestview.com
  SCRAtch /bugs/mlgreen/BUGS
  MFLOp 12

  MAINdir /run/mlgreen/elastic/PROBLEM1/trial2

  RESUlt /hal/mlgreen/PROBLEM1/trial2
```

6.0 TUTORIALS AND EXAMPLES

This chapter is a good starting point for users who are new to **GPBEST.**This chapter forms the bridge between the theory and syntax of **GPBEST,**and the solution of actual problems. Several practical items are discussed. The **GPBEST** learning curve will be greatly reduced if the user becomes acquainted with this section.

Section 6.1 discusses how to determine which options are supported in each analysis type. For example, a user may wonder which kinds of elements are supported for elastic analysis. While **GPBEST** has extensive capabilities, all options are not available for all analysis types. This section will help the user to sort through the relevant **GPBEST** cards for each analysis type. This section also discusses the current limits in **GPBEST,**which the user needs to know before building a model that may be too big to solve.

The path for data transfer in and out of **GPBEST** is described in Section 6.2. This is vital information for the user who uses a graphics pre- and post-processor for modeling and results evaluation. Translators are supplied for BESTVIEW, PATRAN, I-DEAS, IGES, and other systems.

The rest of this chapter lists some example problems. These are simple problems that users can study to gain an understanding of the proper cards to use in **GPBEST.**Experienced users will often consult this section when they are engaged in a new type of analysis or option. The example problems can be modified and rerun in **GPBEST** to study the effect of additional options. In this way, the proper procedure for running **GPBEST** for a given analysis type with certain options can be tested on a small problem before running a large, complex job.

6.1 GPBEST Features and Limits

6.1.1 GPBEST.FEATURES FILE

A new user of **GPBEST** is exposed to a seemingly endless number of cards and options, primarily in Chapter 5 of this manual. Many new users wonder which options are supported for their analysis. For instance, can anisotropic material properties be specified in an elastic analysis? Or can quartic boundary elements be used in a heat transfer analysis? The answer to most of these questions can be answered through a brief discussion of the `GPBEST.FEATURES` file.

As noted in Section 5.1, if the PRIN FEAT (PRINtout-control FEATURES) option is specified in **CASE, then **GPBEST** will create a file called `GPBEST.FEATURES`. This file will contain a table of all the various options and analysis types in **GPBEST.**It is highly recommended that a new user create this file and print it out for ready reference. It constitutes a cross-reference listing of supported and unsupported options and analysis types.

To print out the `GPBEST.FEATURES` file, there is no need to run a full **GPBEST** analysis. The following five cards are sufficient:

```
**CASE
  TITLe      JUST PRINT OUT GPBEST.FEATURES FILE
  PRINtout-control FEATURES
  CHECk-data
$ END OF FILE
```

Once the above file is input to **GPBEST,**the file `GPBEST.FEATURES` is created. Since it is 130 characters wide, it should be printed on wide paper or printed using a reduced font. Once this file is printed for a given version of **GPBEST,**it does not need to be printed again.

To illustrate the important features of `GPBEST.FEATURES`, Table 6.1 includes the first part of the file. `GPBEST.FEATURES` uses a numerical code to signify whether various options are fully implemented in **GPBEST** or not. A value of ‘‘1’’ means that the option is fully implemented and tested. Thus, the user can confidently use any option/analysis type with a value of ‘‘1’’, knowing that it will work properly in **GPBEST.**However, if an option/analysis type has a cross-reference value of ‘‘4’’, the problem may run, but with caution. Other values have similar meaning, as illustrated in Table 6.1.

The second section that is output to the `GPBEST.FEATURES` file is listed in Table 6.2. Each analysis type is assigned an alphabetical letter. For example, periodic acoustic analysis is assigned ‘‘B’’, etc.

The four basic cross-reference tables included in `GPBEST.FEATURES` are 1) two-dimensional analysis, 2) three-dimensional analysis, 3) axisymmetric analysis and 4) generalized axisymmetric analysis.

Table 6.3 shows an excerpt from the `GPBEST.FEATURES` file. In particular, the features (or options) are listed down the left-hand column, while the analysis-type letters are listed across the

top. The implementation status levels appear as numbers in the middle of the table. For instance, SAMPling-points can be used with FORCed-vibration PERIODIC VIBRATION ANALYSIS (L), because the implementation status of these two items is a ‘‘1’’. However, MIXED DIMENSION TYPES cannot be used with ACOUSTIC EIGENFREQUENCY ANALYSIS (A), because the implementation status is a ‘‘6’’. In a similar manner, the user can cross-reference any combination of **GPBEST** options with analysis types to determine if they are supported in the current version of **GPBEST.**

TABLE 6.1
STATUS LEVELS OF SPECIAL FEATURES IN GPBEST.FEATURES

1 - FULLY IMPLEMENTED AND TESTED

2 - IMPLEMENTED, BUT NOT FULLY TESTED (WARNING)

3 - IMPLEMENTED AND TESTED, HOWEVER THIS FEATURE IS ONLY IN THE RESEARCH STAGE AND CAUTION SHOULD BE EXERCISED IN EVALUATING RESULTS (WARNING)

4 - IMPLEMENTED, HOWEVER THIS FEATURE IS NOT SUPPORTED (WARNING)

5 - NOT AVAILABLE IN THE REDUCED VERSION OF GPBEST (FATAL ERROR)

6 - NOT IMPLEMENTED IN GPBEST (FATAL ERROR)

7 - NOT RELEVENT TO THIS ANALYSIS (FATAL ERROR)

TABLE 6.2
ANALYSIS TYPES IN GPBEST.FEATURES

A = ACOUSTIC - EIGENFREQUENCY ANALYSIS
B = ACOUSTIC - PERIODIC ANALYSIS
C = ACOUSTIC - TRANSIENT (LAPLACE DOMAIN) ANALYSIS
D = ACOUSTIC - TRANSIENT (TIME DOMAIN) ANALYSIS
E = CONCURRENT THERMOMECHANICS - STEADY STATE ANALYSIS
F = CONCURRENT THERMOMECHANICS - TRANSIENT ANALYSIS
G = CONCURRENT THERMOMECHANICS - NON-LINEAR ANALYSIS
H = CONSOLIDATION - STEADY STATE ANALYSIS
I = CONSOLIDATION - TRANSIENT ANALYSIS
J = CONSOLIDATION - NON-LINEAR ANALYSIS
K = DYNAMICS - FREE VIBRATION ANALYSIS
L = DYNAMICS - PERIODIC (FORCED VIBRATION) ANALYSIS
M = DYNAMICS - TRANSIENT (LAPLACE DOMAIN) ANALYSIS
N = DYNAMICS - TRANSIENT (TIME DOMAIN) ANALYSIS
O = DYNAMICS - NON-LINEAR TRANSIENT (TIME DOMAIN) ANALYSIS
P = ELASTOSTATICS ANALYSIS
Q = INCREMENTAL ELASTO-PLASTICITY ANALYSIS
R = HEAT CONDUCTION - STEADY STATE ANALYSIS
S = HEAT CONDUCTION - TRANSIENT ANALYSIS
T = POTENTIAL - EIGENFREQUENCY ANALYSIS
U = POTENTIAL - STEADY STATE ANALYSIS
V = POTENTIAL - DIFFUSION ANALYSIS
W = POTENTIAL - PERIODIC ANALYSIS
X = POTENTIAL - TRANSIENT ANALYSIS
Y = VISCOUS FLUID - STEADY STATE, LINEAR ANALYSIS
Z = VISCOUS FLUID - STEADY STATE, NON-LINEAR ANALYSIS
A = VISCOUS FLUID - TRANSIENT, LINEAR ANALYSIS
B = VISCOUS FLUID - TRANSIENT, NON-LINEAR ANALYSIS
C = THERMOVISCOUS FLUID - STEADY STATE, LINEAR ANALYSIS
D = THERMOVISCOUS FLUID - STEADY STATE, NON-LINEAR ANALYSIS
E = THERMOVISCOUS FLUID - TRANSIENT, LINEAR ANALYSIS
F = THERMOVISCOUS FLUID - TRANSIENT, NON-LINEAR ANALYSIS

TABLE 6.3
LISTING OF A PORTION OF GPBEST.FEATURES

```
TABLE 1:  TWO-DIMENSIONAL ANALYSIS
-----------------------------------------------------------------------------------------------

NO.           SPECIAL FEATURES                                                TYPE OF ANALYSIS
                                                          A B C D E F G H I J K L M N O P Q R S T U V . . .
-----------------------------------------------------------------------------------------------
 1 "ANALYSIS WITH NO OPTIONS"                             1 1 2 5 1 1 2 2 1 2 1 1 2 5 5 1 2 1 1 6 6 6

 2 "SAMPLING POINTS"                                      2 1 2 5 1 1 2 2 1 2 2 1 2 5 5 1 2 1 1 6 6 6

 3 "REAL*4 STORAGE"                                       1 1 2 6 1 1 2 2 1 2 1 1 2 6 6 1 2 1 1 6 6 6

 4 "MIXED DIMENSION TYPES IN DIFFERENT REGIONS"           6 6 6 6 1 2 6 2 2 6 2 6 6 6 6 1 6 1 2 6 6 6

 5 "GEOMETRIC AND LOADING SYMMETRY"                       2 1 2 5 2 1 2 2 1 2 2 1 2 5 5 1 2 1 2 6 6 6

 6 "RESTART READ OR RESTART WRITE"                        2 2 2 6 1 2 5 2 2 5 2 2 2 6 6 1 5 1 1 6 6 6
 .
 .
 .
```

6.1.2 GPBEST LIMITS

Another important listing to have is the list of current limits for **GPBEST.**This will help the user determine how many nodes to include in each GMR, how many total boundary elements can be run in a single analysis, etc. The current limits are included in the **GPBEST** results file when PRINtout-control LIMITS is included in **CASE.

Table 6.4 lists the current limits for the base version of **GPBEST**(i.e., **best-s**), which is limited to 24,000 d.o.f. This table also lists the limits for the optional 60,000 d.o.f. version, **best-s80.**

Some users have enhanced versions of **GPBEST,**which permit even larger problems to be run. Users with special versions of **GPBEST** should print out their own limits, because they will likely be different from those shown in Table 6.4.

To print out the limits for the current version of **GPBEST,**run the following short input file:

```
**CASE
  TITLe      PRINT OUT CURRENT GPBEST LIMITS TO RESULTS FILE
  PRINtout-control LIMITS
  CHECk-data
$ END OF FILE
```

TABLE 6.4
LISTING OF GPBEST LIMITS (best-s and best-s80)

ENTITY[1]	best-s	best-s80
GLOBAL PARAMETERS		
total geometric modeling regions	20	80
total degrees of freedom[2]	24000	60000
total residual degrees of freedom (dof on interfaces)	10000	20000
total geometric points (includes non-source points)	16000	54000
total boundary source points	8000	20000
total boundary elements	4000	18000
total line elements	400	400
total enclosing elements	100	100
total discontinuity elements	100	100
REGION (GMR) PARAMETERS		
surfaces in any region	15	15
degrees of freedom in any region[2]	8000	12000
point sources/forces in any region	25	25
OTHER PARAMETERS		
time points for elastic analysis[3]	20	20
table points for boundary condition and body force definition	20	20
temperature points for material properties	21	21
USER SPECIFIED NUMBERING		
points[4]	99999	99999
elements[4]	99999	99999

[1] Definition of the terminology used in this table can be found in Section 4.5.
[2] For analyses involving complex arithmetic, the total degrees of freedom is half.
[3] For periodic analyses, instead of time points, frequencies have been used.
[4] For generalised axisymmetry, the limit is 9999

6.2 DATA FLOW IN AND OUT OF GPBEST

This section describes the flow of data from various file formats and programs to other file formats and programs. Since there are many programs involved (GPBEST, BESTVIEW, PREBEST, POSTBEST, PATRAN, I-DEAS, PRO/ENGINEER), and many kinds of files (data files, neutral files, iges files, etc.), it is useful to determine how to get from one file type to another.

As mentioned in Section 4.2, usually the best initial approach for a new user is to run a few pre-existing GPBEST example problems. This will acquaint the new user with the GPBEST input file format, which is essential. It would also be helpful to modify an existing problem and rerun it, observing changes in the results. This way the new user becomes familiar with the GPBEST input file and Section 5 of the GPBEST User's Manual. Only after this has been accomplished should the user undertake a translation of modeling geometry from a CAD system to GPBEST.

As models become more complex, a graphical CAD package becomes a necessity. Graphical CAD packages allow the user to work with: 1) models consisting of thousands of nodes and elements, 2) complex part geometries, and 3) sophisticated modeling practices, such as dividing the model into regions, known as GMRs.

For these reasons, as well as the wide availability of large scale CAD and finite element modeling programs, the GPBEST user will probably need to transfer data from another program into GPBEST. This section will supply a starting point for this activity.

6.2.1 BESTVIEW

BESTVIEW is the graphical program that has been written specifically to integrate with GPBEST. BESTVIEW is the perfect companion product for GPBEST, since both have been developed together and are closely integrated. BESTVIEW has been written to work on X-Window UNIX workstations, which is the de facto standard computing environment for conducting engineering analysis today.

Even if a GPBEST user uses another CAD program for modeling, we highly recommend obtaining BESTVIEW due to its tight integration with GPBEST. The cost of BESTVIEW is nominal, and users usually find that the time they save in displaying and debugging complex boundary element models in BESTVIEW more than pays for the program.

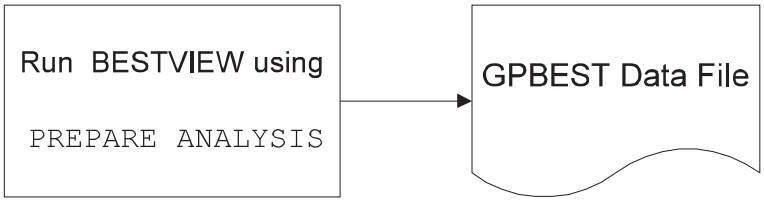
The primary functions of the BESTVIEW program are:

1) To display boundary element models and results of GPBEST analyses
2) To interface with other CAD programs
3) To apply boundary conditions and loads to boundary element models, whether they have been created in BESTVIEW or not

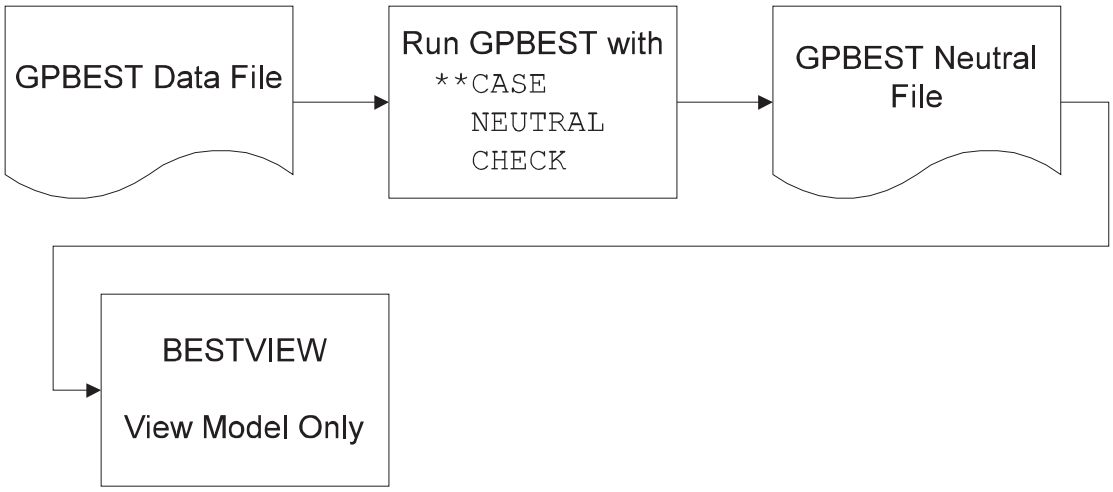
Referring to item 3 on the above list, it should be mentioned that most commercial finite element modeling packages do not integrate well with boundary element models when applying boundary conditions and loads. The finite element method concentrates on point (node) type boundary conditions and loads, whereas boundary element models emphasize distributed (element) loads and boundary conditions, typically involving tractions. Although both methods can handle both kinds of problems, the finite element modeling programs usually do not offer the boundary element analyst a full complement of necessary tools. This is an important reason for a user to use BESTVIEW for applying loads and boundary conditions, due to the fact that all options for GPBEST are supported.

An important aspect of BESTVIEW is that it is designed to work along with other CAD programs, while not necessarily replacing them. Thus, a typical GPBEST user could use PATRAN, for example, to model the part geometry and build the boundary element mesh, then transfer the boundary element model into BESTVIEW where boundary conditions and loads can be added. From there, the GPBEST data set can be output and submitted for execution. Results of the GPBEST run can either be viewed either in BESTVIEW or PATRAN.

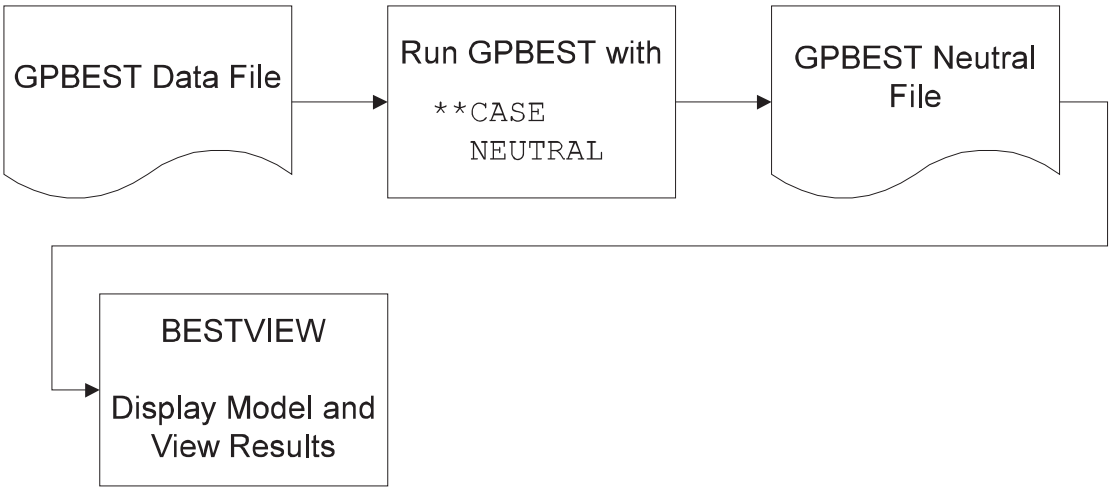
If BESTVIEW is used to create a boundary element model, then it can write out a GPBEST data file using the "Prepare Analysis" option:



If you have a GPBEST data file, and would like to view the model in BESTVIEW, you must run a quick GPBEST "CHECk-data" run. This creates a GPBEST neutral file, which BESTVIEW can read using the POST-RESULTS/READ-GPBEST-RESULTS menu:



If you have a GPBEST data file, and would like to view the model and results of a GPBEST analysis in BESTVIEW:



For more information on BESTVIEW, please refer to the separate BESTVIEW User's Manual. New users of BESTVIEW can effectively start with Section 1.6 "Quick Start Section," and Chapter 8 "A Tutorial on BESTVIEW."

6.2.2 PATRAN

PATRAN has been the most popular finite element modeling program for several years. Previously PATRAN was marketed by PDA. Currently, PATRAN is being marketed by MSC, the owners of MSC/NASTRAN, a very popular finite element analysis program.

A significant advantage of PATRAN's popularity is that it has become somewhat of a standard among finite element modeling and analysis programs. Most popular programs support the PATRAN file structures, both for its neutral file and result files. Even if you do not have PATRAN, your CAD or FEA graphics program is probably able to save data in PATRAN format.

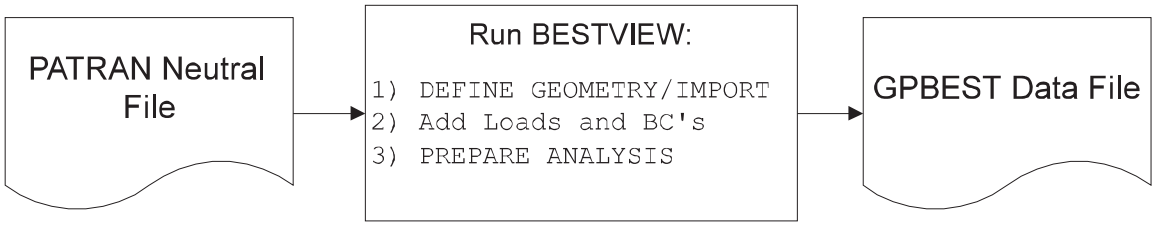
Support for GPBEST is provided through two PATRAN translators:

1) PREBEST - to go from PATRAN to GPBEST
2) POSTBEST - to go from GPBEST to PATRAN

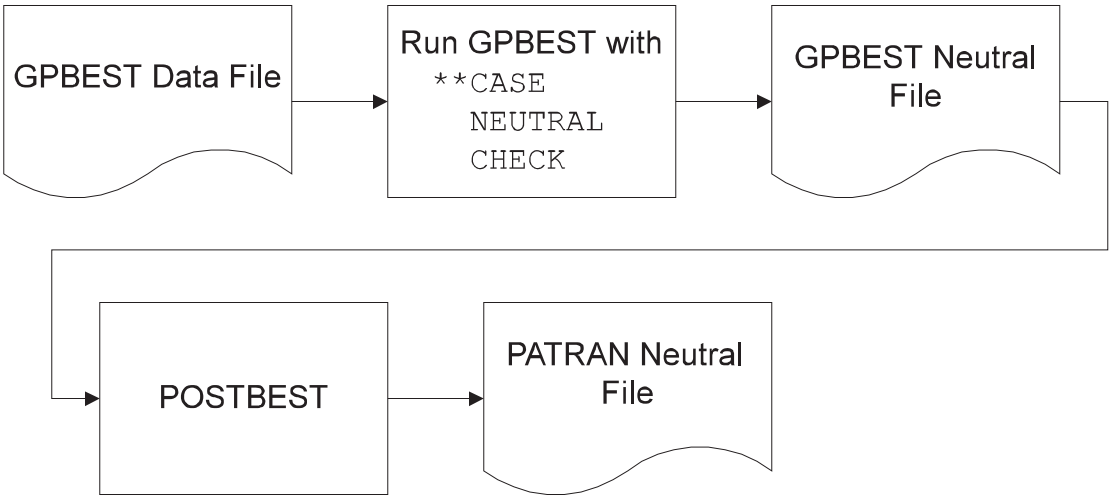
PATRAN can be used to create part geometry, generate a boundary element surface mesh, apply boundary conditions, and view results of a GPBEST analysis.

If a user creates the part geometry, boundary element mesh, and applies loads and boundary conditions in PATRAN (i.e., a complete model), then the GPBEST input file is translated directly by PREBEST:

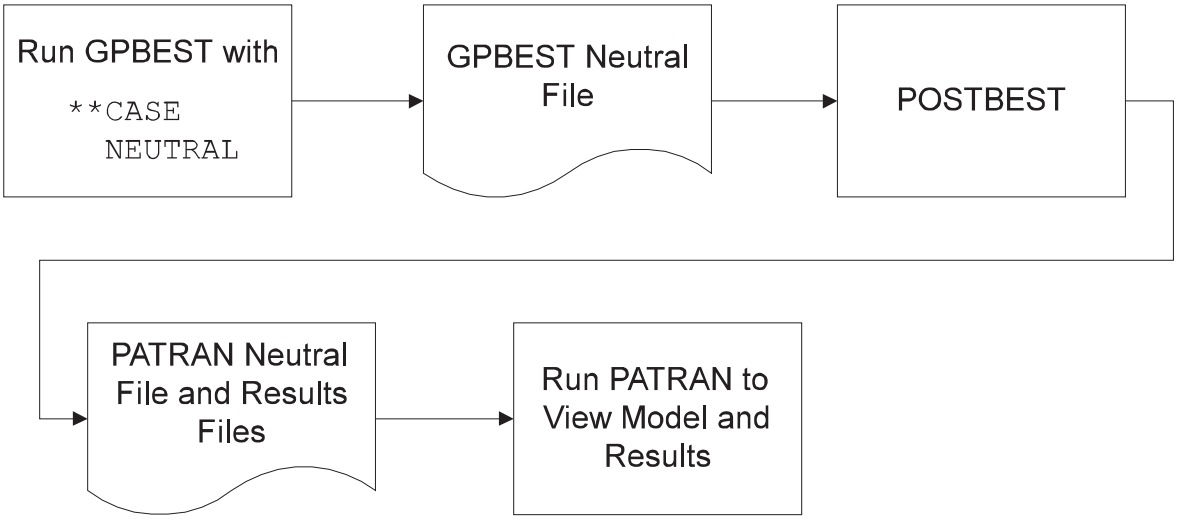
If the user uses PATRAN only to generate the boundary element model, but omits loads and boundary conditions, then BESTVIEW can be used to add these and write out the GPBEST input file:



If the user has a GPBEST data file, and wants to view the model in PATRAN, POSTBEST can be used to generate a PATRAN neutral file:



PATRAN users can also display results of a GPBEST run. GPBEST can output both model and results to the GPBEST neutral file, which POSTBEST can translate into a PATRAN neutral file and PATRAN results files:



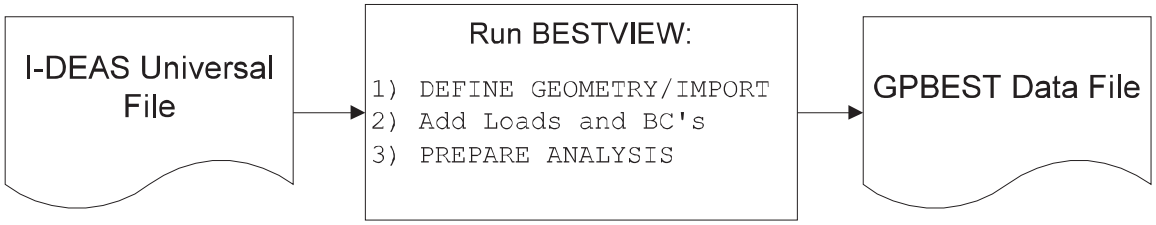
If you desire more information about the PATRAN translators PREBEST and POSTBEST, the GPBEST entities supported from PATRAN and specifying GPBEST loads and boundary conditions inside PATRAN, please see Chapter 9 of the BESTVIEW User's Manual.

6.2.3 I-DEAS

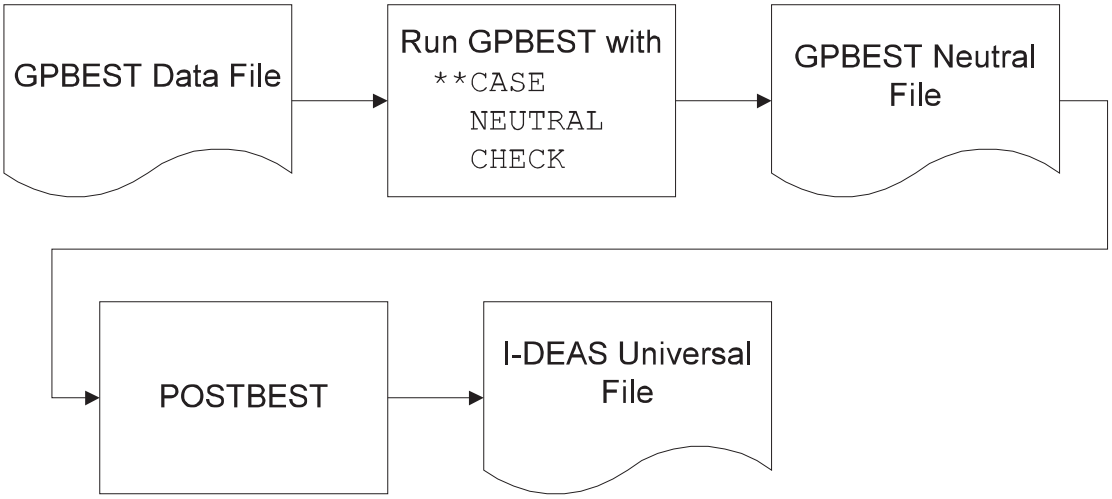
Much of what was stated in Section 6.2.2 regarding PATRAN applies to I-DEAS. Both PATRAN and I-DEAS are fully supported in PREBEST and POSTBEST.

If a user creates the part geometry, boundary element mesh, and applies loads and boundary conditions in I-DEAS, then the GPBEST input file is translated directly by PREBEST:

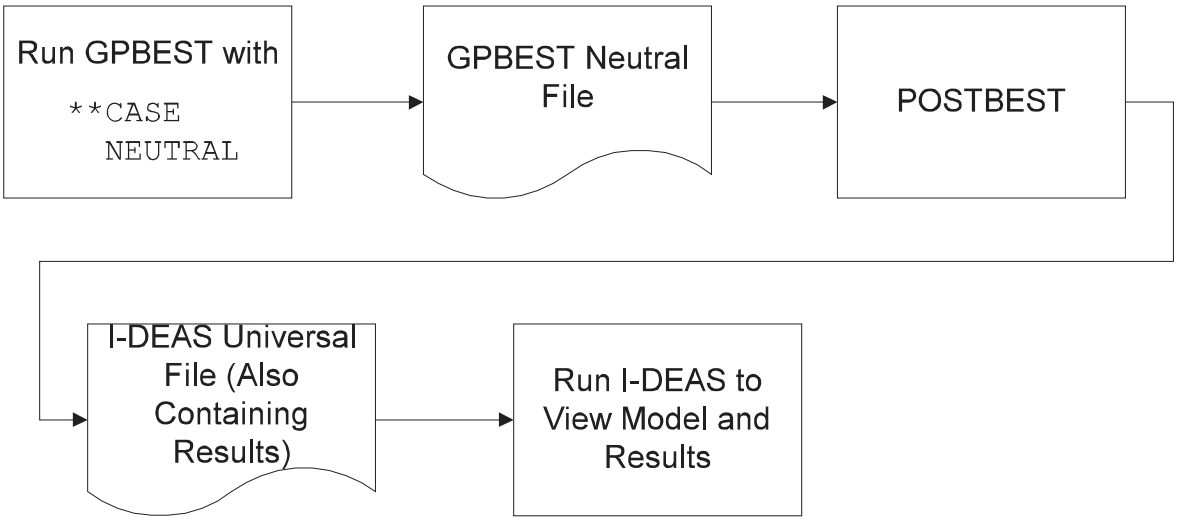
If the analyst uses I-DEAS only to generate the boundary element model, but leaves out loads and boundary conditions, then BESTVIEW can be used to add loads and boundary conditions and write out the GPBEST input file:



If the user has a GPBEST data file, and wants to view the model in I-DEAS, POSTBEST can be used to generate an I-DEAS universal file:



Finally, if the user already has a GPBEST data file, and wants to view the model and results in I-DEAS, POSTBEST can be used to create an I-DEAS universal file containing both model and results:



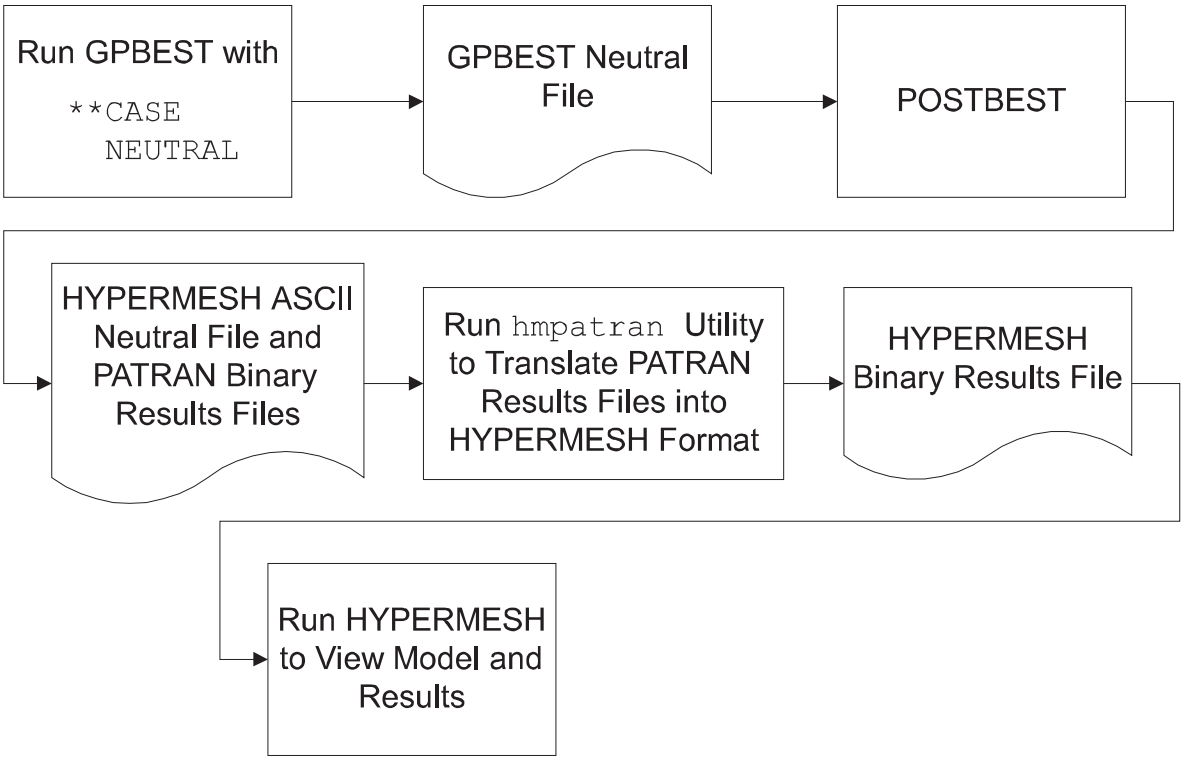
If you desire more information about the I-DEAS translators PREBEST and POSTBEST, the GPBEST entities supported from I-DEAS and specifying GPBEST loads and boundary conditions inside I-DEAS, please see Chapter 9 of the BESTVIEW User's Manual.

6.2.4 HYPERMESH

HYPERMESH is marketed by Altair Computing, Inc. It is primarily inteded for the finite element modeling community, and is quite popular with automotive engineers. It interfaces well with a number of analysis codes.

GPBEST support for HYPERMESH is provided in the PREBEST and POSTBEST translators. The user can create a boundary element model in HYPERMESH, defining nodes, elements, materials, boundary conditions, and GMRs. The model is then exported to a HYPERMESH ASCII neutral file. This file is then used by the PREBEST program, as shown in the following figure:

Results of a GPBEST analysis can likewise be displayed in HYPERMESH. Note that POSTBEST writes out GPBEST results in PATRAN binary result file format. The hmpatran program, which is provided with HYPERMESH, is used to create the binary HYPERMESH results file. The following figure illustrates the procedure:



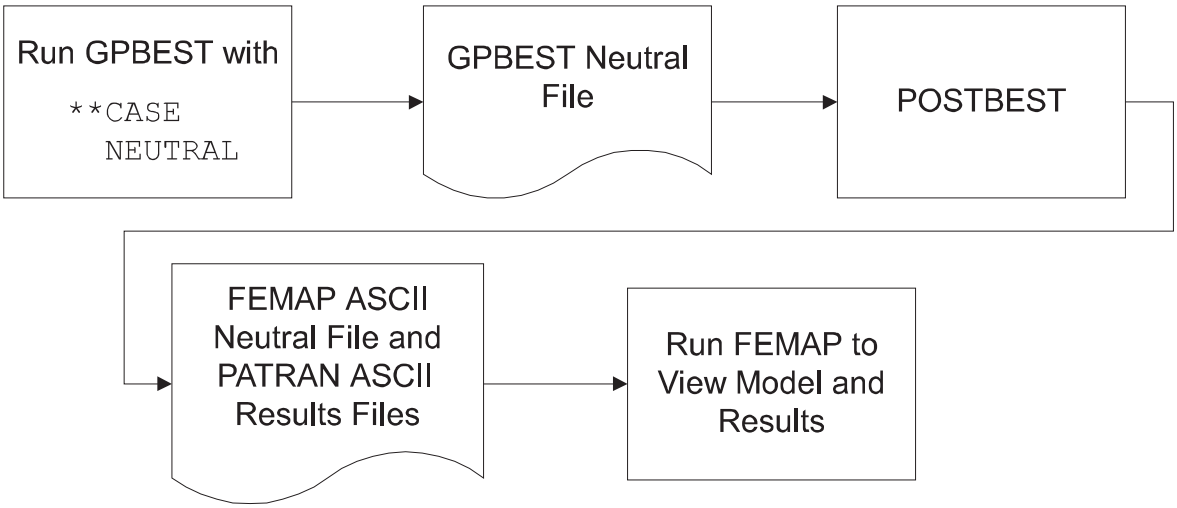
If you desire more information about the HYPERMESH translators PREBEST and POSTBEST, the GPBEST entities supported from HYPERMESH and specifying GPBEST loads and boundary conditions inside HYPERMESH, please see Chapter 9 of the BESTVIEW User's Manual.

6.2.5 FEMAP

FEMAP is marketed by Enterprice Software Products, Inc. It is primarily inteded for the finite element modeling community, and is quite popular with engineers using the PC. It interfaces well with a number of analysis codes.

GPBEST support for FEMAP is provided in the PREBEST and POSTBEST translators. The user can create a boundary element model in FEMAP, defining nodes, elements, materials, boundary conditions, and GMRs. The model is then exported to a FEMAP ASCII neutral file. This file is then used by the PREBEST program, as shown in the following figure:

Results of a GPBEST analysis can likewise be displayed in FEMAP. Note that POSTBEST writes out GPBEST results in PATRAN ASCII result file format. The following figure illustrates the procedure:



If you desire more information about the FEMAP translators PREBEST and POSTBEST, the

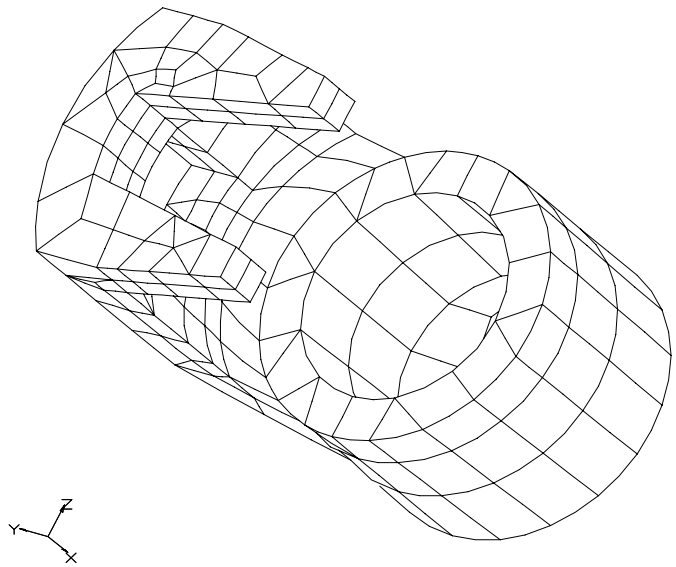
GPBEST entities supported from FEMAP and specifying GPBEST loads and boundary conditions inside FEMAP, please see Chapter 9 of the BESTVIEW User's Manual.

6.2.6 PRO/Engineer

PRO/Engineer has become extremely popular due to the broad range of programs that interface with it because of its extensive capabilities. PRO/Engineer emphasizes describing parts in terms of solid models (via solid modeling). When the geometry is defined in terms of solid entities, the geometry can be sent to CAD/CAM controlled machinery, prototype makers, and finite element analysts. Dealing with solid models is the definite trend for the future in CAD.

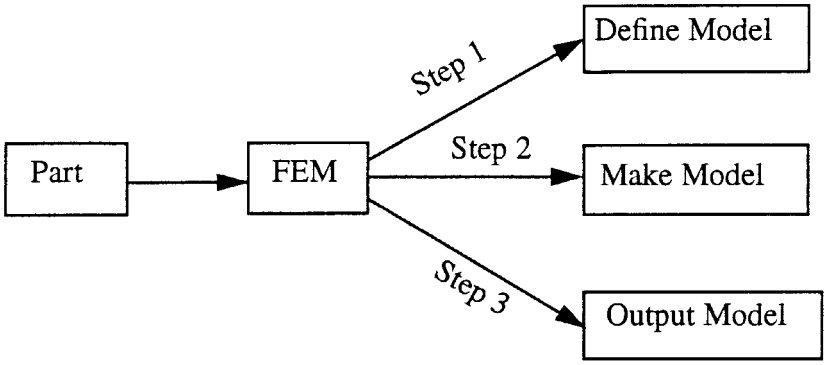
PRO/Engineer reduces the time it takes to build a 3-D boundary element model. Once the component is described in terms of solid modeling primitives, the surface of the solid model can be meshed automatically. The resulting surface mesh is a boundary element mesh that can be used by GPBEST. The traditional approach would involve the additional time-consuming steps of manually extracting the surfaces and meshing them individually.

The following figure illustrates a model that was solid-modeled in PRO/Engineer and then automatically surface meshed. We emphasize that this mesh was not manually defined, it was automatically generated.



In the future, we hope to incorporate a PRO/Engineer translator in the PREBEST and POSTBEST programs. However, for the time being, the best way to get data from PRO/Engineer to GPBEST is through a PATRAN neutral file. Note that you do not have to own the PATRAN program to effect this transfer. The PATRAN neutral file is merely a vehicle for transferring data. PRO/Engineer can write out a PATRAN neutral file.

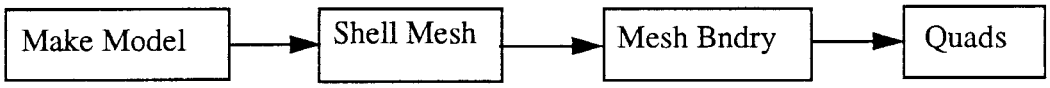
The overall process of generating a model in PRO/Engineer for GPBEST is displayed in the following figure:



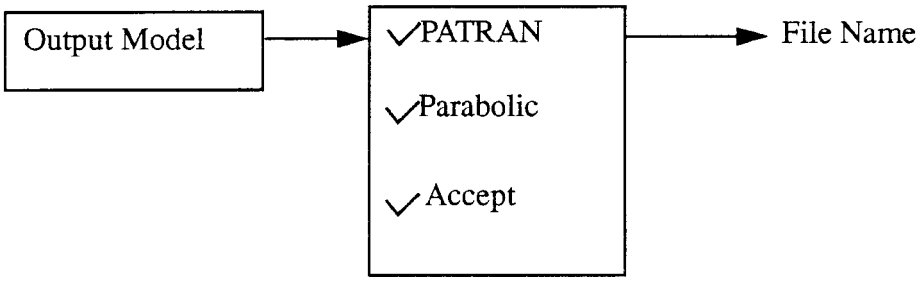
These three steps are:

Step 1: Define Model: Define material properties, mesh control, etc.

Step 2: Make Model: Create the boundary element mesh.



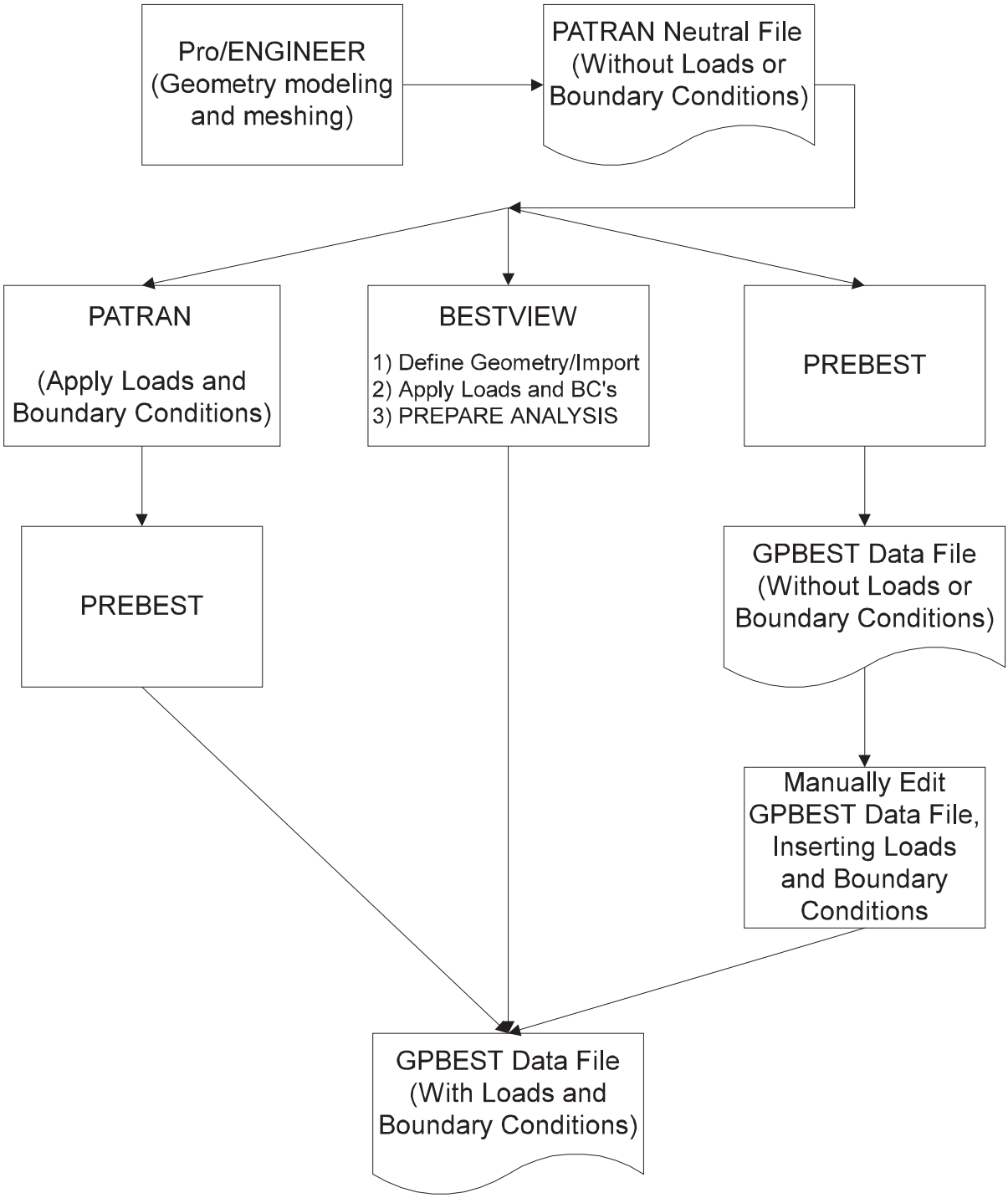
Step 3: Output Model: Output model to a PATRAN neutral file. Use the following options:



Once the boundary element mesh is saved in a PATRAN neutral file, the user needs to apply boundary conditions and loads. This can be done:

1) in PATRAN
2) in BESTVIEW
3) manually edited in the GPBEST data file.

Although the last option is least desirable, this may be feasible for a one-time analysis. One advantage of manually editing boundary conditions and loads in GPBEST is that most of this is done on the element level. So the user often has to make note of only a few elements to specify the loads and boundary conditions. These three paths are illustrated in the following figure:


Pro/ENGINEER
(Geometry modeling
and meshing)
PATRAN Neutral File
(Without Loads or
Boundary Conditions)
PATRAN
(Apply Loads and
Boundary Conditions)
BESTVIEW
1) Define Geometry/Import
2) Apply Loads and BC's
3) PREPARE ANALYSIS
PREBEST
PREBEST
GPBEST Data File
(Without Loads or
Boundary Conditions)
Manually Edit
GPBEST Data File,
Inserting Loads
and Boundary
Conditions
GPBEST Data File
(With Loads and
Boundary Conditions)

There is no direct way at this time to display GPBEST results in PRO/Engineer. However, results can be conveniently displayed in BESTVIEW or in PATRAN. We hope to incorporate PRO/Engineer translators into PREBEST and POSTBEST in the near future.

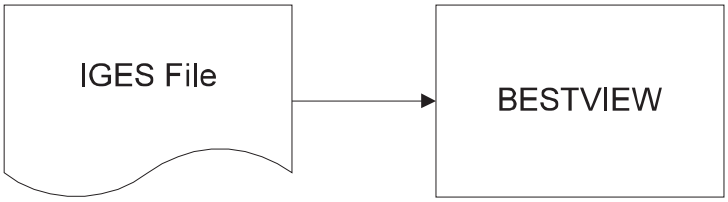
6.2.7 IGES

Most CAD and finite element modeling programs support some form of IGES (Initial Graphics Exchange Specification). There is no computer program named "IGES". IGES is merely a specification of a standard or neutral file format that can be used by numerous programs for the purpose of CAD entity data exchange.

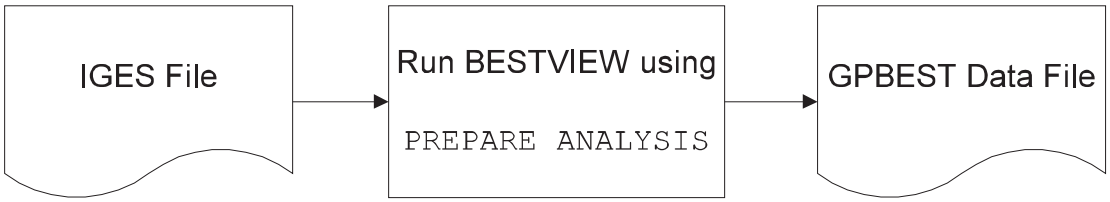
Although IGES is a "standard," most programs only support a subset of IGES entities. So users must be very careful only to use entities that are:

1) Supported by IGES
2) Supported by the translators to and from IGES

When preparing data to be transferred to GPBEST, it is preferable to save the data to a PATRAN neutral file or an I-DEAS universal file if the program supports these. However, if the user must use IGES, then BESTVIEW can read the IGES file directly:



If the user wants to translate an IGES model to create a GPBEST data file, BESTVIEW can write out the GPBEST file. BESTVIEW can also be used to define the loads, boundary conditions and any other missing information that was not included with the IGES file:



Should the user desire more information on IGES data transfer into GPBEST, please consult Section 3.9 of the BESTVIEW User's Manual.

6.2.8 Other Systems

If you are using a privately written CAD or finite element modeling package, then your program may not have direct support for any of the previously mentioned data transfer paths to GPBEST.

It is best to use the following steps to transfer data to GPBEST (in order of most favorable to least favorable options):

1) PATRAN neutral file
2) I-DEAS universal file
3) IGES
4) Contact the Boundary Element Software Technology Corporation. We are trying to support more translators and programs as time goes on.
5) Write your own translator

Although the last option may seem like a big job to undertake, we can provide helpful guidance to assist you in writing your own translator. Also, the GPBEST neutral file format is well documented and fairly straight-forward. In addition, fortran source code for translators is provided with the GPBEST distribution to use as a template for other translators. Hence, the job of writing a data translator from your modeling program to GPBEST may be undertaken without too much difficulty.

For more information about the GPBEST neutral file, please refer to Chapter 10 of the BESTVIEW User's Manual. Also, do not hesitate to call the Boundary Element Software Technology Corporation to discuss your individual needs with us.

6.3 EXAMPLE PROBLEMS

The rest of this chapter illustrates data preparation for **GPBEST** through some example problems. 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 **GPBEST** analysis type:

ACOUSTICS
ACOUSTIC EIGENFREQUENCY ANALYSIS
CONSOLIDATION
CONCURRENT-THERMOELASTICITY
ELASTICITY
FORCED VIBRATION (PERIODIC)
FRACTURE MECHANICS
FREE VIBRATION ANALYSIS
HEAT TRANSFER
STRESS ANALYSIS WITH BODY FORCES
ELASTO-PLASTICITY
CONCURRENT-THERMOPLASTICITY
STRUCTURAL ACOUSTIC ANALYSIS

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 **GPBEST**
4) Selected output from **GPBEST**

It should be noted that since the boundary element models illustrated utilize coarse meshes, the **GPBEST** results may differ somewhat from the theoretical values. However, with a finer mesh, the theoretical values should be obtained. Also, the results may vary slightly depending on the computer system being used to run **GPBEST.**

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 the ELAS605 problem. However, these times will vary a little depending on the computer system being used to run **GPBEST.**

The example problems are located in the `examples` directory under the main **GPBEST** directory. For example, in Section 1.2 (Installation), the main **GPBEST** directory was called `/usr/local/gpbest`. In this case, the examples would be found in the directory `/usr/local/gpbest/examples`. Consult Sections 1.2 and 1.4 for a more complete description of the **GPBEST** directory structure.

A brief listing of the example problem categories is included here. Each example problem starts with four or five alphabetical characters describing the analysis type. For example, the file acou011.dat is a problem involving acoustic analysis.

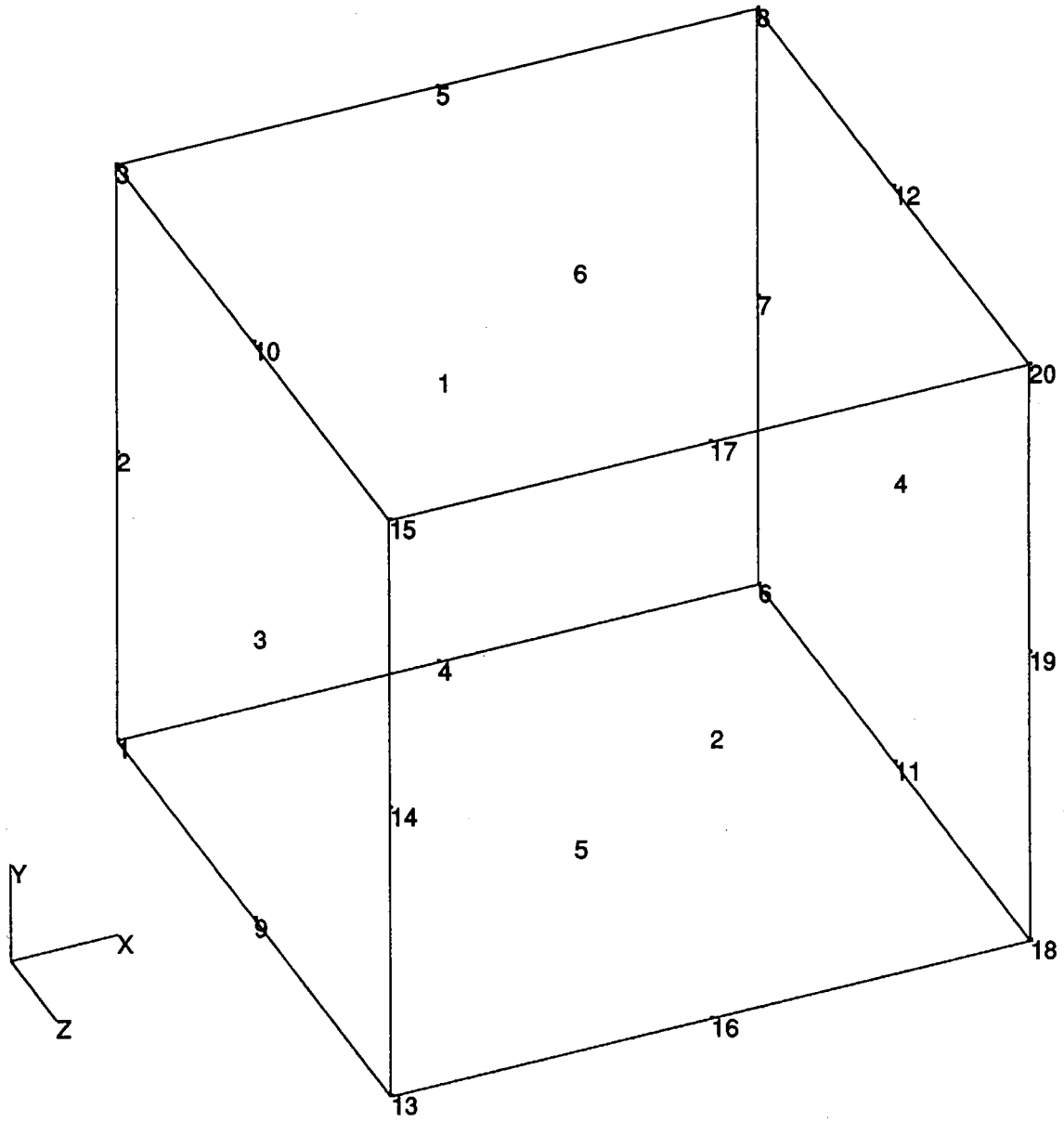
Here is a listing of the example problem analysis types:

Basic Directory	Subdirectories & Files	Explanation
1. examples/		Example problem directory. The example problem files are stored in UNIX `compress` format. To uncompress an entire directory, `cd` to the directory and: `uncompress *.Z`
	EX.summary	File contains a tabular description of the features covered in the example problems.
		The examples directory contains subdirectories of examples of GPBEST data (*.dat) and result (*.res) files. Several problems also include a GPBEST neutral file (*.neu).
	ex.small/	Contains small examples that require very little computing time. Prefix q is used to designate the use of quartic elements and prefix s has been used to designate problems involving body forces or sources.
		The names of the files in subdirectory `ex.small` signify the type of problem as indicated below:
	acou	Acoustic analysis.
	aeig	Acoustic eigenfrequency analysis.
	anis	Elastic anisotropy (includes thermal and mechanical body forces).
	cons	Consolidation.

cpls	Concurrent thermoplasticity (one-way coupling of heat conduction and thermoplasticity).
cthe	Concurrent thermoelasticity (one-way coupling of heat conduction and thermoelasticity).
dist	Thermal distortion with holes (heat from cooling holes causes distortion).
elas	Elasticity (includes thermal and mechanical body forces).
forc	Periodic elastodynamic analyses.
frac	Fracture mechanics.
frvb	Free vibration analysis.
gaxis	Generalized axisymmetric elastic analysis under nonaxisymmetric loading.
heat	Steady-state and transient heat conduction (includes problems with holes).
plas	Elastoplasticity.
SACOustic	Structural acoustic analysis.
ex.large/	Contains a series of data for very large problems of elastic stress analysis involving complex geometries.

6.3.1 ACOUSTIC EXAMPLES

```
EXAMPLE PROBLEM: ACOU601


ANALYSIS TYPE: STEADY-STATE ACOUSTICS
  3-D ANALYSIS

PROBLEM DESCRIPTION:
  VELOCITY SPECIFIED ON ONE FACE OF A CUBICAL ENCLOSURE WITH INPUT
  FREQUENCY OF 1.0.  ZERO PRESSURE SPECIFIED ON THE
  OPPOSITE FACE.  INTERIOR SAMPLING POINTS INCLUDED FOR OUTPUT
  OF PRESSURE.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR, SIX SURFACE ELEMENTS.

SOLUTION POINTS TO VERIFY:
  GMR     NODE     FREQUENCY      PRESSURE
                                  (GPBEST)
  GMR1     23        1.0            0.881


MISCELLANEOUS:
  THE FREQUENCIES SPECIFIED ARE WELL BELOW THE FIRST RESONANT
  FREQUENCY OF 1.5708.
```


8
5
3
12
6
7
10
1
20
2
17
4
15
6
3
19
4
2
1
11
14
5
Y
X
9
18
Z
16
13

```
**CASE
  TITLe   ACOUSTIC RESPONSE IN A CUBICAL ENCLOSURE - VELOCITY SPECIFIED
  ACOUstic  PERIODIC
  FREQuency RADPS 1.0
  NEUTral-file
  ECHO-input


**MATERIAL
  ID-Material MAT1
    SPEEd    1.0
    DENSity  1.0


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  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-surface QUAD
    ELEMents-surface
  1    1  2  3  5  8  7  6  4
  2   13 16 18 19 20 17 15 14
  3    1  9 13 14 15 10  3  2
  4    6  7  8 12 20 19 18 11
  5    1  4  6 11 18 16 13  9
  6    3 10 15 17 20 12  8  5
  NORMal  1 +
  SAMPling-points 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-Bcset PRES15
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 5
  PRESsure-bc
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset VELO16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
  VELOcity-bc
    SPLIst ALL
    T-VAlue 1     1.0

$   ZERO VELOCITY BOUNDARY CONDITION ASSUMED ON ELEMENTS  1 THRU 4

$ END OF DATA
```

```
-----------------------------------------------------------------
          ****          GPBEST RESULTS           ****
-----------------------------------------------------------------


        JOB TITLE:    ACOUSTIC RESPONSE IN A CUBICAL ENCLOSURE - VELOCITY SPECIFIED
        INTERIOR PRESSURE AT EXCITING FREQUENCY =      1.000000         FOR REGION = GMR1

NODE                           PRESSURE
              REAL               IMAG            MAGNITUDE

  21      0.000000E+00      0.000000E+00      0.000000E+00
  22      0.263132E-03      0.455922E+00      0.455922E+00
  23      0.520494E-03      0.881232E+00      0.881232E+00
  24      0.723781E-03      0.125155E+01      0.125155E+01
  25      0.744493E-03      0.154523E+01      0.154523E+01


  END OF GPBEST ANALYSIS
```

EXAMPLE PROBLEM: ACOU602

ANALYSIS TYPE: STEADY-STATE ACOUSTICS
3-D ANALYSIS

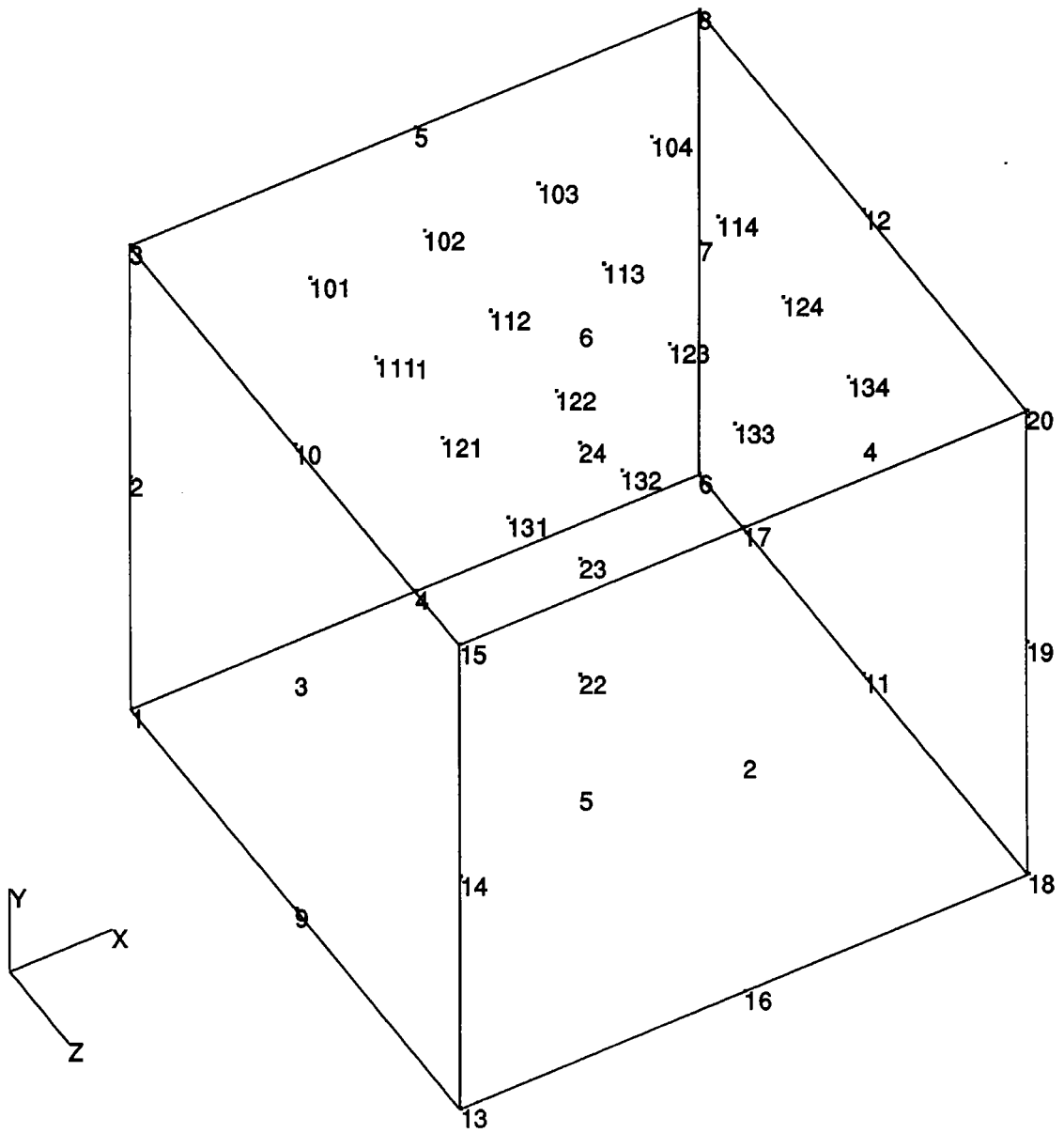
PROBLEM DESCRIPTION:
VELOCITY SPECIFIED ON ONE FACE OF A CUBICAL ENCLOSURE WITH INPUT FREQUENCIES OF 0.01 AND 1.0. ZERO PRESSURE SPECIFIED ON THE OPPOSITE FACE. INTERIOR SAMPLING POINTS INCLUDED FOR OUTPUT OF PRESSURE. POINT SOURCE SPECIFIED

BOUNDARY ELEMENT MODEL:
SINGLE GMR, SIX SURFACE ELEMENTS.

SOLUTION POINTS TO VERIFY:

GMR	ELEMENT	NODE	FREQUENCY	PRESSURE ESTIMATED	PRESSURE (GPBEST)
GMR1	1	5	0.01	2.00E-01	1.891E-01
			1.0	3.10	2.976

MISCELLANEOUS:


8
5
104
103
114
12
3
102
7
101
113
124
112
6
123
1111
134
122
20
133
10
121
24
4
2
132
6
131
17
23
4
15
19
3
22
11
1
2
5
14
18
Y
X
9
16
Z
13

```
**CASE
  TITLe   ACOUSTIC RESPONSE IN A CUBICAL ENCLOSURE - VELOCITY SPECIFIED
  ACOUstic  PERIODIC
  FREQUENCIES RADPS  0.01  1.0
  NEUTral-file
  ECHO-input


**MATERIAL
  ID-Material MAT1
    SPEEd    1.0


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  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
 101       0.2      1.0      0.2
 102       0.4      1.0      0.2
 103       0.6      1.0      0.2
 104       0.8      1.0      0.2
 111       0.2      1.0      0.4
 112       0.4      1.0      0.4
 113       0.6      1.0      0.4
 114       0.8      1.0      0.4
 121       0.2      1.0      0.6
 122       0.4      1.0      0.6
 123       0.6      1.0      0.6
 124       0.8      1.0      0.6
 131       0.2      1.0      0.8
 132       0.4      1.0      0.8
 133       0.6      1.0      0.8
 134       0.8      1.0      0.8
```

```
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
  1    1  2  3  5  8  7  6  4
  2   13 16 18 19 20 17 15 14
  3    1  9 13 14 15 10  3  2
  4    6  7  8 12 20 19 18 11
  5    1  4  6 11 18 16 13  9
  6    3 10 15 17 20 12  8  5
  NORMal  1 +
  SAMPling-points POINTS
       22      0.5    0.25   0.5
       23      0.5    0.5    0.5
       24      0.5    0.75   0.5


**BCSET
  ID-Bcset PRES15
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 5
  PRESsure-bc
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset VELO16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
  VELOcity-bc
    SPLIst ALL
    T-VAlue 1     1.0

$   ZERO VELOCITY BOUNDARY CONDITION ASSUMED ON ELEMENTS  1 THRU 4

**BODY
  CONCentrated
  FREQuencies-concentrated 0.0 1.0
  GMR-concentrated GMR1
  POINts-concentrated 101 102 103 104 111 112 113 114
  POINts-concentrated 121 122 123 124 131 133 133 134
  SOURce-concentrated
  T-COncentrated 1 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 1 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 1 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 1 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 2 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 2 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 2 0.0625 0.0625 0.0625 0.0625
  T-COncentrated 2 0.0625 0.0625 0.0625 0.0625

$ END OF DATA
```

```
-----------------------------------------------------------------
          ****          GPBEST RESULTS           ****
-----------------------------------------------------------------
```

JOB TITLE: ACOUSTIC RESPONSE IN A CUBICAL ENCLOSURE - VELOCITY SPECIFIED
BOUNDARY SOLUTION AT EXCITING FREQUENCY = 0.010000 FOR REGION = GMR1

ELEMENT	NODE NO.	PRESSURE REAL	PRESSURE IMAG	PRESSURE MAGNITUDE	VELOCITY REAL	VELOCITY IMAG	VELOCITY MAGNITUDE
1	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
1	2	0.75861E-06	0.97250E-02	0.97250E-02	0.00000E+00	0.00000E+00	0.00000E+00
1	3	0.13602E-05	0.18368E-01	0.18368E-01	0.00000E+00	0.00000E+00	0.00000E+00
1	5	0.10303E-05	0.18906E-01	0.18906E-01	0.00000E+00	0.00000E+00	0.00000E+00
1	8	0.13602E-05	0.18444E-01	0.18444E-01	0.00000E+00	0.00000E+00	0.00000E+00
1	7	0.75862E-06	0.97599E-02	0.97599E-02	0.00000E+00	0.00000E+00	0.00000E+00
1	6	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
1	4	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

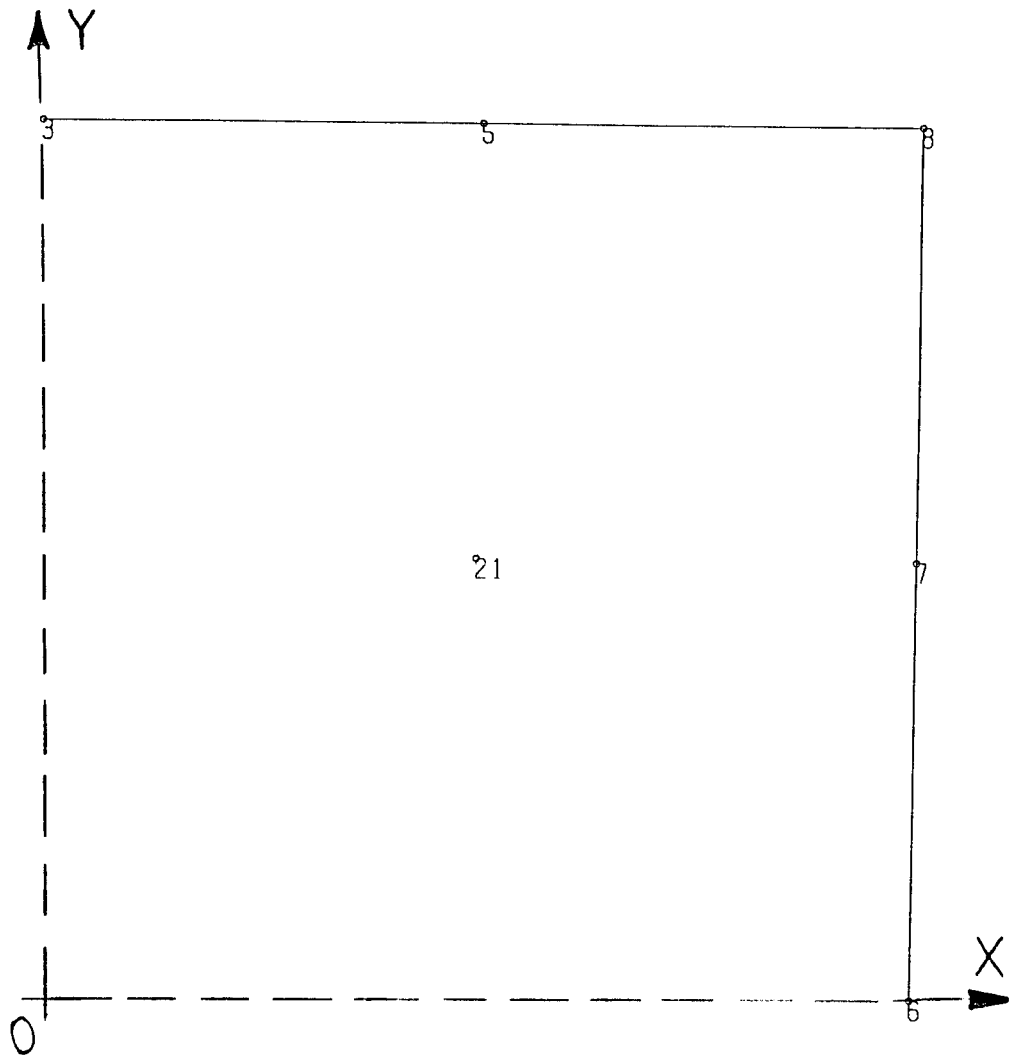
JOB TITLE: ACOUSTIC RESPONSE IN A CUBICAL ENCLOSURE - VELOCITY SPECIFIED
BOUNDARY SOLUTION AT EXCITING FREQUENCY = 1.000000 FOR REGION = GMR1

ELEMENT	NODE NO.	PRESSURE REAL	PRESSURE IMAG	PRESSURE MAGNITUDE	VELOCITY REAL	VELOCITY IMAG	VELOCITY MAGNITUDE
1	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
1	2	0.17637E-01	0.17250E+01	0.17251E+01	0.00000E+00	0.00000E+00	0.00000E+00
1	3	0.32899E-01	0.29262E+01	0.29263E+01	0.00000E+00	0.00000E+00	0.00000E+00
1	5	0.27108E-01	0.29764E+01	0.29765E+01	0.00000E+00	0.00000E+00	0.00000E+00
1	8	0.33491E-01	0.29355E+01	0.29357E+01	0.00000E+00	0.00000E+00	0.00000E+00
1	7	0.17914E-01	0.17294E+01	0.17294E+01	0.00000E+00	0.00000E+00	0.00000E+00
1	6	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
1	4	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

END OF GPBEST ANALYSIS

6.3.2 CONSOLIDATION EXAMPLES

```
EXAMPLE PROBLEM: CONS601

ANALYSIS TYPE: CONSOLIDATION
  2-D QUASISTATIC ANALYSIS

PROBLEM DESCRIPTION:
  UNIDIRECTIONAL LINEAR CONSOLIDATION OF A SATURATED SOIL.
  EIGHT TIME STEPS.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR WITH QUARTER SYMMETRY.  SAMPLING POINTS INCLUDED
  FOR OUTPUT OF DISPLACEMENT, PRESSURE, AND STRESS.

REFERENCE FOR ANALYTICAL SOLUTION:
  LAMBE AND WHITMAN (1969), SOIL MECHANICS, PP408-410.

SOLUTION POINTS TO VERIFY:
  GMR     NODE    TIME             PRESSURE
                             ANALYTICAL        GPBEST

  GMR1      6      2.0         76.4             79.8
                   4.0         46.4             49.3
                   8.0         16.9             18.5

  GMR     NODE    TIME           DISPLACEMENT-Y
                             ANALYTICAL        GPBEST

  GMR1     22      2.0        -4.98            -4.81
                   4.0        -6.89            -6.78
                   8.0        -8.72            -8.65

RUN TIME:
    0.4 X BASE PROBLEM


MISCELLANEOUS:
  THE LOAD BALANCE IS NOT ZERO, BECAUSE SYMMETRY IS EMPLOYED.
```



NODES

```
**CASE
  TITLe    UNIDIRECTIONAL CONSOLIDATION - INTERIOR STRESSES
  PLANe STRAIN
  CONSolidation TRANSIENT
  TIMEs STEP  8  1.0
  SYMMetry QUAD


**MATERIAL
  ID-Material MAT1
    EMODulus 1.0E+04
    POISson 0.1
    DENSity 1.0
    PERMeability 0.1
    FDENsity 0.01


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  POINts
   3       0.0   1000.0
   5     500.0   1000.0
   6    1000.0      0.0
   7    1000.0    500.0
   8    1000.0   1000.0
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface    $ NO ELEMENTS NEEDED ON PLANES OF SYMMETRY
  2    6  7  8
  3    8  5  3
  NORMal  +
  SAMPling-points POINTS
      21     500.0   500.0
      22     500.0  1000.0


**BCSET
  ID-Bcset DISP12
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 2
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0
**BCSET
  ID-Bcset TRAC16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 3
```

```
    TRACtion-bc 2
      SPLIst ALL
      T-VAlue 1  -100.0
    PRESsure-bc
      SPLIst ALL
      T-VAlue 1     0.0     $  DRAINAGE BOUNDARY UNDER LOAD

  $   ZERO FLUX AND TRACTION BOUNDARY CONDITION ASSUMED ON
  $   REMAINING ELEMENTS

  $ END OF DATA
```

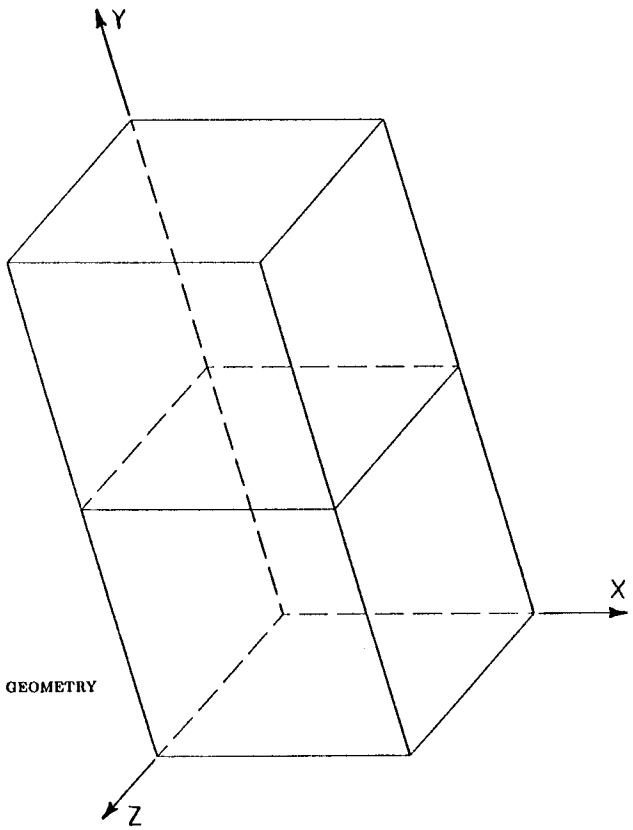
```
JOB TITLE:     UNIDIRECTIONAL CONSOLIDATION - INTERIOR STRESSES
INTERIOR DISPLACEMENT AT TIME =      2.000000           FOR REGION = GMR1

NODE      X DISPLACEMENT      Y DISPLACEMENT      PORE PRESSURE

 21        0.104549E-01       -0.142159E+01        0.566604E+02
 22        0.290749E-01       -0.481486E+01        0.000000E+00
```

```
JOB TITLE:     UNIDIRECTIONAL CONSOLIDATION - INTERIOR STRESSES
INTERIOR DISPLACEMENT AT TIME =      4.000000           FOR REGION = GMR1

NODE      X DISPLACEMENT      Y DISPLACEMENT      PORE PRESSURE

 21       -0.114905E-02       -0.278084E+01        0.337212E+02
 22       -0.387094E-02       -0.677953E+01        0.000000E+00
```

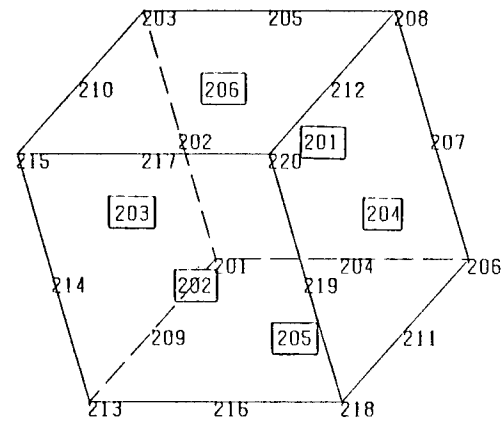
```
JOB TITLE:     UNIDIRECTIONAL CONSOLIDATION - INTERIOR STRESSES
INTERIOR DISPLACEMENT AT TIME =      8.000000           FOR REGION = GMR1

NODE      X DISPLACEMENT      Y DISPLACEMENT      PORE PRESSURE

 21       -0.288728E-02       -0.410394E+01        0.124628E+02
 22       -0.629600E-02       -0.865785E+01        0.000000E+00
```

6.3.3 CONCURRENT-THERMOELASTICITY EXAMPLES

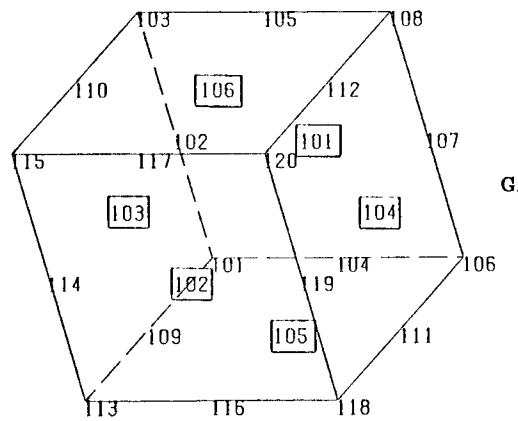
```
EXAMPLE PROBLEM: CTHE601

ANALYSIS TYPE: CONCURRENT THERMOELASTICITY
  3-D, STEADY-STATE THERMOELASTICITY

PROBLEM DESCRIPTION:
  UNIDIRECTIONAL DEFORMATION OF A BLOCK SUBJECTED TO A
  STEADY-STATE TEMPERATURE DISTRIBUTION, ALONG WITH UNIAXIAL
  COMPRESSIVE TRACTIONS.

BOUNDARY ELEMENT MODEL:
  TWO GMR MODEL, WITH SIX SURFACE ELEMENTS IN EACH GMR.
  THE MODEL IS CONSTRAINED IN BOTH THE X AND Z-DIRECTIONS.

REFERENCE FOR ANALYTICAL SOLUTION:
  BOLEY AND WEINER (1960), THEORY OF THERMAL STRESSES, PP. 272-275.

SOLUTION POINTS TO VERIFY:
  GMR     NODE                  DISPLACEMENT-Y
                        ANALYTICAL          GPBEST

  GMR2    201           -0.208E-03        -0.208E-03
  GMR2    203            0.000E+00         0.782E-08

RUN TIME:
    3 X BASE PROBLEM


MISCELLANEOUS:
  THE THERMAL ELONGATION IS EXACTLY COUNTERACTED BY THE APPLIED
  TRACTIONS PRODUCING ZERO OVERALL DISPLACEMENT IN THE Y-DIRECTION.
  RESTART INTEGRATION FILES ARE SAVED.
```



GEOMETRY



GMR2



GMR1

NODES AND ELEMENTS

```
**CASE
  TITLe   ONE-DIMENSIONAL STEADY-STATE THERMOELASTICITY - TWO REGIONS
  CTHErmal STEADY
  RESTart WRITE


**MATERIAL
  ID-Material MAT1
  TEMPeratures   0.0
    EMODulus 10.0E+06
    POISson 0.25
    DENSity 0.1
    ALPHa 10.0E-06
    SPECific 2000.
    CONDuctivity 25.


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration   0.0
  TREFerence   0.0
  POINts
   101       0.0      0.0      0.0
   102       0.0   .25000      0.0
   103       0.0   .50000      0.0
   104    .25000      0.0      0.0
   105    .25000   .50000      0.0
   106    .50000      0.0      0.0
   107    .50000   .25000      0.0
   108    .50000   .50000      0.0
   109       0.0      0.0   .25000
   110       0.0   .50000   .25000
   111    .50000      0.0   .25000
   112    .50000   .50000   .25000
   113       0.0      0.0   .50000
   114       0.0   .25000   .50000
   115       0.0   .50000   .50000
   116    .25000      0.0   .50000
   117    .25000   .50000   .50000
   118    .50000      0.0   .50000
   119    .50000   .25000   .50000
   120    .50000   .50000   .50000
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
  101      101  102  103  105  108  107  106  104
  102      113  114  115  117  120  119  118  116
  103      101  102  103  110  115  114  113  109
  104      106  107  108  112  120  119  118  111
  105      101  104  106  111  118  116  113  109
  106      103  105  108  112  120  117  115  110
```

```
  NORMal  101 +


**GMR
  ID-Gmr GMR2
  MATErial MAT1
  TINTegration   0.0
  TREFerence   0.0
  POINts
   201       0.0   .50000      0.0
   202       0.0   .75000      0.0
   203       0.0  1.00000      0.0
   204    .25000   .50000      0.0
   205    .25000  1.00000      0.0
   206    .50000   .50000      0.0
   207    .50000   .75000      0.0
   208    .50000  1.00000      0.0
   209       0.0   .50000   .25000
   210       0.0  1.00000   .25000
   211    .50000   .50000   .25000
   212    .50000  1.00000   .25000
   213       0.0   .50000   .50000
   214       0.0   .75000   .50000
   215       0.0  1.00000   .50000
   216    .25000   .50000   .50000
   217    .25000  1.00000   .50000
   218    .50000   .50000   .50000
   219    .50000   .75000   .50000
   220    .50000  1.00000   .50000
  SURFace SURF2
    TYPE-surface QUAD
    ELEMents-surface
  201      201  202  203  205  208  207  206  204
  202      213  214  215  217  220  219  218  216
  203      201  202  203  210  215  214  213  209
  204      206  207  208  212  220  219  218  211
  205      201  204  206  211  218  216  213  209
  206      203  205  208  212  220  217  215  210
  NORMal  201 +


**INTERFACE
   GMR-1 GMR1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1 106
   GMR-2 GMR2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2 205
$
$   GMR1 BOUNDARY CONDITIONS
$
**BCSET
  ID-Bcset DISP11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 101  102
  DISPlacement-bc 3
```

```
      SPLIst ALL
      T-VAlue 1 0.0

**BCSET
  ID-Bcset DISP13
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 103  104
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset DISP15
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 105
  DISPlacement-bc 2
    SPLIst ALL
    T-VAlue 1 0.0
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     0.0

$
$   GMR2 BOUNDARY CONDITIONS
$
**BCSET
  ID-Bcset DISP21
  VALUe
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc 201  202
  DISPlacement-bc 3
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset DISP23
  VALUe
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc 203  204
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset TRAC26
  VALUe
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc 206
  TRACtion-bc 2
    SPLIst ALL
    T-VAlue 1  -10000.0   $   COMPRESSIVE Y-TRACTION
```

```
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1   100.0

$   END OF DATA
```

NODE	DISPLACEMENT X/Y/Z	STRESS XX/YY/ZZ	STRESS XY/XZ/YZ	STRAIN XX/YY/ZZ	STRAIN XY/XZ/YZ
201	0.00000E+00	-0.10000E+05	-0.29982E+00	-0.21613E-08	-0.37477E-07
	-0.20832E-03	-0.10001E+05	0.00000E+00	-0.10962E-06	0.00000E+00
	0.00000E+00	-0.10000E+05	-0.29947E+00	-0.21488E-08	-0.37433E-07
202	0.00000E+00	-0.13333E+05	-0.48196E-01	0.12702E-07	-0.60245E-08
	-0.15626E-03	-0.99999E+04	0.00000E+00	0.41667E-03	0.00000E+00
	0.00000E+00	-0.13333E+05	-0.48196E-01	0.12617E-07	-0.60245E-08
203	0.00000E+00	-0.16667E+05	0.57129E-01	-0.55304E-07	0.71411E-08
	0.78219E-08	-0.99993E+04	0.00000E+00	0.83343E-03	0.00000E+00
	0.00000E+00	-0.16667E+05	0.57126E-01	-0.55243E-07	0.71408E-08

EXAMPLE PROBLEM: CTHE602

ANALYSIS TYPE: CONCURRENT THERMOELASTICITY
  2-D QUASISTATIC ANALYSIS

PROBLEM DESCRIPTION:
  SUDDEN HEATING OF A CIRCULAR DISC UNDER PLANE STRAIN CONDITIONS.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR WITH QUARTER SYMMETRY. FOUR BOUNDARY ELEMENTS AND
  ELEVEN INTERIOR SAMPLING POINTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  NOWACKI (1986), THERMOELASTICITY, PP338-340.

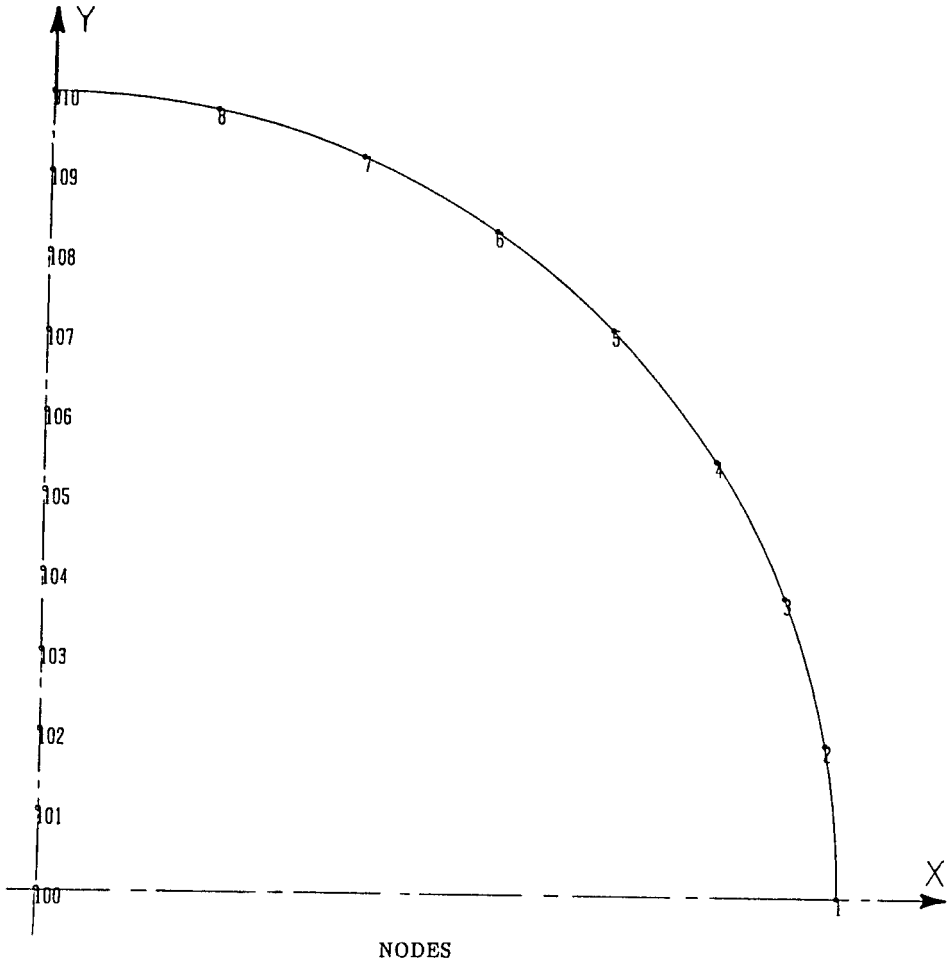
SOLUTION POINTS TO VERIFY:

GMR	NODE	TIME	TEMPERATURE ANALYTICAL	GPBEST
GMR1	105	0.04	0.111	0.105

GMR	NODE	TIME	STRESS-YY ANALYTICAL	GPBEST
GMR1	105	0.04	0.182	0.181

RUN TIME:
    0.9 X BASE PROBLEM

MISCELLANEOUS:
  NO DISPLACEMENT BOUNDARY CONDITIONS ARE NEEDED DUE TO SYMMETRY.

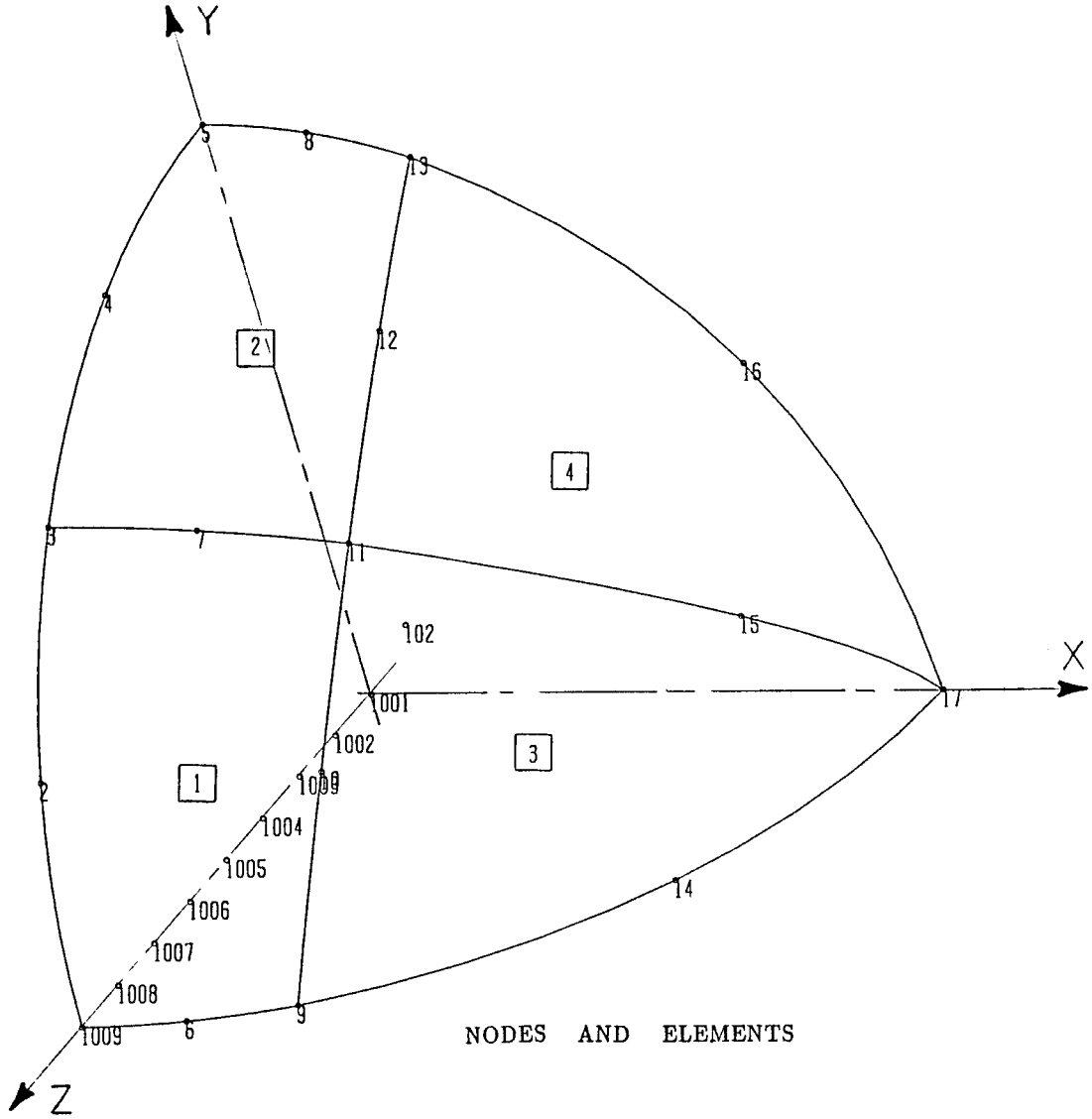

Y
X
110
109
108
107
106
105
104
103
102
101
100
8
7
6
5
4
3
2
1
NODES

```
**CASE
  TITLe  THERMAL STRESSES IN A CIRCULAR DISC - PLANE STRAIN
  PLANe STRAIN
  CTHErmal  TRANSIENT
  TIMEs STEP  8   0.005
  SYMMetry  QUAD
  PRINtout-control BOUNDARY
  PRINtout-control LOAD


**MATERIAL
  ID-Material MAT1
  TEMPeratures 460.0
    EMODulus 1.0
    POISson 0.0
    DENSity 1.0
    ALPHa 1.0
    SPECific 1.0
    CONDuctivity 1.0


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration   0.0
  TREFerence   0.0
  POINts
       1       1.00000       .00000
       2        .98163       .19081
       3        .92718       .37461
       4        .83867       .54464
       5        .70711       .70711
       6        .55919       .82904
       7        .39073       .92050
       8        .20791       .97815
       9        .00000      1.00000
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
   1      1   2   3
   2      3   4   5
   3      5   6   7
   4      7   8   9
  NORMal  +

  SAMPling-points  POINTS
     100        .00000      0.00000
     101        .00000      0.10000
     102        .00000      0.20000
     103        .00000      0.30000
     104        .00000      0.40000
     105        .00000      0.50000
```

```
      106        .00000      0.60000
      107        .00000      0.70000
      108        .00000      0.80000
      109        .00000      0.90000
      110        .00000      1.00000


**BCSET
  ID-Bcset TMP1
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
   ELEMents-bc    1   2   3   4
   TEMPeratures-bc
    SPLIst  ALL
    T-VAlue 1    1.0

$   ZERO TRACTION BOUNDARY CONDITION ASSUMED ON ALL ELEMENTS
$   SYMMETRY PREVENTS RIGID BODY MOTION

$ END OF DATA
```

```
JOB TITLE:   THERMAL STRESSES IN A CIRCULAR DISC - PLANE STRAIN
LOAD CALCULATION AT TIME =      0.040000

LOADS FOR REGION  GMR1

          ELEMENT          X              Y            HEAT RATE
             1         0.00000E+00    0.00000E+00   -0.95151E+00
             2         0.00000E+00    0.00000E+00   -0.99480E+00
             3         0.00000E+00    0.00000E+00   -0.95143E+00
             4         0.00000E+00    0.00000E+00   -0.99473E+00

     LOAD BALANCE      0.00000E+00    0.00000E+00   -0.38925E+01


JOB TITLE:   THERMAL STRESSES IN A CIRCULAR DISC - PLANE STRAIN
INTERIOR DISPLACEMENT AT TIME =      0.040000           FOR REGION = GMR1


NODE      X DISPLACEMENT      Y DISPLACEMENT       TEMPERATURE

100        0.000000E+00        0.951531E-18        0.328110E-02
101        0.477637E-15        0.203781E-01        0.468869E-02
102        0.350836E-15        0.410140E-01        0.100590E-01
103        0.205213E-15        0.623372E-01        0.231994E-01
104       -0.299425E-09        0.852605E-01        0.513498E-01
105       -0.451982E-09        0.111445E+00        0.105387E+00
106       -0.927502E-10        0.143572E+00        0.198112E+00
107        0.186537E-10        0.185277E+00        0.339727E+00
108        0.110639E-09        0.240613E+00        0.531365E+00
109        0.159841E-14        0.312966E+00        0.760033E+00
110       -0.790051E-01        0.387680E+00        0.100000E+01


JOB TITLE:   THERMAL STRESSES IN A CIRCULAR DISC - PLANE STRAIN
INTERIOR STRESS AT TIME =      0.040000           FOR REGION = GMR1


NODE        SIGMAXX            SIGMAYY             TAUXY

100      0.200297E+00       0.200246E+00       0.123232E-16
101      0.199254E+00       0.199880E+00       0.162620E-15
102      0.195079E+00       0.198680E+00      -0.235548E-15
103      0.184626E+00       0.195986E+00      -0.339902E-15
104      0.161838E+00       0.190610E+00      -0.605634E-09
105      0.117560E+00       0.180827E+00       0.939535E-10
106      0.412506E-01       0.164358E+00       0.107748E-08
107     -0.749871E-01       0.138852E+00       0.804248E-09
108     -0.230750E+00       0.102658E+00       0.902096E-10
109     -0.413118E+00       0.555446E-01       0.818251E-13
110     -0.960512E+00      -0.308364E-03      -0.644024E-02
```

```
EXAMPLE PROBLEM: CTHE603

ANALYSIS TYPE: CONCURRENT THERMOELASTICITY
  3-D QUASISTATIC ANALYSIS

PROBLEM DESCRIPTION:
  CONVECTIVE COOLING OF A STEEL SPHERE.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR WITH OCTAL SYMMETRY. FOUR BOUNDARY ELEMENTS AND
  ELEVEN INTERIOR SAMPLING POINTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS,
    PP237-238.
  NOWACKI (1986), THERMOELASTICITY, PP223-225 (INFINITE H ONLY)

SOLUTION POINTS TO VERIFY:
  GMR     NODE    TIME            TEMPERATURE
                             ANALYTICAL        GPBEST

  GMR1    1009    5.00         -8.97            -9.20
                 10.00        -13.45           -13.76

  GMR     NODE    TIME             STRESS-YY
                             ANALYTICAL        GPBEST

  GMR1    1001    5.00          N.A.             -767.
                 10.00          N.A.            -1204.

RUN TIME:
    10 X BASE PROBLEM

MISCELLANEOUS:
  NO DISPLACEMENT BOUNDARY CONDITIONS ARE NEEDED DUE TO SYMMETRY.
  ONLY THE BOUNDARY AND INTERIOR NODAL-BASED SUMMARIES ARE PRINTED.
```


Y
X
Z
1
2
3
4
5
6
7
8
9
11
12
13
14
15
16
17
102
1001
1002
1004
1005
1006
1007
1008
1009
NODES AND ELEMENTS

```
**CASE
    TITLe  COOLING OF A STEEL SPHERE BY IMMERSION IN AN OIL BATH
    CTHErmal   TRANSIENT
    TIMEs STEP  8    1.25
    SYMMetry OCTA
    PRINtout-control NODAL


**MATERIAL
  ID-Material MAT1
  TEMPeratures      0.0
  EMODulus    30.E+6
  POISson    0.30
  DENSity    0.283
  ALPHa    6.0E-6
  SPECific    1000.
  CONDuctivity    5.8


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration     0.0
  TREFerence     0.0
  POINts
    1       .00       .00    1.50000
    2       .00    0.57408   1.38576
    3       .00    1.06056   1.06056
    4       .00    1.38576   0.57408
    5       .00    1.50000       .00
    6    0.26040       .00    1.47720
    7    0.36288   1.02912   1.02912
    8    0.26040   1.47720       .00
    9    0.51312       .00    1.40952
   10    0.64416   0.51840   1.25160
   11    0.68640   0.94296   0.94296
   12    0.64416   1.25160   0.51840
   13    0.51312   1.40952       .00
   14    1.22880       .00    0.86040
   15    1.34424   0.47064   0.47064
   16    1.22880   0.86040       .00
   17    1.50000       .00       .00
 SURFace  SURF1
   TYPE-surface QUAD
   ELEMents-surface
    1       1   2   3   7  11  10   9   6
    2       3   4   5   8  13  12  11   7
    3       9  10  11  15  17  14   0   0
    4      11  12  13  16  17  15   0   0
  NORMal  1  -

  SAMPling-points POINTS
```

```
    102     0.4330  0.4330  0.4330
   1001     0.0     0.0     0.0
   1002     0.0     0.0     0.1875
   1003     0.0     0.0     0.3750
   1004     0.0     0.0     0.5625
   1005     0.0     0.0     0.7500
   1006     0.0     0.0     0.9375
   1007     0.0     0.0     1.1250
   1008     0.0     0.0     1.3125
   1009     0.0     0.0     1.5000


**BCSET
  ID-Bcset BCS4
  RELAtion
  GMR-bc GMR1
  SURFace-bc SURF1
   ELEMents-bc    1  2  3  4
$
$   H = 1.26
$   TEMP(AMBIENT) = -100.0
    CONVection    1.26    -100.0



$   ZERO TRACTION BOUNDARY CONDITION ASSUMED ON ALL ELEMENTS
$   SYMMETRY PREVENTS RIGID BODY MOTION

$ END OF DATA
```

JOB TITLE: COOLING OF A STEEL SPHERE BY IMMERSION IN AN OIL BATH
INTERIOR DISPLACEMENT AT TIME = 10.000000 FOR REGION = GMR1

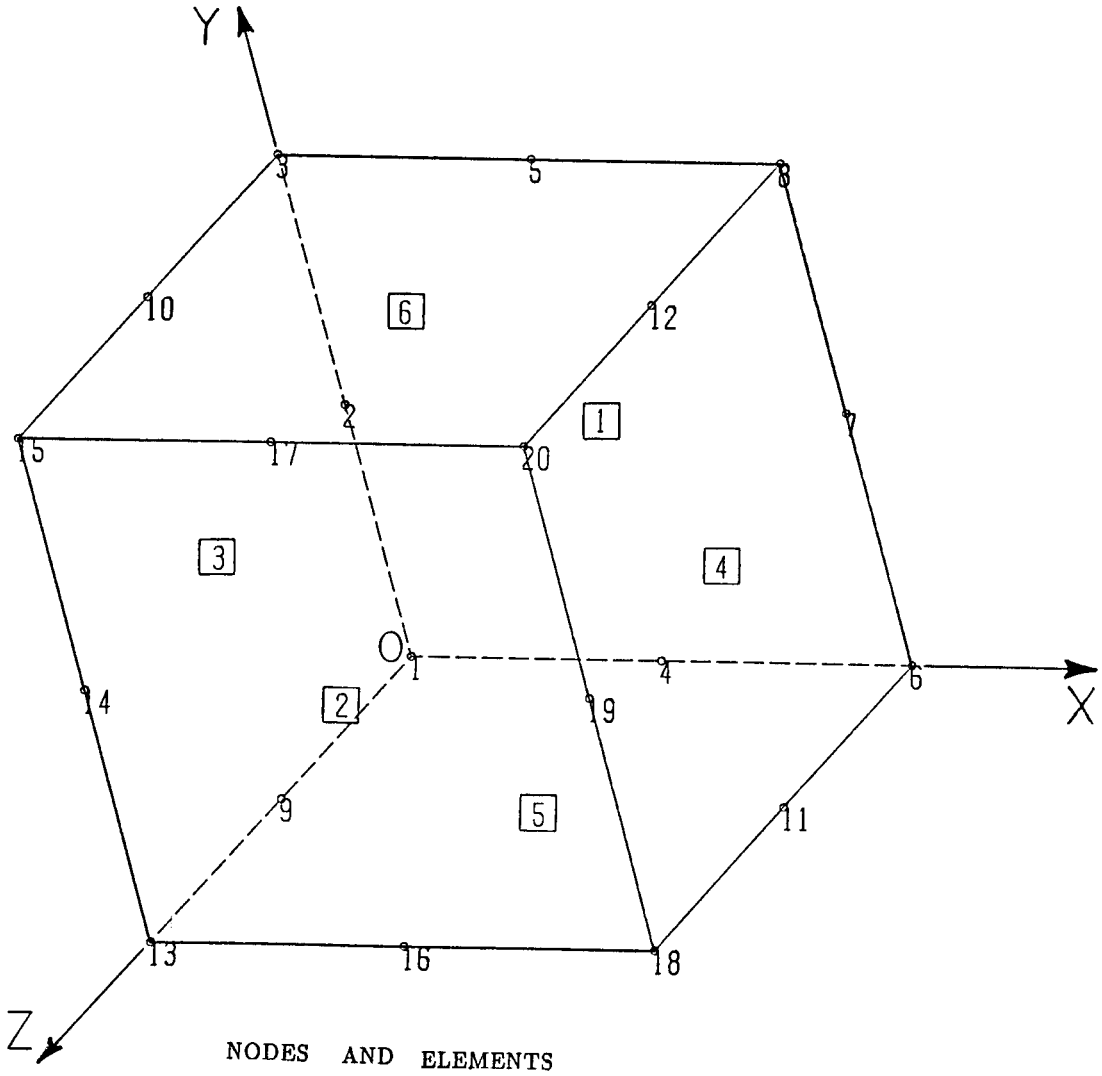
NODE	X DISPLACEMENT	Y DISPLACEMENT	Z DISPLACEMENT	TEMPERATURE
102	-0.135480E-04	-0.135084E-04	-0.135101E-04	-0.418353E+01
1001	0.000000E+00	0.271166E-21	0.134449E-21	-0.160702E+01
1002	0.000000E+00	0.441140E-21	-0.488115E-05	-0.174817E+01
1003	0.000000E+00	-0.313655E-21	-0.101165E-04	-0.218627E+01
1004	0.000000E+00	-0.217509E-21	-0.161161E-04	-0.296514E+01
1005	0.000000E+00	-0.102257E-20	-0.233751E-04	-0.415221E+01
1006	0.000000E+00	-0.486125E-21	-0.324771E-04	-0.581122E+01
1007	0.000000E+00	-0.862803E-22	-0.440803E-04	-0.801040E+01
1008	0.000000E+00	0.162602E-20	-0.588947E-04	-0.107388E+02
1009	0.000000E+00	0.000000E+00	-0.773412E-04	-0.137628E+02

JOB TITLE: COOLING OF A STEEL SPHERE BY IMMERSION IN AN OIL BATH
INTERIOR STRESS AT TIME = 10.000000 FOR REGION = GMR1

NODE	SIGMAXX	SIGMAYY	SIGMAZZ	TAUXY	TAUXZ	TAUYZ
102	-0.765737E+03	-0.761778E+03	-0.761851E+03	-0.920018E+02	-0.919688E+02	-0.933237E+02
1001	-0.120894E+04	-0.120328E+04	-0.120343E+04	0.000000E+00	0.000000E+00	-0.114620E-14
1002	-0.118014E+04	-0.117410E+04	-0.118850E+04	0.971702E-06	0.000000E+00	-0.997182E-14
1003	-0.108907E+04	-0.108347E+04	-0.114400E+04	0.149551E-05	0.000000E+00	-0.109778E-13
1004	-0.925671E+03	-0.922283E+03	-0.106662E+04	-0.305821E-06	0.000000E+00	0.328405E-14
1005	-0.676918E+03	-0.675127E+03	-0.950245E+03	-0.143540E-06	0.000000E+00	-0.936873E-14
1006	-0.329512E+03	-0.328381E+03	-0.788998E+03	0.270395E-06	0.000000E+00	0.765976E-14
1007	0.127424E+03	0.132433E+03	-0.572289E+03	0.738496E-07	0.000000E+00	-0.107172E-14
1008	0.697645E+03	0.701332E+03	-0.302943E+03	0.607076E-07	0.000000E+00	0.650441E-13
1009	0.132811E+04	0.132800E+04	0.269392E+00	0.000000E+00	0.000000E+00	0.000000E+00

6.3.4 ELASTICITY EXAMPLES

```
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:
                    (X-DISPLACEMENT)
      NODE      ANALYTICAL     GPBEST
       12         0.0001       0.0001

                    (Y-DISPLACEMENT)
      NODE        ANALYTICAL         GPBEST
       12        -0.00003000      -0.00002999

                         (X-STRESS)
      NODE      ANALYTICAL     GPBEST
       12         1000.         999.9

RUN TIME:
  1.0 X BASE PROBLEM  ( THIS IS THE BASE PROBLEM! )

MISCELLANEOUS:
```



NODES AND ELEMENTS

```
**CASE
  TITLe     A UNIT CUBE IN SIMPLE TENSION
  ELAStic


**MATERIAL
  ID-Material MAT1
  TEMPeratures  70.0
    EMODulus 10.E6
    POISson 0.3


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration  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-surface QUAD
    ELEMents-surface
  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 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-Bcset DISP1
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 3
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1     0.0

**BCSET
  ID-Bcset DISP2
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 5
  DISPlacement-bc 2
    SPLIst ALL
    T-VAlue 1     0.0

**BCSET
  ID-Bcset DISP3
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 1
  DISPlacement-bc 3
    SPLIst ALL
    T-VAlue 1     0.0

**BCSET
  ID-Bcset TRAC1
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 4
  TRACtion-bc 1
    SPLIst ALL
    T-VAlue 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
    AXISYMMETRY   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 DYNAMIC     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
        NODAL DISP. ,STRESS AND STRAIN     1
        LOAD CALCULATION                   1



         **** MATERIAL INPUT ****

    MATERIAL NAME: MAT1

    ELASTIC
    ISOTROPIC MATERIAL

        DENSITY:          0.0000E+00
        DAMPING COEFF:    0.0000
        POISSONS RATIO:   0.3000


           TEMP          ALPHA            E
       0.70000E+02    0.00000E+00    0.10000E+08



         **** GMR INPUT ****
```

```
REGION     1
NAME GMR1              MATERIAL MAT1

REFERENCE TEMPERATURE          70.00000
INITIAL TEMPERATURE OF GMR      0.00000

NODES    25            ELEMENTS     6       SURFACES     1
SOURCE POINTS    20    CELLS     0          INFINITE ELEMENTS    0
ADDITIONAL JUMP TERM  0.000000              ENCLOSING 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
  NODE       X            Y            Z
    1      0.00000      0.00000      0.00000
    2      0.00000      0.50000      0.00000
    3      0.00000      1.00000      0.00000
    4      0.50000      0.00000      0.00000
    5      0.50000      1.00000      0.00000
    6      1.00000      0.00000      0.00000
    7      1.00000      0.50000      0.00000
    8      1.00000      1.00000      0.00000
    9      0.00000      0.00000      0.50000
   10      0.00000      1.00000      0.50000
   11      1.00000      0.00000      0.50000
   12      1.00000      1.00000      0.50000
   13      0.00000      0.00000      1.00000
   14      0.00000      0.50000      1.00000
   15      0.00000      1.00000      1.00000
   16      0.50000      0.00000      1.00000
   17      0.50000      1.00000      1.00000
   18      1.00000      0.00000      1.00000
   19      1.00000      0.50000      1.00000
   20      1.00000      1.00000      1.00000
   21      0.50000      0.00000      0.50000
   22      0.50000      0.25000      0.50000
   23      0.50000      0.50000      0.50000
   24      0.50000      0.75000      0.50000
   25      0.50000      1.00000      0.50000

SURFACE SURF1         QUADRATIC VARIATION
ELEMENT       NODES
     1          1      2      3      5      8      7      6      4
     2         13     16     18     19     20     17     15     14
     3          1      9     13     14     15     10      3      2
     4          6      7      8     12     20     19     18     11
     5          1      4      6     11     18     16     13      9
     6          3     10     15     17     20     12      8      5

SOURCE POINT LIST
```

```
                1     2     3     4     5     6     7     8     9    10    11    12    13    14    15    16    17
18    19    20


                **** BOUNDARY CONDITION INPUT ****

           BOUNDARY CONDITION SET NAME DISP1           TYPE: VALUE
           GMR GMR1           SURFACE SURF1

           ELEMENT LIST
               3

           SOURCE POINT LIST
               1     9    13    14    15    10     3     2

           COMPONENT   1   DISPLACEMENT INPUT

           DATA VALUES:
            0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00



                **** BOUNDARY CONDITION INPUT ****

           BOUNDARY CONDITION SET NAME DISP2           TYPE: VALUE
           GMR GMR1           SURFACE SURF1

           ELEMENT LIST
               5

           SOURCE POINT LIST
               1     4     6    11    18    16    13     9

           COMPONENT   2   DISPLACEMENT INPUT

           DATA VALUES:
            0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00



                **** BOUNDARY CONDITION INPUT ****

           BOUNDARY CONDITION SET NAME DISP3           TYPE: VALUE
           GMR GMR1           SURFACE SURF1

           ELEMENT LIST
               1

           SOURCE POINT LIST
               1     2     3     5     8     7     6     4

           COMPONENT   3   DISPLACEMENT INPUT

           DATA VALUES:
            0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00
```

```
          **** BOUNDARY CONDITION INPUT ****

     BOUNDARY CONDITION SET NAME TRAC1           TYPE: VALUE
     GMR GMR1           SURFACE SURF1

     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  0.10000E+04



         ** ERROR SUMMARY AFTER INPUT PHASE **

    FATAL ERRORS:               0
    WARNING MESSAGES:           0
    INFORMATIONAL MESSAGES:     1

    INFORMATIONAL MESSAGES LISTED ABOVE:
           1001
1


     BEGIN SURFACE INTEGRATION OF GMR GMR1

          HIGH PRECISION

          SOURCE POINTS

          INTERIOR POINTS

     END SURFACE INTEGRATION OF GMR GMR1

          MATRIX DECOMPOSITION - DIAGONAL BLOCK    1
               CONDITION NUMBER   0.16294E+03




          JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
          BOUNDARY SOLUTION AT TIME =     0.0000       FOR REGION = GMR1
```

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
1	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.22270E+00
1	2	0.00000E+00	-0.15005E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.26818E+00
1	3	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.10535E+00
1	5	0.50004E-04	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00	-0.58124E-01
1	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.20986E+00
1	7	0.10001E-03	-0.15000E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.20890E+00
1	6	0.10001E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.36048E+00
1	4	0.50009E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	-0.16577E+00

2	13	0.00000E+00	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	16	0.50004E-04	0.00000E+00	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	19	0.10001E-03	-0.15000E-04	-0.29997E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	17	0.50000E-04	-0.30010E-04	-0.30010E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	15	0.00000E+00	-0.30005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	14	0.00000E+00	-0.15005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
3	1	0.00000E+00	0.00000E+00	0.00000E+00	-0.10005E+04	0.00000E+00	0.00000E+00
3	9	0.00000E+00	0.00000E+00	-0.15005E-04	-0.10001E+04	0.00000E+00	0.00000E+00
3	13	0.00000E+00	0.00000E+00	-0.30004E-04	-0.10005E+04	0.00000E+00	0.00000E+00
3	14	0.00000E+00	-0.15005E-04	-0.30005E-04	-0.10002E+04	0.00000E+00	0.00000E+00
3	15	0.00000E+00	-0.30005E-04	-0.30005E-04	-0.10004E+04	0.00000E+00	0.00000E+00
3	10	0.00000E+00	-0.30005E-04	-0.15005E-04	-0.10002E+04	0.00000E+00	0.00000E+00
3	3	0.00000E+00	-0.30004E-04	0.00000E+00	-0.10005E+04	0.00000E+00	0.00000E+00
3	2	0.00000E+00	-0.15005E-04	0.00000E+00	-0.10001E+04	0.00000E+00	0.00000E+00
4	6	0.10001E-03	0.00000E+00	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	7	0.10001E-03	-0.15000E-04	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	12	0.10001E-03	-0.29997E-04	-0.15000E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	19	0.10001E-03	-0.15000E-04	-0.29997E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	11	0.10001E-03	0.00000E+00	-0.15000E-04	0.10000E+04	0.00000E+00	0.00000E+00
5	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.22275E+00	0.00000E+00
5	4	0.50009E-04	0.00000E+00	0.00000E+00	0.00000E+00	-0.16570E+00	0.00000E+00
5	6	0.10001E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.36057E+00	0.00000E+00
5	11	0.10001E-03	0.00000E+00	-0.15000E-04	0.00000E+00	0.20893E+00	0.00000E+00
5	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.00000E+00	0.21049E+00	0.00000E+00
5	16	0.50004E-04	0.00000E+00	-0.30011E-04	0.00000E+00	-0.58155E-01	0.00000E+00
5	13	0.00000E+00	0.00000E+00	-0.30004E-04	0.00000E+00	0.10541E+00	0.00000E+00
5	9	0.00000E+00	0.00000E+00	-0.15005E-04	0.00000E+00	0.26828E+00	0.00000E+00

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
6	3	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
6	10	0.00000E+00	-0.30005E-04	-0.15005E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	15	0.00000E+00	-0.30005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	17	0.50000E-04	-0.30010E-04	-0.30010E-04	0.00000E+00	0.00000E+00	0.00000E+00

6	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	12	0.10001E-03	-0.29997E-04	-0.15000E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
6	5	0.50004E-04	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
LOAD CALCULATION AT TIME = 0.000000

LOADS FOR REGION GMR1

ELEMENT	X	Y	Z
1	0.00000E+00	0.00000E+00	0.95296E-02
2	0.00000E+00	0.00000E+00	0.00000E+00
3	-0.10000E+04	0.00000E+00	0.00000E+00
4	0.10000E+04	0.00000E+00	0.00000E+00
5	0.00000E+00	0.95156E-02	0.00000E+00
6	0.00000E+00	0.00000E+00	0.00000E+00
LOAD BALANCE	-0.20203E-01	0.95156E-02	0.95296E-02

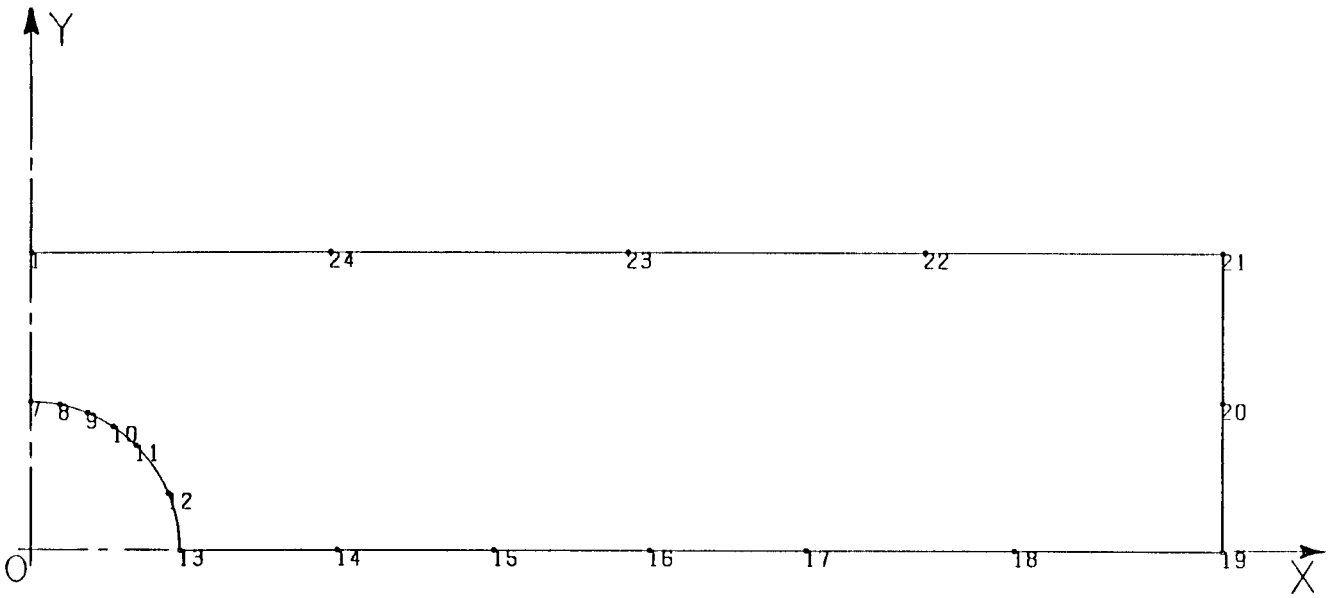
JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
NODAL OUTPUT AT TIME = 0.000000 FOR REGION = GMR1

NODE	DISPLACEMENT	STRESS		STRAIN	
	X/Y/Z	XX/YY/ZZ	XY/XZ/YZ	XX/YY/ZZ	XY/XZ/YZ
1	0.00000E+00	0.10003E+04	0.00000E+00	0.10004E-03	0.00000E+00
	0.00000E+00	-0.12547E+00	0.00000E+00	-0.30017E-04	0.00000E+00
	0.00000E+00	-0.12546E+00	0.00000E+00	-0.30017E-04	0.00000E+00
2	0.00000E+00	0.10002E+04	-0.42452E-01	0.10003E-03	-0.55188E-08
	-0.15005E-04	-0.36594E-01	0.00000E+00	-0.30004E-04	0.00000E+00
	0.00000E+00	-0.19430E+00	-0.96197E-04	-0.30025E-04	-0.12506E-10
3	0.00000E+00	0.10003E+04	-0.53161E-01	0.10003E-03	-0.69109E-08
	-0.30004E-04	0.11022E+00	0.00000E+00	-0.29997E-04	0.00000E+00
	0.00000E+00	-0.35271E-01	-0.13619E-02	-0.30016E-04	-0.17705E-09
4	0.50009E-04	0.10001E+04	0.46846E-01	0.10001E-03	0.60900E-08
	0.00000E+00	-0.66880E-02	0.46853E-01	-0.30002E-04	0.60909E-08
	0.00000E+00	-0.66681E-02	0.00000E+00	-0.30002E-04	0.00000E+00
5	0.50004E-04	0.99992E+03	-0.41151E-01	0.99994E-04	-0.53496E-08
	-0.30011E-04	0.13870E-01	0.49553E-01	-0.29994E-04	0.64420E-08
	0.00000E+00	-0.90094E-01	0.00000E+00	-0.30007E-04	0.00000E+00
6	0.10001E-03	0.99975E+03	0.60773E-01	0.99990E-04	0.79005E-08
	0.00000E+00	-0.25114E+00	0.60773E-01	-0.30010E-04	0.79005E-08
	0.00000E+00	-0.25113E+00	0.00000E+00	-0.30010E-04	0.00000E+00

```
  7         0.10001E-03    0.99991E+03    0.40438E-01    0.99995E-04    0.52569E-08
           -0.15000E-04    0.17284E-01    0.00000E+00   -0.29991E-04    0.00000E+00
            0.00000E+00   -0.14902E+00   -0.16546E-01   -0.30013E-04   -0.21510E-08

  8         0.99994E-04    0.99976E+03   -0.33954E-02    0.99976E-04   -0.44141E-09
           -0.29991E-04    0.10892E+00    0.66612E-01   -0.29978E-04    0.86596E-08
            0.00000E+00   -0.12321E+00   -0.24540E-01   -0.30008E-04   -0.31902E-08

  9         0.00000E+00    0.10002E+04    0.00000E+00    0.10003E-03    0.00000E+00
            0.00000E+00   -0.19431E+00   -0.42451E-01   -0.30025E-04   -0.55186E-08
           -0.15005E-04   -0.36621E-01   -0.10669E-03   -0.30004E-04   -0.13870E-10

 10         0.00000E+00    0.10002E+04    0.00000E+00    0.10002E-03    0.00000E+00
           -0.30005E-04    0.52551E-01   -0.29686E-01   -0.30003E-04   -0.38592E-08
           -0.15005E-04    0.36850E-01   -0.20376E-02   -0.30005E-04   -0.26489E-09

 11         0.10001E-03    0.99991E+03    0.00000E+00    0.99995E-04    0.00000E+00
            0.00000E+00   -0.14889E+00    0.40439E-01   -0.30013E-04    0.52571E-08
           -0.15000E-04    0.17349E-01   -0.16598E-01   -0.29991E-04   -0.21578E-08

 12         0.10001E-03    0.99990E+03    0.00000E+00    0.99985E-04    0.00000E+00
           -0.29997E-04    0.10396E+00    0.32747E-01   -0.29988E-04    0.42571E-08
           -0.15000E-04    0.59250E-01    0.89620E-02   -0.29994E-04    0.11651E-08

 13         0.00000E+00    0.10003E+04    0.00000E+00    0.10003E-03    0.00000E+00
            0.00000E+00   -0.35310E-01   -0.53161E-01   -0.30016E-04   -0.69109E-08
           -0.30004E-04    0.11018E+00   -0.13642E-02   -0.29997E-04   -0.17735E-09


    JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
    NODAL OUTPUT AT TIME =      0.000000      FOR REGION = GMR1


NODE       DISPLACEMENT    --------- STRESS ---------    --------- STRAIN ---------
              X/Y/Z          XX/YY/ZZ      XY/XZ/YZ        XX/YY/ZZ       XY/XZ/YZ
-------------------------------------------------------------------------------------

 14         0.00000E+00    0.10002E+04   -0.29636E-01    0.10002E-03   -0.38526E-08
           -0.15005E-04    0.36823E-01    0.00000E+00   -0.30005E-04    0.00000E+00
           -0.30005E-04    0.52612E-01   -0.20376E-02   -0.30003E-04   -0.26489E-09

 15         0.00000E+00    0.10002E+04   -0.37327E-01    0.10002E-03   -0.48525E-08
           -0.30005E-04    0.10503E+00   -0.37366E-01   -0.30000E-04   -0.48576E-08
           -0.30005E-04    0.10514E+00   -0.25862E-02   -0.30000E-04   -0.33621E-09

 16         0.50004E-04    0.99992E+03    0.49539E-01    0.99994E-04    0.64401E-08
            0.00000E+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         0.50000E-04    0.99986E+03   -0.43995E-01    0.99987E-04   -0.57194E-08
           -0.30010E-04   -0.18402E-01   -0.43992E-01   -0.29997E-04   -0.57189E-08
           -0.30010E-04   -0.18250E-01    0.00000E+00   -0.29997E-04    0.00000E+00

 18         0.99994E-04    0.99976E+03    0.66575E-01    0.99976E-04    0.86548E-08
            0.00000E+00   -0.12329E+00   -0.33861E-02   -0.30008E-04   -0.44020E-09
```

```
                -0.29991E-04    0.10891E+00   -0.24587E-01   -0.29978E-04   -0.31963E-08

     19          0.10001E-03    0.99990E+03    0.32693E-01    0.99985E-04    0.42501E-08
                -0.15000E-04    0.58983E-01    0.00000E+00   -0.29994E-04    0.00000E+00
                -0.29997E-04    0.10406E+00    0.89480E-02   -0.29988E-04    0.11632E-08

     20          0.99987E-04    0.99976E+03   -0.20816E-01    0.99969E-04   -0.27061E-08
                -0.29994E-04    0.10793E+00   -0.20753E-01   -0.29985E-04   -0.26979E-08
                -0.29994E-04    0.10841E+00    0.34034E-01   -0.29985E-04    0.44244E-08


        JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
        INTERIOR DISPLACEMENT AT TIME =     0.0000       FOR REGION = GMR1

    NODE      X DISPLACEMENT    Y DISPLACEMENT    Z DISPLACEMENT

     21      0.500135E-04      0.000000E+00     -0.150089E-04
     22      0.500090E-04     -0.750441E-05     -0.150050E-04
     23      0.500105E-04     -0.150052E-04     -0.150052E-04
     24      0.500067E-04     -0.225076E-04     -0.150046E-04
     25      0.500091E-04     -0.300125E-04     -0.150074E-04


        JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
        INTERIOR STRESS AT TIME =     0.0000       FOR REGION = GMR1

    NODE      SIGMA-XX          SIGMA-YY          SIGMA-ZZ          TAU-XY            TAU-XZ            TAU-YZ

     21      0.100017E+04      0.981304E-01     -0.332382E-01      0.000000E+00      0.546809E-03      0.000000E+00
     22      0.999768E+03      0.227315E+00     -0.385985E-01     -0.165191E+00     -0.125456E-02      0.643814E-
01
     23      0.100012E+04     -0.556403E-01     -0.556958E-01     -0.231984E-02     -0.232729E-02      0.584437E-
02
     24      0.999633E+03      0.393963E+00     -0.154854E+00      0.160049E+00     -0.247152E-03     -0.514198E-
01
     25      0.100003E+04      0.000000E+00     -0.882590E-01      0.000000E+00      0.225028E-02      0.000000E+00


    END OF ANALYSIS
```

```
EXAMPLE PROBLEM: ELAS606

ANALYSIS TYPE:
  2-D, PLANE STRESS, ELASTICITY

PROBLEM DESCRIPTION :
  STRIP WITH A HOLE IN UNIFORM TENSION.  HOLE DIAMETER IS EXACTLY
  HALF OF THE PLATE WIDTH.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR, 9 BOUNDARY ELEMENTS, 19 BOUNDARY NODES.  QUARTER
  SYMMETRY IS USED TO MODEL REGION IN THE POSITIVE XY QUADRANT.
  SAMPLING POINTS ON THE CENTERLINE ARE SUPPLIED TO MONITOR INTERNAL
  BEHAVIOR OF STRESS, ESPECIALLY CLOSE TO HOLE.

REFERENCE FOR ANALYTICAL SOLUTION:
  PETERSON, STRESS CONCENTRATION FACTORS (1958), PG. 150.
  STRESS AT HOLE IS 4.3 TIMES THE UNIFORM STRESS APPLIED AWAY FROM
  HOLE.  NOTE THAT THE WELL-KNOWN VALUE OF 3.0 APPLIES TO A
  HOLE IN AN INFINITE PLATE.  HERE WE HAVE A FINITE PLATE, SO THE
  STRESS CONCENTRATION FACTOR IS HIGHER.
    THE COMPLETE ANALYTICAL SOLUTION IS FOUND IN PHIL. TRANS. ROYAL
  SOCIETY (LONDON) A, VOL. 229 (1929-30), P. 67, R. HOWLAND.

SOLUTION POINTS TO VERIFY :

      (STRESS SIG-XX AT TOP OF HOLE)
     NODE     ANALYTICAL      GPBEST
       7         4.3            4.1

RUN TIME:
   0.2 X BASE PROBLEM

MISCELLANEOUS :
   NOTE THAT THE MESH IS UNEVENLY SPACED ON TOP AND BOTTOM SIDES
   OF THE STRIP.  MORE ELEMENTS ARE PLACED ON THE BOTTOM WHERE
   THE SOLUTION CHANGES MORE RAPIDLY.
```



NODES

```
**CASE
  TITLe  2-D PLATE WITH A HOLE IN UNIFORM TENSION
  PLANe STRESS           $ SPECIFIES 2-D PLANE STRESS ANALYSIS.
  ELAStic
  SYMMetry HALF              $ MODEL IS SYMMETRIC ABOUT Y-Z PLANE


**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus  10.E6
  POISson  .3


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration 70.0
  POINts
    1       0.00000      1.00000
    7       0.00000      0.50000
    8       0.09750      0.49039
    9       0.19134      0.46194
   10       0.27780      0.41570
   11       0.35350      0.35350
   12       0.46194      0.19134
   13       0.50000      0.00000
   14       1.02500      0.00000
   15       1.55000      0.00000
   16       2.07500      0.00000
   17       2.60000      0.00000
   18       3.30000      0.00000
   19       4.00000      0.00000
   20       4.00000      0.50000
   21       4.00000      1.00000
   22       3.00000      1.00000
   23       2.00000      1.00000
   24       1.00000      1.00000
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
       4       7     8     9
       5       9    10    11
       6      11    12    13
       7      13    14    15
       8      15    16    17
       9      17    18    19
      10      19    20    21
      11      21    22    23
      12      23    24     1
   NORMal +
   SAMPling-points POINTS
```

```
        61       0.00000      0.90000
        62       0.00000      0.80000
        63       0.00000      0.70000
        64       0.00000      0.60000
        65       0.00000      0.51000
        66       0.00000      0.50500


  **BCSET
    ID-Bcset  DISP12
    GMR-bc  GMR1
    SURFace-bc  SURF1
      ELEMents-bc     7   8   9
    DISPlacement-bc  2
      SPLIst ALL
        T-VAlue  1   0.0

  $  SPECIFY UNIFORM TENSION:
  **BCSET
    ID-Bcset  TRAC
    GMR-bc  GMR1
    SURFace-bc  SURF1
      ELEMents-bc    10
    TRACtion-bc 1
      SPLIst ALL
        T-VAlue  1   1.0
  $
  $  END OF DATA
```

JOB TITLE: 2-D PLATE WITH A HOLE IN UNIFORM TENSION
BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	X-TRAC.	Y-TRAC.
4	7	-0.29516E-16	-0.84159E-07	0.00000E+00	0.00000E+00
4	8	0.39693E-07	-0.81619E-07	0.00000E+00	0.00000E+00
4	9	0.77259E-07	-0.74677E-07	0.00000E+00	0.00000E+00
5	9	0.77259E-07	-0.74677E-07	0.00000E+00	0.00000E+00
5	10	0.11090E-06	-0.64605E-07	0.00000E+00	0.00000E+00
5	11	0.13959E-06	-0.52572E-07	0.00000E+00	0.00000E+00
6	11	0.13959E-06	-0.52572E-07	0.00000E+00	0.00000E+00
6	12	0.17897E-06	-0.26187E-07	0.00000E+00	0.00000E+00
6	13	0.19276E-06	0.00000E+00	0.00000E+00	0.00000E+00
7	13	0.19276E-06	0.00000E+00	0.00000E+00	0.94969E+00
7	14	0.20685E-06	0.00000E+00	0.00000E+00	-0.11990E+00
7	15	0.24139E-06	0.00000E+00	0.00000E+00	-0.72287E-01
8	15	0.24139E-06	0.00000E+00	0.00000E+00	-0.72287E-01
8	16	0.28676E-06	0.00000E+00	0.00000E+00	-0.75955E-01
8	17	0.33722E-06	0.00000E+00	0.00000E+00	-0.12352E-01
9	17	0.33722E-06	0.00000E+00	0.00000E+00	-0.12352E-01
9	18	0.40690E-06	0.00000E+00	0.00000E+00	0.18092E-02
9	19	0.47698E-06	0.00000E+00	0.00000E+00	0.22946E-01
10	19	0.47698E-06	0.00000E+00	0.10000E+01	0.00000E+00
10	20	0.47730E-06	-0.15723E-07	0.10000E+01	0.00000E+00
10	21	0.47783E-06	-0.31134E-07	0.10000E+01	0.00000E+00
11	21	0.47783E-06	-0.31134E-07	0.00000E+00	0.00000E+00
11	22	0.37789E-06	-0.29935E-07	0.00000E+00	0.00000E+00
11	23	0.27705E-06	-0.26323E-07	0.00000E+00	0.00000E+00
12	23	0.27705E-06	-0.26323E-07	0.00000E+00	0.00000E+00
12	24	0.14812E-06	-0.32305E-07	0.00000E+00	0.00000E+00
12	1	0.35408E-15	-0.10721E-06	0.00000E+00	0.00000E+00

JOB TITLE: 2-D PLATE WITH A HOLE IN UNIFORM TENSION
LOAD CALCULATION AT TIME = 0.000000

LOADS FOR REGION GMR1

ELEMENT	X	Y
4	0.00000E+00	0.00000E+00
5	0.00000E+00	0.00000E+00
6	0.00000E+00	0.00000E+00
7	0.00000E+00	0.69613E-01
8	0.00000E+00	-0.67981E-01
9	0.00000E+00	0.41606E-02
10	0.10000E+01	0.00000E+00
11	0.00000E+00	0.00000E+00
12	0.00000E+00	0.00000E+00
LOAD BALANCE	0.10000E+01	0.57925E-02

JOB TITLE: 2-D PLATE WITH A HOLE IN UNIFORM TENSION
NODAL OUTPUT AT TIME = 0.000000 FOR REGION = GMR1

NODE	DISPLACEMENT	STRESS		STRAIN	
	X/Y	XX/YY	XY/ZZ	XX/YY	XY/ZZ
1	0.35408E-15	0.15771E+01	0.00000E+00	0.15771E-06	0.00000E+00
	-0.10721E-06	0.00000E+00	0.00000E+00	-0.47314E-07	-0.47314E-07
7	-0.29516E-16	0.41031E+01	-0.78481E-02	0.41031E-06	-0.10202E-08
	-0.84159E-07	0.15011E-04	0.00000E+00	-0.12309E-06	-0.12309E-06
8	0.39693E-07	0.36451E+01	-0.72505E+00	0.36018E-06	-0.94256E-07
	-0.81619E-07	0.14422E+00	0.00000E+00	-0.94930E-07	-0.11368E-06
9	0.77259E-07	0.25789E+01	-0.10678E+01	0.24462E-06	-0.13882E-06
	-0.74677E-07	0.44217E+00	0.00000E+00	-0.33149E-07	-0.90630E-07
10	0.11090E-06	0.14000E+01	-0.93620E+00	0.12122E-06	-0.12171E-06
	-0.64605E-07	0.62606E+00	0.00000E+00	0.20606E-07	-0.60781E-07
11	0.13959E-06	0.44176E+00	-0.44694E+00	0.30608E-07	-0.58102E-07
	-0.52572E-07	0.45226E+00	0.00000E+00	0.31974E-07	-0.26821E-07
12	0.17897E-06	-0.10805E+00	0.26072E+00	0.80684E-08	0.33894E-07
	-0.26187E-07	-0.62911E+00	0.00000E+00	-0.59670E-07	0.22115E-07
13	0.19276E-06	-0.10573E+00	0.88199E-02	0.22658E-07	0.11466E-08
	0.00000E+00	-0.11077E+01	0.00000E+00	-0.10760E-06	0.36403E-07
14	0.20685E-06	0.49908E+00	0.00000E+00	0.46311E-07	0.00000E+00
	0.00000E+00	0.11990E+00	0.00000E+00	-0.29820E-08	-0.18570E-07
15	0.24139E-06	0.85578E+00	0.00000E+00	0.83409E-07	0.00000E+00
	0.00000E+00	0.72287E-01	0.00000E+00	-0.18445E-07	-0.27842E-07
16	0.28676E-06	0.93542E+00	0.00000E+00	0.91264E-07	0.00000E+00

```
        0.00000E+00    0.75955E-01    0.00000E+00   -0.20467E-07   -0.30341E-07

 17     0.33722E-06    0.10048E+01    0.00000E+00    0.10011E-06    0.00000E+00
        0.00000E+00    0.12352E-01    0.00000E+00   -0.28910E-07   -0.30516E-07

 18     0.40690E-06    0.99776E+00    0.00000E+00    0.99830E-07    0.00000E+00
        0.00000E+00   -0.18092E-02    0.00000E+00   -0.30114E-07   -0.29879E-07

 19     0.47698E-06    0.99854E+00    0.00000E+00    0.10046E-06    0.00000E+00
        0.00000E+00   -0.20263E-01    0.00000E+00   -0.31983E-07   -0.29348E-07

 20     0.47730E-06    0.10000E+01    0.00000E+00    0.10034E-06    0.00000E+00
       -0.15723E-07   -0.11335E-01    0.00000E+00   -0.31134E-07   -0.29660E-07

 21     0.47783E-06    0.99748E+00    0.00000E+00    0.99825E-07    0.00000E+00
       -0.31134E-07   -0.25455E-02    0.00000E+00   -0.30179E-07   -0.29848E-07

 22     0.37789E-06    0.10039E+01    0.00000E+00    0.10039E-06    0.00000E+00
       -0.29935E-07    0.00000E+00    0.00000E+00   -0.30117E-07   -0.30117E-07


          JOB TITLE:   2-D PLATE WITH A HOLE IN UNIFORM TENSION
          NODAL OUTPUT AT TIME =      0.000000      FOR REGION = GMR1


NODE   DISPLACEMENT    --------- STRESS ---------    --------- STRAIN ---------
           X/Y            XX/YY         XY/ZZ           XX/YY          XY/ZZ
-------------------------------------------------------------------------------

 23     0.27705E-06    0.11031E+01    0.00000E+00    0.11031E-06    0.00000E+00
       -0.26323E-07    0.00000E+00    0.00000E+00   -0.33093E-07   -0.33093E-07

 24     0.14812E-06    0.13852E+01    0.00000E+00    0.13852E-06    0.00000E+00
       -0.32305E-07    0.00000E+00    0.00000E+00   -0.41557E-07   -0.41557E-07


         JOB TITLE:   2-D PLATE WITH A HOLE IN UNIFORM TENSION
         INTERIOR DISPLACEMENT AT TIME =     0.0000       FOR REGION = GMR1

  NODE      X DISPLACEMENT    Y DISPLACEMENT

   61      0.565058E-16     -0.101368E-06
   62      0.359710E-17     -0.977514E-07
   63     -0.495568E-16     -0.949293E-07
   64     -0.102774E-15     -0.915923E-07
   65     -0.150529E-15     -0.853088E-07
   66      0.000000E+00     -0.841588E-07


         JOB TITLE:   2-D PLATE WITH A HOLE IN UNIFORM TENSION
         INTERIOR STRESS AT TIME =     0.0000       FOR REGION = GMR1

  NODE      SIGMA-XX          SIGMA-YY          TAU-XY            SIGMA-ZZ
```

```
61      0.154204E+01      0.309403E-01      0.189964E-08
62      0.177599E+01      0.225919E+00      0.193089E-08
63      0.209788E+01      0.352586E+00      0.197992E-08
64      0.269986E+01      0.367918E+00      0.203968E-08
65      0.388281E+01      0.795972E-01      0.209667E-08
66      0.410310E+01      0.150111E-04      0.000000E+00
```

EXAMPLE PROBLEM: ELAS607

ANALYSIS TYPE : 2D ANISOTROPIC ELASTOSTATICS

PROBLEM DESCRIPTION :
CENTRIFUGAL BODY FORCE PROBLEM FOR A SOLID CIRCULAR DISK MADE OF A MATERIAL THAT CAN BE CONSIDERED CROSS-ANISOTROPIC. (NOTE: CROSS ANISOTROPY IS A SPECIAL CASE OF ORTHOTROPY).

THIS IS AN EXAMPLE OF A THIN SOLID DISK MADE OF A FIBRE REINFORCED COMPOSITE WITH THE TRADE NAME T300/5208 (VOLUME FRACTION OF FIBRES = 0.6). BECAUSE OF AXISYMMETRY ONLY A QUARTER PLATE (SINGLE GMR) ANALYSIS HAS BEEN CARRIED OUT. THE STRAIGHT EDGES CORRESPONDING TO THE RADIAL DIRECTIONS ALONG THE COORDINATE AXES ARE PLACED ON ROLLER SUPPORTS. THE DIRECTIONS OF THE PRINCIPAL MATERIAL AXES ARE COINCIDENT WITH THOSE OF THE COORDINATE AXES.

BOUNDARY ELEMENT MODEL :
46 QUADRATIC ELEMENTS ALONG THE BOUNDARY OF THE QUARTER PLATE; 15 ALONG EACH OF THE TWO STRAIGHT EDGES AND 16 ALONG THE CURVED BOUNDARY. THE STRAIGHT EDGES WHICH ARE ALONG RADIAL DIRECTIONS COINCIDING WITH THE GLOBAL COORDINATE AXES AND ARE SUPPORTED ON ROLLERS.

REFERENCE FOR ANALYTICAL SOLUTION:
LEKHNITSKII, 'ANISOTROPIC PLATES',PP.148.

SOLUTION POINTS TO VERIFY :

(RADIAL STRESS SIG-XX)

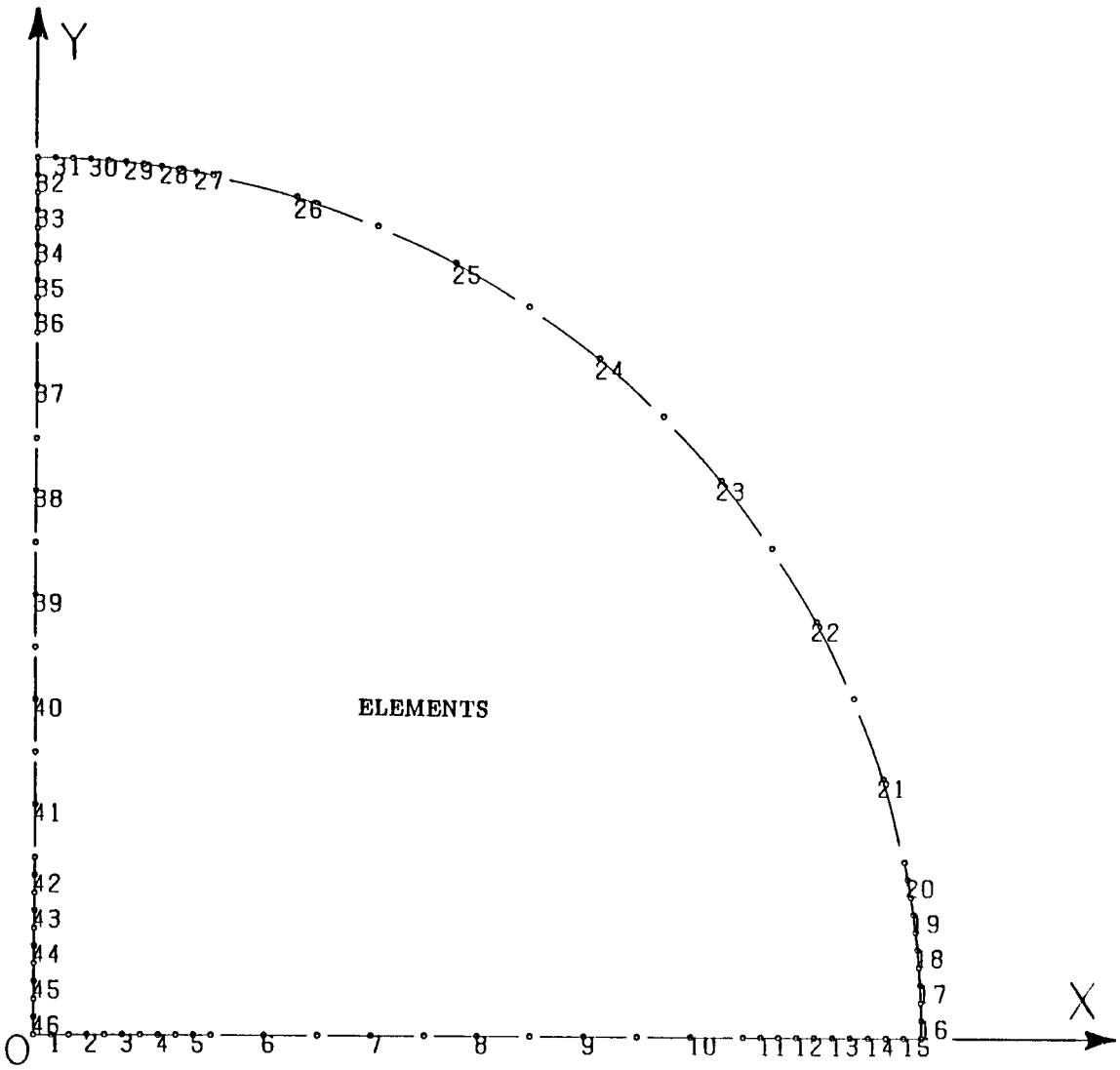
NODE	ANALYTICAL	GPBEST
101	39.74E+4	39.70E+4
108	32.05E+4	32.05E+4

(TANGENTIAL STRESS SIG-YY)

NODE	ANALYTICAL	GPBEST
101	39.74E+4	39.72E+4
108	36.02E+4	36.02E+4

RUN TIME :
2 X BASE PROBLEM

MISCELLANEOUS:


Y
X
O
ELEMENTS
31 30 29 28 27
26
25
24
23
22
21
20
19
18
17
16
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46

```
**CASE
  TITLe  THIN SOLID (T300/5208)DISK ROTATING WITH A CONSTANT SPEED
  PLANe STRESS
  PRINtout-control BOUNDARY
  PRINtout-control LOAD


**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  DENSity 1.0         $ NOTE : THIS DENSITY IS FICTITIOUS AND DOES
                      $ NOT CORRESPOND TO THAT OF THE MATERIAL
                      $ MENTIONED ABOVE
  ANISotropy ORTHO
  TECHnical 1.31E+05  0.13E+05  0.13E+05  0.064E+05  0.038  0.038  0.492
$        E1         E2         E3         G12       NU12   NU13   NU23
  ORIEntation 0.


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration 70.0
  POINts
      1           .00000           .00000
      2           .20000           .00000
      3           .40000           .00000
      4           .60000           .00000
      5           .80000           .00000
      6          1.00000           .00000
      7          1.20000           .00000
      8          1.40000           .00000
      9          1.60000           .00000
     10          1.80000           .00000
     11          2.00000           .00000
     12          2.60000           .00000
     13          3.20000           .00000
     14          3.80000           .00000
     15          4.40000           .00000
     16          5.00000           .00000
     17          5.60000           .00000
     18          6.20000           .00000
     19          6.80000           .00000
     20          7.40000           .00000
     21          8.00000           .00000
     22          8.20000           .00000
     23          8.40000           .00000
     24          8.60000           .00000
     25          8.80000           .00000
     26          9.00000           .00000
     27          9.20000           .00000
     28          9.40000           .00000
```

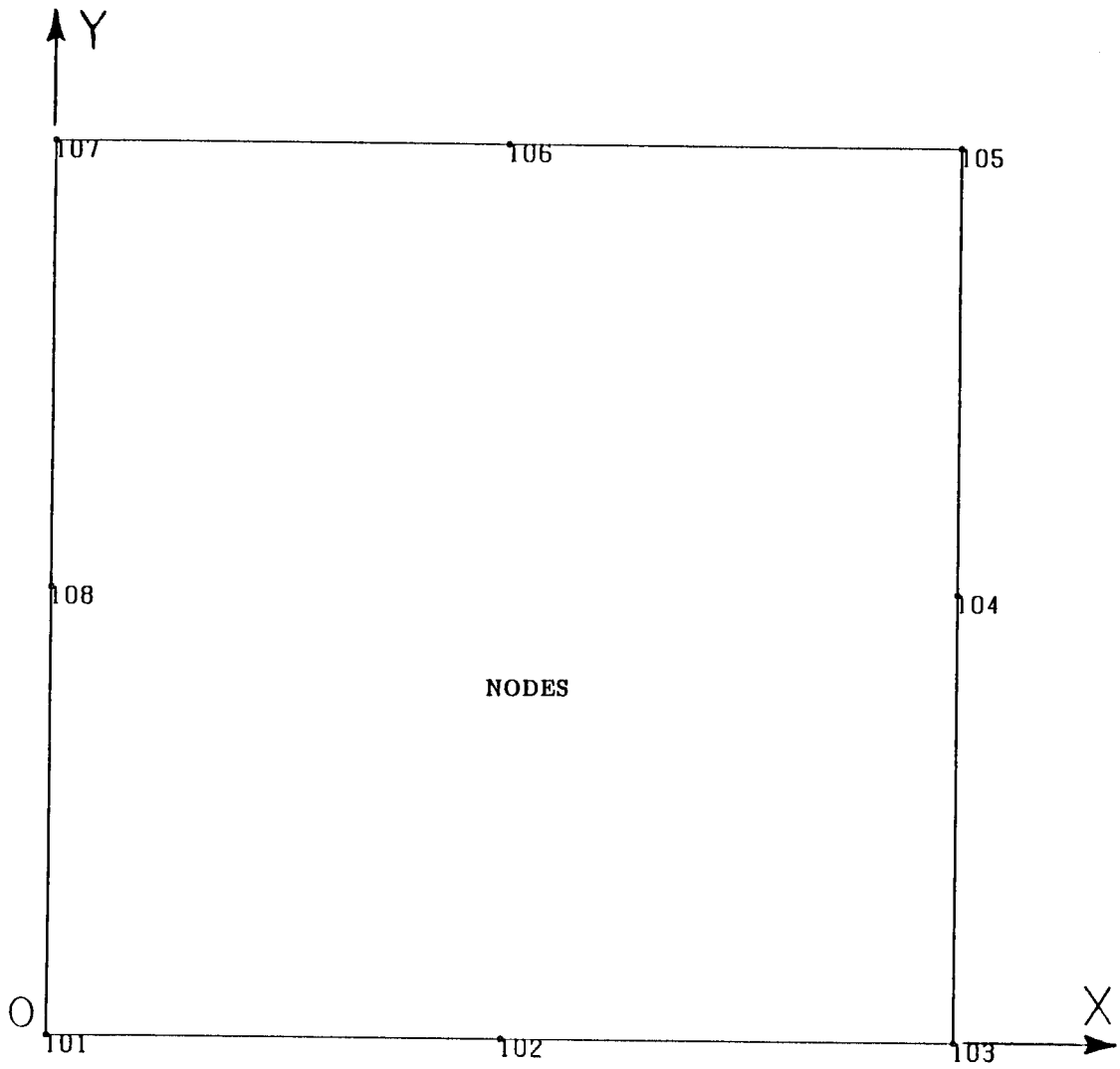
29	9.60000	.00000
30	9.80000	.00000
31	10.00000	.00000
32	9.99800	.19957
33	9.99200	.39953
34	9.98200	.59934
35	9.96801	.79890
36	9.95003	.99815
37	9.92806	1.19700
38	9.90213	1.39536
39	9.87223	1.59317
40	9.83838	1.79034
41	9.80060	1.98680
42	9.56047	2.93203
43	9.22940	3.84937
44	8.81055	4.73011
45	8.30790	5.56585
46	7.72623	6.34865
47	7.07107	7.07107
48	6.34865	7.72623
49	5.56585	8.30790
50	4.73011	8.81055
51	3.84937	9.22940
52	2.93203	9.56047
53	1.98680	9.80060
54	1.79039	9.83838
55	1.59327	9.87222
56	1.39551	9.90211
57	1.19719	9.92804
58	.99839	9.95000
59	.79919	9.96798
60	.59967	9.98198
61	.39992	9.99198
62	.20000	9.99799
63	.00000	10.00000
64	.00000	9.80000
65	.00000	9.60000
66	.00000	9.40000
67	.00000	9.20000
68	.00000	9.00000
69	.00000	8.80000
70	.00000	8.60000
71	.00000	8.40000
72	.00000	8.20000
73	.00000	8.00000
74	.00000	7.40000
75	.00000	6.80000
76	.00000	6.20000
77	.00000	5.60000
78	.00000	5.00000
79	.00000	4.40000
80	.00000	3.80000
81	.00000	3.20000
82	.00000	2.60000
83	.00000	2.00000
84	.00000	1.80000
85	.00000	1.60000
86	.00000	1.40000

```
    87            .00000          1.20000
    88            .00000          1.00000
    89            .00000           .80000
    90            .00000           .60000
    91            .00000           .40000
    92            .00000           .20000
 SURFace SUR1
   TYPE-surface QUAD
   ELEMents-surface
     1    1    2    3
     2    3    4    5
     3    5    6    7
     4    7    8    9
     5    9   10   11
     6   11   12   13
     7   13   14   15
     8   15   16   17
     9   17   18   19
    10   19   20   21
    11   21   22   23
    12   23   24   25
    13   25   26   27
    14   27   28   29
    15   29   30   31
    16   31   32   33
    17   33   34   35
    18   35   36   37
    19   37   38   39
    20   39   40   41
    21   41   42   43
    22   43   44   45
    23   45   46   47
    24   47   48   49
    25   49   50   51
    26   51   52   53
    27   53   54   55
    28   55   56   57
    29   57   58   59
    30   59   60   61
    31   61   62   63
    32   63   64   65
    33   65   66   67
    34   67   68   69
    35   69   70   71
    36   71   72   73
    37   73   74   75
    38   75   76   77
    39   77   78   79
    40   79   80   81
    41   81   82   83
    42   83   84   85
    43   85   86   87
    44   87   88   89
    45   89   90   91
    46   91   92    1
  NORMal +
SAMPling-points POINTS
 101     0.0      0.0
```

```
  102     0.4      0.0
  103     0.8      0.0
  104     1.2      0.0
  105     1.6      0.0
  106     2.0      0.0
  107     3.2      0.0
  108     4.4      0.0


**BCSET
  ID-Bcset  DISP1
  VALUe
  GMR-bc  GMR1
SURFace-bc  SUR1
    ELEMents-bc  32 33 34 35 36 37 38 39 40 41
    ELEMents-bc  42 43 44 45 46
  DISPlacement-bc 1
    SPLIst  ALL
      T-VAlue  1  0.

**BCSET
  ID-Bcset  DISP2
  VALUe
  GMR-bc  GMR1
SURFace-bc  SUR1
    ELEMents-bc   1   2   3   4   5   6   7   8   9   10
    ELEMents-bc  11  12  13  14  15
  DISPlacement-bc 2
    SPLIst  ALL
      T-VAlue  1  0.


**BODY
  CENTrifugal                              $ CENTRIFUGAL LOAD
  TIMEs-centrifugal      1.
  SPEEd-centrifugal       954.92966            $ RPM
$
$  END OF DATA
```

```
      LOAD BALANCE     -0.33333E+07   -0.33334E+07


   JOB TITLE:   THIN SOLID (T300/5208)DISK ROTATING WITH A CONSTANT SPEED
   INTERIOR DISPLACEMENT AT TIME =     0.0000       FOR REGION = GMR1

NODE      X DISPLACEMENT    Y DISPLACEMENT

101      0.000000E+00      0.000000E+00
102      0.116575E+01      0.000000E+00
103      0.232870E+01      0.000000E+00
104      0.348400E+01      0.000000E+00
105      0.462786E+01      0.000000E+00
106      0.575651E+01      0.000000E+00
107      0.901285E+01      0.000000E+00
108      0.119947E+02      0.000000E+00


   JOB TITLE:   THIN SOLID (T300/5208)DISK ROTATING WITH A CONSTANT SPEED
   INTERIOR STRESS AT TIME =     0.0000       FOR REGION = GMR1

NODE      SIGMA-XX          SIGMA-YY          TAU-XY            SIGMA-ZZ

101      0.396968E+06      0.397166E+06      0.000000E+00
102      0.396916E+06      0.397078E+06      0.000000E+00
103      0.394828E+06      0.396159E+06      0.000000E+00
104      0.391728E+06      0.394611E+06      0.000000E+00
105      0.387236E+06      0.392471E+06      0.000000E+00
106      0.381524E+06      0.389841E+06      0.000000E+00
107      0.356719E+06      0.377752E+06      0.000000E+00
108      0.320506E+06      0.360202E+06      0.000000E+00
```

```
EXAMPLE PROBLEM: ELAS608

ANALYSIS TYPE : 2D ANISOTROPIC ELASTOSTATICS

PROBLEM DESCRIPTION :
  AS AN EXTENSION OF THE ORDINARY PLANE STRAIN PROBLEM, THE PRESENT
  CODE ALLOWS FOR PROVIDING NON-ZERO STRAIN IN A DIRECTION
  PERPENDICULAR TO THE (X-Y)PLANE OF 2D BEHAVIOUR. THIS NON-ZERO
  STRAIN CAN AT MOST BE A LINEAR FUNCTION OF X AND Y AND
  SHOULD BE INDEPENDENT OF Z. THIS OPTION WOULD FACILITATE APPROXIMATE
  ANALYSIS OF CERTAIN 3D PROBLEMS. AN ANALYSIS OF THIS KIND IS TERMED
  HERE AS 'GENERALISED PLANE STRAIN'-TYPE.

  A SQUARE PLATE IS SUBJECT TO A CONSTANT STRAIN IN THE Z-DIRECTION.
  THE PLATE IS SUPPORTED ON ROLLERS ON ALL SIDES SO THAT ANY
  DEFORMATION IN THE X-Y PLANE IS CONSTRAINED. CROSS ANISOTROPIC
  MATERIAL PROPERTIES ARE CHOSEN. ANALYTICAL SOLUTIONS TO THIS
  PROBLEM CAN BE FOUND EASILY.

BOUNDARY ELEMENT MODEL :
  4 QUADRATIC ELEMENTS ALONG THE BOUNDARY, ALL SIDES ROLLER SUPPORTED.

REFERENCE FOR ANALYTICAL SOLUTION:
  DEB AND BANERJEE, "2D BEM FOR GENERAL ANISOTROPIC ELASTICITY USING
  PARTICULAR INTEGRALS" (SUBMITTED TO 'JOURNAL OF APPLIED MECHANICS')

SOLUTION POINTS TO VERIFY :

              (STRESS IN THE X-DIRECTION)
  NODE        ANALYTICAL            GPBEST
   1            9713.                9711.

              (STRESS IN THE Y-DIRECTION)
  NODE        ANALYTICAL            GPBEST
   1           10429.               10436.

RUN TIME :
  0.1 X BASE PROBLEM

MISCELLANEOUS:
```

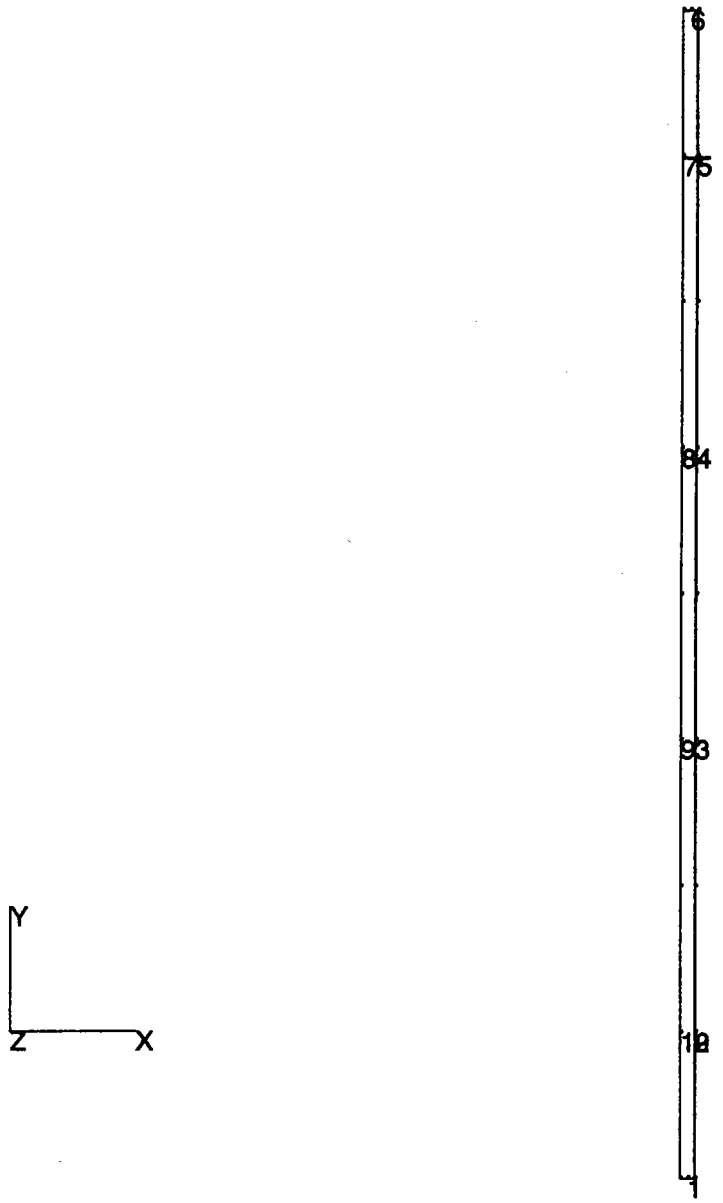

Y
107
106
105
108
104
NODES
O
X
101
102
103

```
**CASE
  TITLe   GENERALISED PLANE STRAIN, SQUARE PLATE
  PLANe STRAIN
  ELAStic              $ PARTICULAR INTEGRALS USED TO MODEL
                       $ NON-ZERO STRAIN THE Z-DIRECTION
  PRINtout-control BOUNDARY


**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  DENSity 1.0
$ NOTE : THE ELASTIC PROPERTIES GIVEN BELOW ARE FOR A CROSS ANISOTROPIC
$ MATERIAL. HOWEVER, CROSS ANISOTROPY IS ONLY A SPECIAL CASE OF
$ ORTHOTROPY.
  ANISotropy ORTHO
  TECHnical  21.E06  1.55E06 1.55E06  0.65E06  0.31  0.31  0.49
$
$ NOTE : THE FOLLOWING 'COMP' OR 'STIF' CARDS (ONLY ONE SET AT A TIME)
$ CAN ALSO BE USED IN LIEU OF THE 'TECH' CARD GIVEN ABOVE TO DESCRIBE
$ ELASTIC MATERIAL PROPERTIES.
$ COMPLIANCES 4.7619E-8  6.4516E-7  6.4516E-7  1.5385E-6
$ COMPLIANCES -1.4762E-8
$ COMPLIANCES -1.4762E-8  -3.1613E-7
$ STIFFNESSES 2.1601E+7  2.0832E+6  2.0832E+6  0.65E6
$ STIFFNESSES 0.9691E+6  0.9691E6  1.0430E6


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration 70.0
  POINts
            101          .0000          .0000
            102          .5000          .0000
            103         1.0000          .0000
            104         1.0000          .5000
            105         1.0000         1.0000
            106          .5000         1.0000
            107          .0000         1.0000
            108          .0000          .5000
  SURFace SUR1
    TYPE-surface QUAD
    ELEMents-surface
       101          101       102       103
       102          103       104       105
       103          105       106       107
       104          107       108       101
   NORMal +
  SAMPling-points POINTS
  1   0.50    0.50
  2   0.25    0.25
```

```
  3   0.25    0.75
  4   0.75    0.25
  5   0.75    0.75
  GENE  0.01   0.  0.


**BCSET
  ID-Bcset  DISP1
  VALUe
  GMR-bc  GMR1
SURFace-bc  SUR1
    ELEMents-bc  102 104
  DISPlacement-bc 1
    SPLIst  ALL
      T-VAlue 1   0.

**BCSET
  ID-Bcset  DISP2
  VALUe
  GMR-bc  GMR1
SURFace-bc  SUR1
    ELEMents-bc  101  103
  DISPlacement-bc 2
    SPLIst  ALL
      T-VAlue 1   0.
$
$  END OF DATA
```

```
 JOB TITLE:    GENERALISED PLANE STRAIN, SQUARE PLATE
 BOUNDARY SOLUTION AT TIME =     0.0000       FOR REGION = GMR1

ELEMENT  NODE NO.      X-DISPL.       Y-DISPL.       X-TRAC.        Y-TRAC.

   101      101       0.00000E+00    0.00000E+00    0.00000E+00   -0.10435E+05
   101      102       0.41654E-10    0.00000E+00    0.00000E+00   -0.10440E+05
   101      103       0.00000E+00    0.00000E+00    0.00000E+00   -0.10435E+05


   102      103       0.00000E+00    0.00000E+00    0.97119E+04    0.00000E+00
   102      104       0.00000E+00   -0.84746E-09    0.97156E+04    0.00000E+00
   102      105       0.00000E+00    0.00000E+00    0.97119E+04    0.00000E+00


   103      105       0.00000E+00    0.00000E+00    0.00000E+00    0.10435E+05
   103      106      -0.56766E-10    0.00000E+00    0.00000E+00    0.10440E+05
   103      107       0.00000E+00    0.00000E+00    0.00000E+00    0.10435E+05


   104      107       0.00000E+00    0.00000E+00   -0.97119E+04    0.00000E+00
   104      108       0.00000E+00   -0.79090E-10   -0.97156E+04    0.00000E+00
   104      101       0.00000E+00    0.00000E+00   -0.97119E+04    0.00000E+00

 JOB TITLE:    GENERALISED PLANE STRAIN, SQUARE PLATE
 INTERIOR DISPLACEMENT AT TIME =     0.0000       FOR REGION = GMR1

NODE      X DISPLACEMENT    Y DISPLACEMENT

  1     -0.435776E-10     -0.497829E-09
  2     -0.196064E-06     -0.610340E-06
  3     -0.169029E-06      0.602126E-06
  4      0.211259E-06     -0.609556E-06
  5      0.184301E-06      0.601449E-06


 JOB TITLE:    GENERALISED PLANE STRAIN, SQUARE PLATE
 INTERIOR STRESS AT TIME =     0.0000       FOR REGION = GMR1

NODE      SIGMA-XX          SIGMA-YY          TAU-XY            SIGMA-ZZ

  1      0.971150E+04      0.104360E+05     -0.171923E-03
  2      0.971269E+04      0.104355E+05      0.149145E+01
  3      0.969950E+04      0.104359E+05      0.649189E+01
  4      0.972436E+04      0.104351E+05     -0.878042E+00
  5      0.971120E+04      0.104355E+05     -0.711148E+01
```

```
EXAMPLE PROBLEM: ELAS609


ANALYSIS TYPE: ELASTIC ANALYSIS
  GENERALIZED AXI-SYMMETRY

PROBLEM DESCRIPTION:
  THIN HOLLOW CYLINDER SUBJECTED TO BENDING

BOUNDARY ELEMENT MODEL:
  SINGLE GMR, TEN SURFACE ELEMENTS.

SOLUTION POINTS TO VERIFY

  POINTS         X-DISPLACEMENT            BENDING THEORY
  300021         3.208E+3                  3.200E+3

MISCELLANEOUS:
```

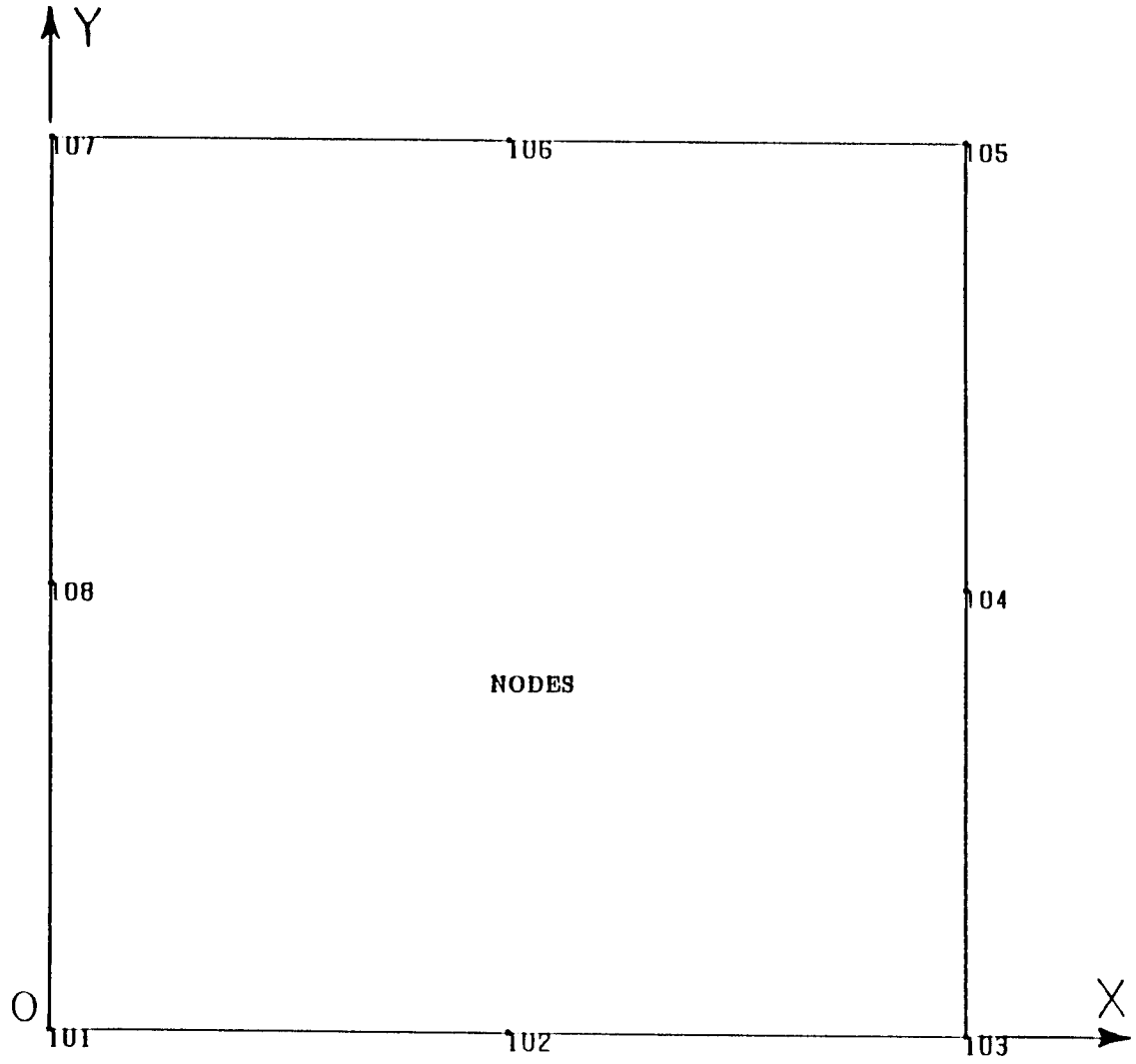

6
75
84
93
12
Y
Z
X

```
**CASE
  TITLe   PURE BEDING OF A CIRCULAR CYLINDER
  AXISymmetry   GENERAL
  PRINtout-control BOUNDARY
  PRINtout-control STRESS
  PRINtout-control LOAD
  NEUTral-file
**MATERIAL
  ID-Material MAT1
    EMODulus 1.
    POISson 0.3
**GMR
  ID-Gmr GMR1
  MATErial MAT1
  POINts
     11    9.5    0.0
     12   10.0    0.0
     13   10.5    0.0
     14   10.5    10.0
     15   10.5    20.0
     16   10.5    30.0
     17   10.5    40.0
     18   10.5    50.0
     19   10.5    60.0
     20   10.5    70.0
     21   10.5    80.0
     22   10.0    80.0
     23    9.5    80.0
     24    9.5    70.0
     25    9.5    60.0
     26    9.5    50.0
     27    9.5    40.0
     28    9.5    30.0
     29    9.5    20.0
     30    9.5    10.0
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
         1    11  12  13
         2    13  14  15
         3    15  16  17
         4    17  18  19
         5    19  20  21
         6    21  22  23
         7    23  24  25
         8    25  26  27
         9    27  28  29
        10    29  30  11
  NORMal  +
**BCSET
  ID-Bcset DISP2
  GMR-bc GMR1
  SURFace-bc SURF1
```

```
    ELEMents-bc 1
      DISPlacement-bc 2
      SPLIst 11 12 13
      T-VAlue 1 0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
      T-VAlue 1 0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
      T-VAlue 1 0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
**BCSET
  ID-Bcset DISP9
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 1
    POINts-bc 12
      DISPlacement-bc 1
      SPLIst 12
T-VAlue 1  1.5E+1   1.38582E+1  1.06066E+1  5.74025 0.
T-VAlue 1 -5.74025 -1.06066E+1 -1.38582E+1 -1.5E+1
**BCSET
  ID-Bcset DISP1
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 1
    POINts-bc   12
      DISPlacement-bc 3
      SPLIst 12
T-VAlue 1  0.     5.74025    1.06066E+1 1.38582E+1
T-VAlue 1  1.5E+1 1.38582E+1 1.06066E+1 5.74025    0.
**BCSET
  ID-Bcset TRAC2
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
      TRACtion-bc 2
      SPLIst  21  22  23
T-VAlue 1  -1.05E+01 -9.700735 -7.424621 -4.018176 0.  4.018176
T-VAlue 1   7.424621  9.700735  1.05E+01
$
T-VAlue 1  -1.00E+01 -9.238795 -7.071068 -3.826834 0.  3.826834
T-VAlue 1   7.071068  9.238795  1.00E+01
$
T-VAlue 1  -9.5      -8.776856 -6.717514 -3.635493 0.  3.635493
T-VAlue 1   6.717514  8.776856  9.5
```

```
          JOB TITLE:    PURE BEDING OF A CIRCULAR CYLINDER
                NODAL OUTPUT AT TIME=  0.00      FOR REGION = GMR1


       NODE       ------- DISPLACEMENT -------    ---------------------------- STRESS/STRAIN --------------------
-------
                    X           Y           Z            X-X         Y-Y         Z-Z         X-Y         X-Z         Y-
Z


      300021    3.208E+03   0.000E+00  -8.379E+02  -3.861E-03  -5.088E-04  -1.048E+01   0.000E+00   0.000E+00   0.000E+00
                                                     3.139E+00   7.981E+01  -1.048E+01   0.000E+00  -7.981E+01   0.000E+00
      310021    3.203E+03   1.167E+01  -7.741E+02  -2.125E-03   6.550E-05  -9.680E+00  -2.084E-03  -8.743E-05   2.111E-
04
                                                     1.243E+01   6.251E+01  -9.678E+00  -2.419E+01  -5.643E+01  -5.643E+01
```

```
VERIFICATION PROBLEM:  ELAS019

ANALYSIS TYPE:
  ELASTICITY,
  2-D 'GENERALIZED' PLANE STRAIN, STATIC, ELASTIC ANALYSIS

PROBLEM DISCRIPTION:
  A UNIT CUBE SUBJECTED TO A CONSTANT STRAIN IN THE THIRD DIRECTION

BOUNDARY ELEMENT MODEL:
  A SQUARE REGION OF FOUR QUADRATIC BOUNDARY ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  3-D HOOKES LAW

SOLUTION POINTS TO VERIFY:

  TIME 1: DISPLACEMENTS ARE LINEAR:

                 X-DISPLACEMENT              Y-DISPLACEMENT
     NODE       ANAL.       GPBEST          ANAL.        GPBEST
      105        -.3        -.30000          -.15        -.15000
        1        -.15       -.15000           .0          .00000

RUN TIME:
    0.05 X BASE PROBLEM

MISCELLANEOUS:
  NONE
```


Y
107
106
105
108
104
NODES
O
X
101
102
103

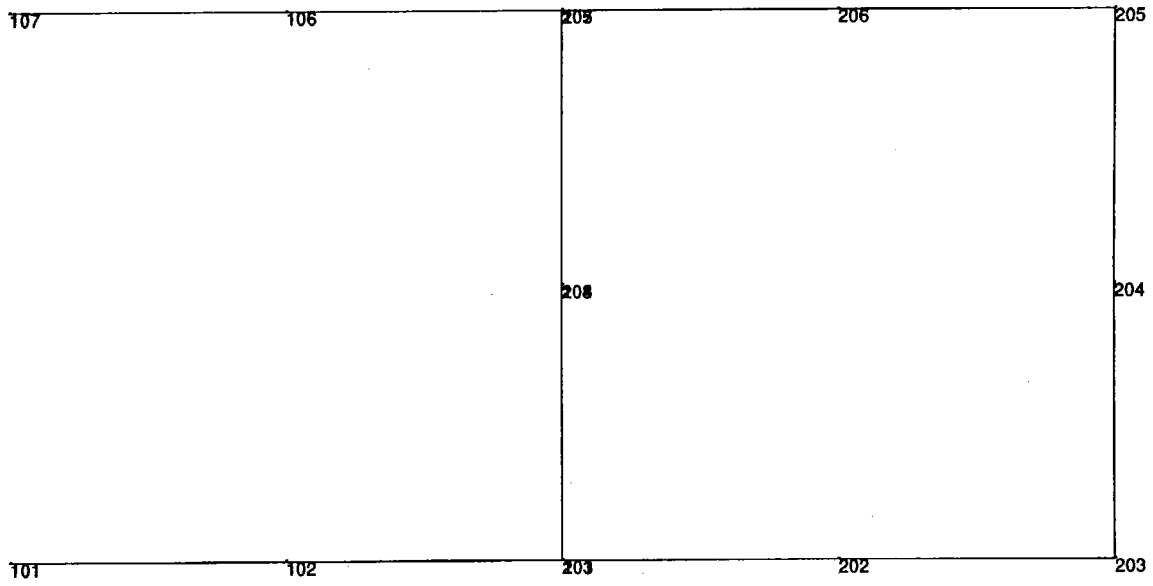
```
**CASE
  TITLe - ELAS019 -  2-D 'GENERALIZED' PLANE STRAIN ANALYSIS OF A CUBE
  PLANe STRAIN
  ECHO-input

**MATERIAL
  ID-Material MAT1
  TEMPeratures 70.0
  EMODulus 100.
  POISson  0.3

**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration 70.0
  POINts
             101          .0000          .0000
             102          .5000          .0000
             103         1.0000          .0000
             104         1.0000          .5000
             105         1.0000         1.0000
             106          .5000         1.0000
             107          .0000         1.0000
             108          .0000          .5000
  SURFace SUR1
    TYPE-surface QUAD
    ELEMents-surface
       101          101       102       103
       102          103       104       105
       103          105       106       107
       104          107       108       101
   NORMal +
  SAMPling-points POINTS
  1   0.50    0.50

$  B.C. TO PREVENT RIGID BODY MOVEMENT

**BCSET
  ID-Bcset  DISP1
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SUR1
    ELEMents-bc  104
  DISPlacement-bc 1
    SPLIst  ALL
      T-VAlue 1   0.

**BCSET
  ID-Bcset  DISP2
  VALUe
  GMR-bc  GMR1
  SURFace-bc  SUR1
    ELEMents-bc  102 104
    POINts-bc 104 108
```

```
  DISPlacement-bc 2
    SPLIst  ALL
      T-VAlue 1   0.

$  CONSTANT STRAIN IN THE THIRD DIRECTION INPUT AS A BODY FORCE

**BODY
  GENEralized
  TIMEs-generalized  1.0
  GMR-generalized GMR1
    CONStant  1.0

$ END OF DATA
```

```
JOB TITLE:  - ELAS019 -  2-D 'GENERALIZED' PLANE STRAIN ANALYSIS OF A CUBE
BOUNDARY SOLUTION AT TIME =      1.000000         FOR REGION = GMR1
```

ELEMENT	NODE NO.	X DISPL	Y DISPL	X TRACTION	Y TRACTION
101	101	.00000E+00	.15000E+00	.00000E+00	.00000E+00
101	102	-.15000E+00	.15000E+00	.00000E+00	.00000E+00
101	103	-.30000E+00	.15000E+00	.00000E+00	.00000E+00
102	103	-.30000E+00	.15000E+00	.00000E+00	.00000E+00
102	104	-.30000E+00	.00000E+00	.00000E+00	-.16922E-08
102	105	-.30000E+00	-.15000E+00	.00000E+00	.00000E+00
103	105	-.30000E+00	-.15000E+00	.00000E+00	.00000E+00
103	106	-.15000E+00	-.15000E+00	.00000E+00	.00000E+00
103	107	.00000E+00	-.15000E+00	.00000E+00	.00000E+00
104	107	.00000E+00	-.15000E+00	.13735E-06	.00000E+00
104	108	.00000E+00	.00000E+00	.13156E-06	-.96228E-06
104	101	.00000E+00	.15000E+00	-.58661E-06	.00000E+00

```
EXAMPLE PROBLEM:  ELAS046


ANALYSIS TYPE:
  THERMOELASTICITY (VIA CONTINUOUS FUNCTION),
  AXISYMMETRY, STATIC, ELASTIC ANALYSIS WITH THERMAL BODY FORCE

PROBLEM DISCRIPTION:
  TWO REGION,  EXPANSION OF A CYLINDRICAL ROD SUBJECTED TO A
  UNIFORM, LINEAR,  RADIAL, ETC. TEMPERATURE DISTRIBUTION
  QUARTIC ELEMENTS

BOUNDARY ELEMENT MODEL:
  TWO SQUARE REGIONS, WITH QUARTIC BOUNDARY 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: UNIFORM TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE LINEAR,
          AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

                 R-DISPLACEMENT                  RR-STRESS
     NODE       ANAL.       GPBEST           ANAL.         GPEST
      104        1.0         .9999            .0          .00635
      204        2.0        1.9998            .0          .00000
      210        1.25       1.2499            .0         -.00738


  TIME 2: LINEAR TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE QUADRATIC
          AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

                 R-DISPLACEMENT                  RR-STRESS
     NODE       ANAL.       GPBEST           ANAL.         GPEST
      104        0.5         .4999            .0          .00317
      204        1.0         .9999            .0          .00000
      210         .625       .6249            .0         -.00369


  TIME 3: RADIAL TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE QUADRATIC
          AND STRESSES ARE NOT ZERO !

                 R-DISPLACEMENT                  RR-STRESS
     NODE       ANAL.       GPBEST           ANAL.         GPEST


MISCELLANEOUS:
```


107
106
205
206
205
204
204
101
102
203
202
203

```
   **CASE
     TITLe  - ELAS046 -  THERMELASTIC, AXISYMMETRIC ROD
     TIMEs-output 1.  2.  3.  4.  5.  6.
     AXISymmetry
     ECHO-input

   **MATERIAL
     ID-Material MAT1
     TEMPeratures 70.
     EMODulus 100.
     POISson 0.3
     ALPHa  1.

   **GMR
     ID-Gmr GMR1
     TINTegration 70. $ TEMP. AT WHICH MATERIAL CONSTANTS ARE EVALUATED FOR
INTEGRATION
     TREFerence 0.0 $ CHANGE IN TEMPERATURE DATUM
     MATErial MAT1
     POINts
       101  0.0  0.0
       102  0.5  0.0
       103  1.0  0.0
       104  1.0  0.5
       105  1.0  1.0
       106  0.5  1.0
       107  0.0  1.0
     SURFace SURF1
       TYPE-surface QUAR
       ELEMents-surface
         101   101 102 103
         102   103 104 105
         103   105 106 107
       NORMal +
     SAMPling-points POINTS
       109   .5  .5

   **GMR
     ID-Gmr GMR2
     TINTegration 70. $ TEMP. AT WHICH MATERIAL CONSTANTS ARE EVALUATED FOR
INTEGRATION
     TREFerence 0.0 $ CHANGE IN TEMPERATURE DATUM
     MATErial MAT1
     POINts
       201  1.0  0.0
       202  1.5  0.0
       203  2.0  0.0
       204  2.0  0.5
       205  2.0  1.0
       206  1.5  1.0
       207  1.0  1.0
       208  1.0  0.5
     SURFace SURF2
```

```
      TYPE QUAR
      ELEMents
        201   201 202 203
        202   203 204 205
        203   205 206 207
        204   207 208 201
      NORMal +
    SAMPling-points POINTS
     209   1.5  .5
     210   1.25 .5

  $  DEFINE INTERFACE CONNECTION

  **INTERFACE
    GMR-1 GMR1
    SURFace-gmr-1 SURF1
      ELEMents-gmr-1  102
    GMR-2 GMR2
    SURFace-gmr-2 SURF2
      ELEMents-gmr-2  204

  $  FIX BODY TO PREVENT RIGID BODY DISPLACEMENT BUT ALLOW FREE EXPANSION

  **BCSET
    ID-Bcset DISP3
    GMR-bc GMR2
    SURFace-bc SURF2
      ELEMents-bc  202
      POINT 204
      DISPlacement-bc 2
        SPLIst 204
        T-VAlue 1 0.0

  $ DEFINE TEMPERATURE DISTRIBUTION (IN TIME)

  **BODY
    THERmal
    TIMEs-thermal 1.  2.  3.  4.  5.  6.
    GMR-thermal GMR1
    TEMPeratures-thermal CONT
     CONS    1.   0.   0.   0.   0.   0.
     YVAL    0.   1.   0.   0.   0.   0.
     XVAL    0.   0.   1.   0.   0.   0.
     XXVAL   0.   0.   0.   1.   0.   0.
     YYVAL   0.   0.   0.   0.   1.   0.
     XYVAL   0.   0.   0.   0.   0.   1.
    GMR-thermal GMR2
    TEMPeratures-thermal CONT
     CONS    1.   0.   0.   0.   0.   0.
     YVAL    0.   1.   0.   0.   0.   0.
     XVAL    0.   0.   1.   0.   0.   0.
     XXVAL   0.   0.   0.   1.   0.   0.
     YYVAL   0.   0.   0.   0.   1.   0.
     XYVAL   0.   0.   0.   0.   0.   1.

  $ END OF DATA
```

```
          JOB TITLE:   - ELAS046 -  THERMELASTIC, AXISYMMETRIC ROD
          BOUNDARY SOLUTION AT TIME =      1.000000         FOR REGION = GMR1
```

ELEMENT	NODE NO.	R DISPL	Z DISPL	R TRACTION	Z TRACTION
101	101	.46492E-05	-.49998E+00	.00000E+00	.00000E+00
101	102	.50000E+00	-.49999E+00	.00000E+00	.00000E+00
101	103	.10000E+01	-.49999E+00	.00000E+00	.00000E+00
101	200001	.25000E+00	-.49999E+00	.00000E+00	.00000E+00
101	200002	.75000E+00	-.49999E+00	.00000E+00	.00000E+00
102	103	.10000E+01	-.49999E+00	.61072E-03	-.91918E-03
102	104	.10000E+01	.77391E-09	-.32503E-04	-.11456E-04
102	105	.10000E+01	.49999E+00	.56262E-03	.88221E-03
102	200003	.10000E+01	-.25000E+00	.28390E-03	-.52073E-04
102	200004	.10000E+01	.25000E+00	.30523E-03	.64174E-04
103	105	.10000E+01	.49999E+00	.00000E+00	.00000E+00
103	106	.50000E+00	.49999E+00	.00000E+00	.00000E+00
103	107	.46492E-05	.49998E+00	.00000E+00	.00000E+00
103	200005	.75000E+00	.49999E+00	.00000E+00	.00000E+00
103	200006	.25000E+00	.49999E+00	.00000E+00	.00000E+00

6.3.5 FORCED VIBRATION EXAMPLES

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 GEOMATRY AND LOAD, ONLY ONE QUARTER OF THE GEOMETRY IS MODELLED. 4 ELEMENTS ARE USED TO DISCRETIZE THE SOIL-FOUNDATION INTERFACE WHILE THE FREE-SURFACE IS MODELLED WITH 6 ELEMENTS.

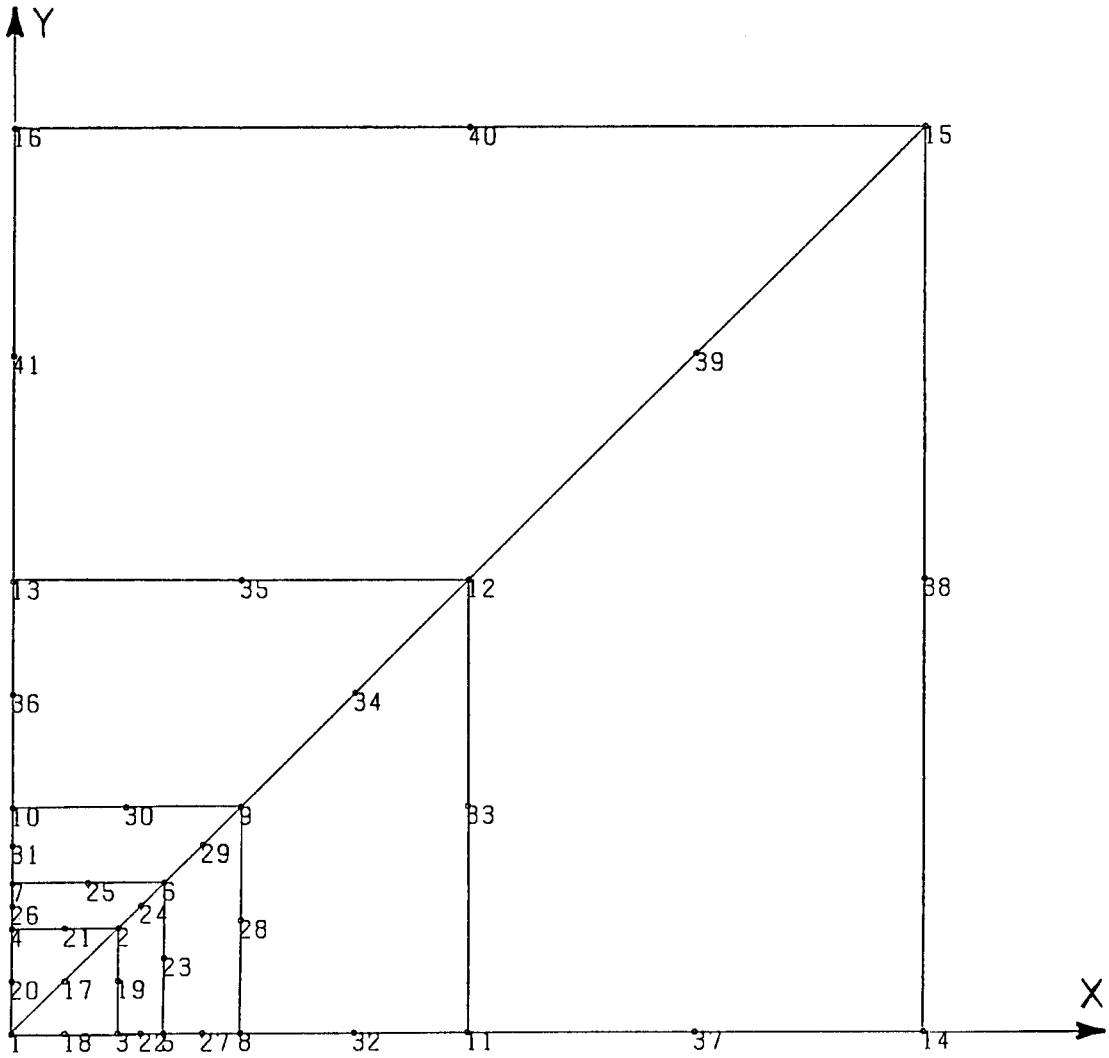
THE FOUNDATION IS SUBJECTED TO UNIT VERTICAL DISPLACEMENT (DIRECTION 3) WHILE TRACTIONS ON THE OTHER TWO DIRECTIONS ARE ASSUMED TO BE ZERO. THIS CORRESPONDS TO THE SO-CALLED 'RELAXED' CONDITION. THE FREE-SURFACE IS ASSUMED TO BE TRACTION FREE.

SOLUTION POINTS TO VERIFY:

EXCITING CIRCULAR FREQUENCY	ELEMENT	VERTICAL LOAD (GPBEST) (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:


Y
X
16
40
15
41
39
13
35
12
38
36
34
10
30
9
33
31
29
7
25
6
26
24
4
21
2
28
23
20
17
19
1
18
3
22
5
27
8
32
11
37
14

NODES

```
**CASE
  TITLe     SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
   SYMMetry QUAD
   FORCed-vibration   .001   1.5
   PRINtout-control LOAD


**MATERIAL
    ID-Material  MAT1
    TEMPeratures 70.
    EMODulus 2.66666
    POISson 0.33333
    DENSity 1.0
    DAMPing 0.0


**GMR
    ID-Gmr GMR1
    MATErial MAT1
    TREFerence 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
        31       0.000000       1.250000       0.000000
        32       2.250000       0.000000       0.000000
        33       3.000000       1.500000       0.000000
        34       2.250000       2.250000       0.000000
        35       1.500000       3.000000       0.000000
        36       0.000000       2.250000       0.000000
        37       4.500000       0.000000       0.000000
        38       6.000000       3.000000       0.000000
        39       4.500000       4.500000       0.000000
        40       3.000000       6.000000       0.000000
        41       0.000000       4.500000       0.000000
  SURFace SURF
    TYPE-surface QUAD
    ELEMents-surface
            1     3    19     2    17     1    18
            2     2    21     4    20     1    17
            3     3    22     5    23     6    24     2    19
            4     2    24     6    25     7    26     4    21
            5     5    27     8    28     9    29     6    23
            6     6    29     9    30    10    31     7    25
            7     8    32    11    33    12    34     9    28
            8     9    34    12    35    13    36    10    30
            9    11    37    14    38    15    39    12    33
           10    12    39    15    40    16    41    13    35
   NORMal  1 +


**BCSET
   ID-Bcset DISP3
    VALUe
    GMR-bc GMR1
    SURFace-bc SURF
    ELEMents-bc   1  2  3  4
    DISPlacement-bc  3
    SPLIst  ALL
    T-VAlue 1    1.
$
$ END OF DATA
```

```
JOB TITLE:      SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
 LOAD CALCULATION AT EXCITING FREQUENCY =      0.001000

 LOADS FOR REGION GMR1
 ELEMENT             X                      Y                      Z
    1       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.3553E+00  0.2878E-03
    2       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.3553E+00  0.2878E-03
    3       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.5434E+00  0.4398E-03
    4       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.5434E+00  0.4398E-03
    5       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    6       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    7       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    8       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    9       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
   10       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00


JOB TITLE:      SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
 LOAD CALCULATION AT EXCITING FREQUENCY =      1.500000

 LOADS FOR REGION GMR1
 ELEMENT             X                      Y                      Z
    1       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.1242E+00  0.5871E+00
    2       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.1242E+00  0.5871E+00
    3       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.4411E+00  0.8158E+00
    4       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.4411E+00  0.8159E+00
    5       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    6       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    7       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    8       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
    9       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
   10       0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

EXAMPLE PROBLEM: FORC602

ANALYSIS TYPE: DYNAMIC ANALYSIS
2-D, STEADY-STATE

PROBLEM DESCRIPTION:
DYNAMIC STIFFNESS OF RIGID EMBEDDED STRIP FOOTING BONDED TO ELASTIC HALF-SPACE (SOIL).

BOUNDARY ELEMENT MODEL:
BECAUSE OF HALF SYMMETRY IN GEOMATRY AND LOAD, ONLY ONE HALF OF THE GEOMETRY IS MODELLED. 5 ELEMENTS ARE USED TO DISCRETIZE THE SOIL-FOUNDATION INTERFACE WHILE THE FREE-SURFACE IS MODELLED WITH 8 ELEMENTS. IN ADDITION, THE REGION IS ENCLOSED WITH 4 'ENCLOSING ELEMENTS'.

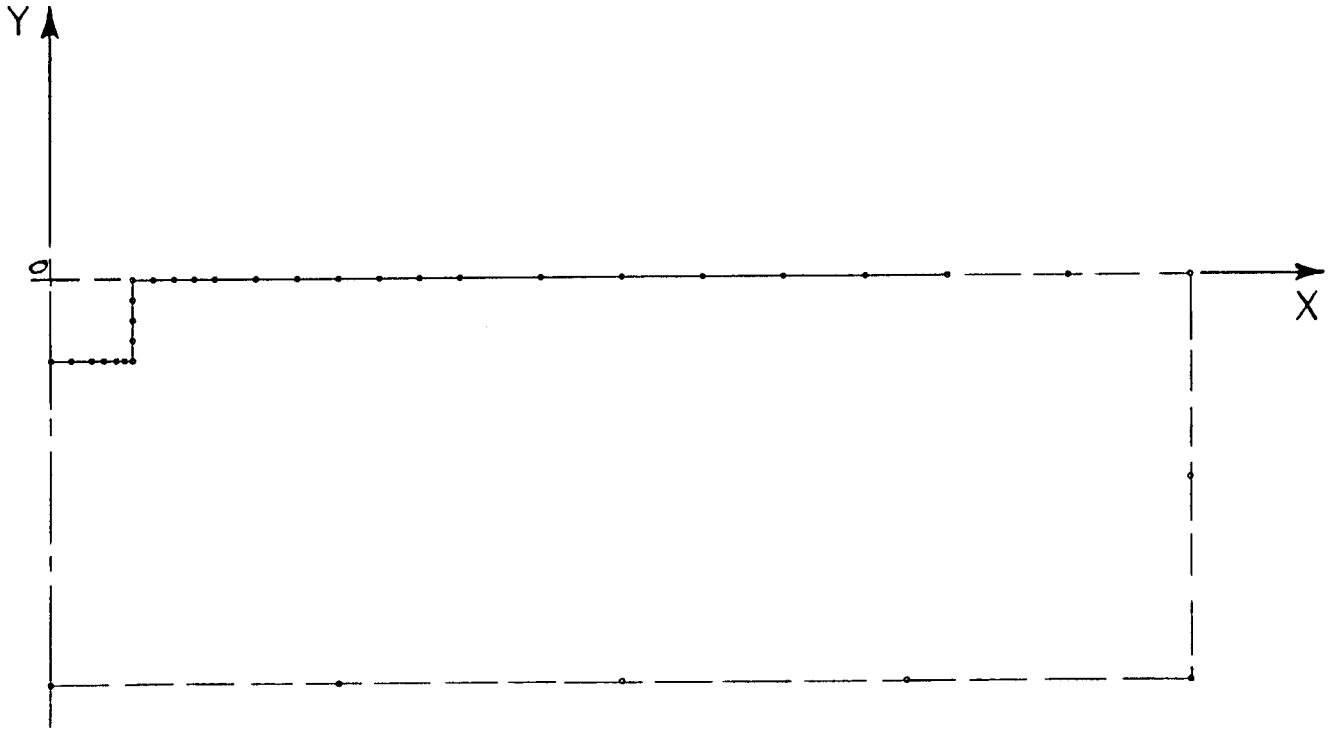
THE FOUNDATION IS SUBJECTED TO UNIT VERTICAL DISPLACEMENT (DIRECTION 2) WHILE DISPLACEMENT IN THE OTHER DIRECTION IS ASSUMED TO BE ZERO. THIS CORRESPONDS TO THE 'BONDED' CONTACT. THE FREE-SURFACE IS ASSUMED TO BE TRACTION-FREE.

SOLUTIONS POINTS TO VERIFY:

EXCITING CIRCULAR FREQUENCY	ELEMENT	VERTICAL LOAD (GPBEST) (DIRECTION 2) REAL PART	COMPLEX PART
0.001	9	-0.4387E-01	-0.1790E-01
0.001	13	-0.8448E-01	-0.2465E-01
1.5	9	-0.4671E-01	-0.5642E+00
1.5	13	0.1752E+00	-0.1233E+01

RUN TIME:
3 X BASE PROBLEM

MISCELLANEOUS:



GEOMETRY

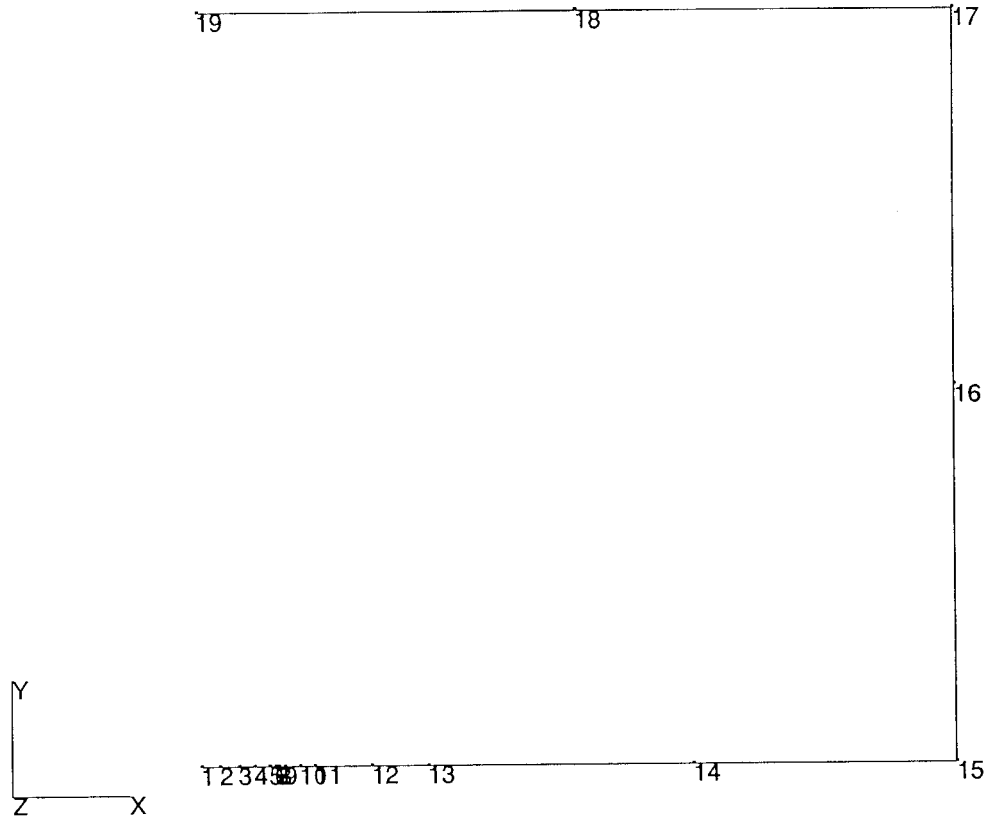
```
**CASE
  TITLe    EMBEDDED STRIP FOOTING
   PLANe STRAIN
   SYMMetry HALF
   FORCed-vibration   .001   0.9   1.5
   PRINtout-control LOAD


**MATERIAL
    ID-Material  MAT1
    TEMPeratures 70.
    EMODulus 2.66666
    POISson 0.33333
    DENSity 1.0
    DAMPing 0.05


**GMR
    ID-Gmr GMR1
    MATErial MAT1
    TREFerence 70.
    POINts
      1      11.0        0.
      2      10.0        0.
      3       9.0        0.
      4       8.0        0.
      5       7.0        0.
      6       6.0        0.
      7       5.0        0.
      8       4.5        0.
      9       4.0        0.
     10       3.5        0.
     11       3.0        0.
     12       2.5        0.
     13       2.0        0.
     14       1.75       0.
     15       1.5        0.
     16       1.25       0.
     17       1.00       0.
     18       1.00      -0.25
     19       1.00      -0.5
     20       1.00      -0.75
     21       1.00      -1.00
     22       0.90      -1.00
     23       0.80      -1.00
     24       0.65      -1.00
     25       0.50      -1.00
     26       0.25      -1.00
     27       0.00      -1.00
    101       0.0       -5.0
    102       3.5       -5.0
    103       7.0       -5.0
    104      10.5       -5.0
```

```
      105      14.0       -5.0
      106      14.0       -2.5
      107      14.0        0.0
      108      12.5        0.0
   SURFace SURF
      TYPE-surface QUAD
      ELEMents-surface
           1        1      2      3
           2        3      4      5
           3        5      6      7
           4        7      8      9
           5        9     10     11
           6       11     12     13
           7       13     14     15
           8       15     16     17
           9       17     18     19
          10       19     20     21
          11       21     22     23
          12       23     24     25
          13       25     26     27
    NORMal +
    ENCLosing-elements
        101      101    102    103
        101      103    104    105
        101      105    106    107
        101      107    108      1


 **BCSET
    ID-Bcset DISP2
     VALUe
     GMR-bc GMR1
     SURFace-bc SURF
     ELEMents-bc   9  10  11  12  13
     DISPlacement-bc  2
     SPLIst  ALL
     T-VAlue 1   -1.0

 **BCSET
    ID-Bcset DISP1
     VALUe
     GMR-bc GMR1
     SURFace-bc SURF
     ELEMents-bc   9  10  11  12  13
     DISPlacement-bc  1
     SPLIst  ALL
     T-VAlue 1   0.0
 $
 $ END OF DATA
```

```
JOB TITLE:     EMBEDDED STRIP FOOTING
     LOAD CALCULATION AT EXCITING FREQUENCY =      0.00100

     LOADS FOR REGION GMR1
ELEMENT            X                      Y
 1      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 2      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 3      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 4      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 5      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 6      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 7      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00



 8      0.00000E+00  0.00000E+00   0.00000E+00  0.00000E+00
 9      0.11804E+00  0.30636E-01  -0.43874E-01 -0.17897E-01
10      0.21062E-01  0.41851E-02  -0.83191E-01 -0.26865E-01
11      0.14095E-01 -0.23720E-02  -0.51458E-01 -0.16420E-01
12      0.10145E-01  0.11675E-02  -0.60279E-01 -0.17570E-01
13      0.15487E-02  0.98650E-04  -0.84475E-01 -0.24653E-01
```

6.3.6 FRACTURE MECHANICS EXAMPLES

```
EXAMPLE PROBLEM: FRACE601

ANALYSIS TYPE: LINEAR ELASTIC FRACTURE MECHANICS
  2-D ANALYSIS

PROBLEM DESCRIPTION:
  RECTANGULAR PLATE WITH CENTER CRACK - SYMMETRY ABOUT X-AXIS
  PLANE STRESS. HALF CRACK LENGTH = 1.0

BOUNDARY ELEMENT MODEL:
  SINGLE GMR WITH 9 ELEMENTS, 19 SOURCE POINTS

REFERENCE FOR ANALYTICAL SOLUTION:
MECHANICS OF BRITTLE FRACTURE - CHEREPANOV

SOLUTION POINTS TO VERIFY:

  GMR     NODE                 STRESS INTESITY FACTORS
                              ANALYTICAL            GPBEST
                              MODE-I  MODE-II     MODE-I  MODE-II

  GMR1     6                  178.3     0.0       177.82    0.0

RUN TIME:
 0.57 X BASE PROBLEM


MISCELLANEOUS:  NONE
```



NODES

```
**CASE
  TITLe  CENTER CRACK IN INFINITE DOMAIN - 1 SYMMETRY
  PLANe STRESS
  FRACture
  TIMEs-output 1.
  SYMMetry HALF

**MATERIAL
  ID-Material MAT1
    TEMPeratures 70.
    EMODulus 1000.0
    POISson 0.25

**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TREFerence 70.
  TINTegration 0.
  POINts
      1     0.0    0.0
      2     0.25   0.0
      3     0.5    0.0
      4     0.7    0.0
      5     0.9    0.0
      6     0.975  0.0
      7     1.0    0.0
      8     1.025  0.0
      9     1.1    0.0
     10     1.3    0.0
     11     1.5    0.0
     12     2.25   0.0
     13     3.0    0.0
     14     6.5    0.0
     15    10.0    0.0
     16    10.0    5.0
     17    10.0   10.0
     18     5.0   10.0
     19     0.0   10.0
  SURFace SEG1
    TYPE-surface QUAD
    ELEMents-surface
       1  1   2  3
       2  3   4  5
    D  3  5   6  7
    T  4  7   8  9
       5  9  10 11
       6  11 12 13
       7  13 14 15
       8  15 16 17
       9  17 18 19
NORMal    +
  CRACk 7
```

```
**BCSET
  ID-Bcset DISP2
  VALUe
  GMR-bc GMR1
  SURFace-bc SEG1
    ELEMents-bc 4 5 6 7
    DISPlacement-bc 2
      SPLIst ALL
      T-VAlue 1  0.0
**BCSET
  ID-Bcset TRAC2
  VALUe
  GMR-bc GMR1
  SURFace-bc SEG1
    ELEMents-bc  9
     TRACtion-bc 2
      SPLIst ALL
      T-VAlue 1  100.0
$
$ END OF DATA
```

```
          JOB TITLE:   CENTER CRACK IN INFINITE DOMAIN - 1 SYMMETRY
          BOUNDARY SOLUTION AT TIME =      1.000000         FOR REGION = GMR1


ELEMENT  NODE NO.    X DISPL      Y DISPL    X TRACTION   Y TRACTION

    1       1     0.12012E-16  0.20239E+00  0.00000E+00  0.00000E+00
    1       2    -0.25487E-01  0.19591E+00  0.00000E+00  0.00000E+00
    1       3    -0.50596E-01  0.17534E+00  0.00000E+00  0.00000E+00


    2       3    -0.50596E-01  0.17534E+00  0.00000E+00  0.00000E+00
    2       4    -0.71831E-01  0.14439E+00  0.00000E+00  0.00000E+00
    2       5    -0.90948E-01  0.87947E-01  0.00000E+00  0.00000E+00


    3       5    -0.90948E-01  0.87947E-01  0.00000E+00  0.00000E+00
    3       6    -0.98820E-01  0.44865E-01  0.00000E+00  0.00000E+00
    3       7    -0.10121E+00  0.00000E+00  0.00000E+00  0.00000E+00



          JOB TITLE:   CENTER CRACK IN INFINITE DOMAIN - 1 SYMMETRY
          STRESS INTENSITY FACTORS AT TIME =   1.00000000       FOR REGION = GMR1


          NODE        MODE-I          MODE-II
         ----------------------------------------


           6     0.17781561E+03  0.00000000E+00
```

EXAMPLE PROBLEM: FRAC602

ANALYSIS TYPE: LINEAR ELASTIC FRACTURE MECHANICS
3-D ANALYSIS

PROBLEM DESCRIPTION:
CIRCULAR CRACK BURIED IN A CYLINDER
RADIUS OF CRACK = 1.0, OUTER RADIUS OF CYLINDER = 10.0
HALF LENGTH OF CYLINDER = 10.0

BOUNDARY ELEMENT MODEL:
66 QUADRATIC ELEMENTS AND 184 NODES

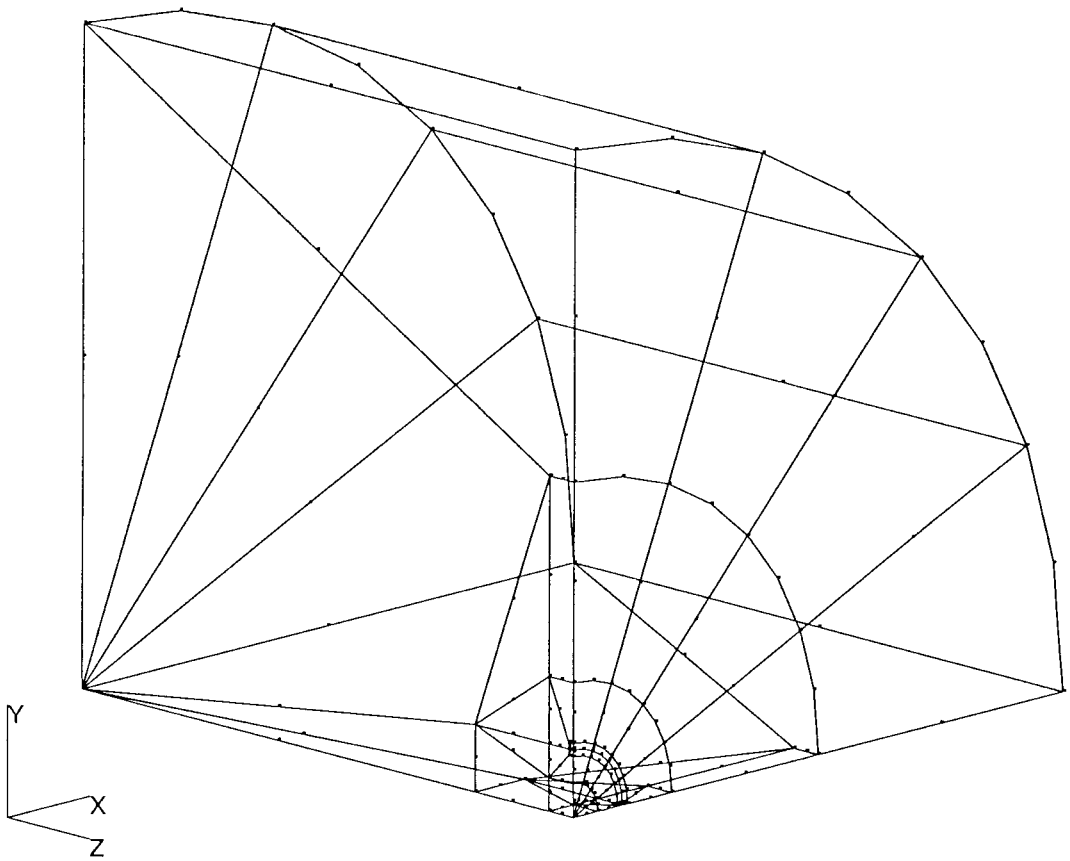
REFERENCE FOR ANALYTICAL SOLUTION:
MECHANICS OF BRITTLE FRACTURE - CHEREPANOV

SOLUTION POINTS TO VERIFY: ANALYTICAL SOLUTION GIVEN IS FOR FINITE BODY. FOR INFINITE BODIES, THE VALUE IS SLIGHTLY INCREASED.

GMR	NODE	STRESS INTESITY FACTORS ANALYTICAL MODE-I	ANALYTICAL MODE-II	GPBEST MODE-I	GPBEST MODE-II
GMR1	6	1000.0	0.0	1010.0	0.0
GMR1	206	1000.0	0.0	1028.0	0.0
GMR1	406	1000.0	0.0	1018.0	0.0
GMR1	606	1000.0	0.0	1035.0	0.0
GMR1	806	1000.0	0.0	1009.0	0.0

RUN TIME:
86 X BASE PROBLEM

MISCELLANEOUS: NONE


Y
X
Z

```
**CASE
  TITLe BURIED CIRCULAR CRACK - NO SYMMETRY - a/w=0.1
  FRACture
  TIMEs-output 1.0

**MATERIAL
  ID-Material MAT1
  EMODulus 2.0E05
  POISson  0.3
  DENSity 0.873

**GMR
  ID-Gmr REG1
  MATErial MAT1
  POINts
         1        .000000        .000000        .000000
         2        .250000        .000000        .000000
       202        .230970        .095671        .000000
       402        .176777        .176777        .000000
       602        .095671        .230970        .000000
       802        .000000        .250000        .000000
         3        .500000        .000000        .000000
       103        .490393        .097545        .000000
       203        .461940        .191342        .000000
       303        .415735        .277785        .000000
       403        .353553        .353553        .000000
       503        .277785        .415735        .000000
       603        .191342        .461940        .000000
       703        .097545        .490393        .000000
       803        .000000        .500000        .000000
         4        .700000        .000000        .000000
       204        .646716        .267878        .000000
       404        .494975        .494975        .000000
       604        .267878        .646716        .000000
       804        .000000        .700000        .000000
         5        .900000        .000000        .000000
       105        .882707        .175581        .000000
       205        .831492        .344415        .000000
       305        .748323        .500013        .000000
       405        .636396        .636396        .000000
       505        .500013        .748323        .000000
       605        .344415        .831492        .000000
       705        .175581        .882707        .000000
       805        .000000        .900000        .000000
         6        .975000        .000000        .000000
       206        .900782        .373116        .000000
       406        .689429        .689429        .000000
       606        .373116        .900782        .000000
       806        .000000        .975000        .000000
         7       1.000000        .000000        .000000
       107        .980785        .195090        .000000
       207        .923880        .382683        .000000
```

307	.831470	.555570	.000000
407	.707107	.707107	.000000
507	.555570	.831470	.000000
607	.382683	.923880	.000000
707	.195090	.980785	.000000
807	.000000	1.000000	.000000
8	1.025000	.000000	.000000
208	.946976	.392250	.000000
408	.724784	.724784	.000000
608	.392250	.946976	.000000
808	.000000	1.025000	.000000
9	1.100000	.000000	.000000
109	1.078864	.214599	.000000
209	1.016267	.420952	.000000
309	.914617	.611127	.000000
409	.777817	.777817	.000000
509	.611127	.914617	.000000
609	.420952	1.016267	.000000
709	.214599	1.078864	.000000
809	.000000	1.100000	.000000
20	.900000	.000000	-.050000
820	.000000	.900000	-.050000
21	1.000000	.000000	-.050000
821	.000000	1.000000	-.050000
22	1.100000	.000000	-.050000
822	.000000	1.100000	-.050000
30	.900000	.000000	-.100000
830	.000000	.900000	-.100000
31	.950000	.000000	-.100000
831	.000000	.950000	-.100000
32	1.000000	.000000	-.100000
832	.000000	1.000000	-.100000
33	1.050000	.000000	-.100000
833	.000000	1.050000	-.100000
34	1.100000	.000000	-.100000
834	.000000	1.100000	-.100000
40	.000000	.000000	-.250000
41	.500000	.000000	-.250000
841	.000000	.500000	-.250000
42	.700000	.000000	-.300000
842	.000000	.700000	-.300000
46	1.000000	.000000	-.300000
846	.000000	1.000000	-.300000
50	.000000	.000000	-.500000
51	.250000	.000000	-.500000
851	.000000	.250000	-.500000
52	.500000	.000000	-.500000
852	.000000	.500000	-.500000
53	.750000	.000000	-.500000
853	.000000	.750000	-.500000
54	1.000000	.000000	-.500000
854	.000000	1.000000	-.500000
60	.000000	.000000	-1.250000
61	.750000	.000000	-1.250000
861	.000000	.750000	-1.250000
62	1.000000	.000000	-1.250000
862	.000000	1.000000	-1.250000
70	.000000	.000000	-2.000000

71	.500000	.000000	-2.000000
871	.000000	.500000	-2.000000
72	1.000000	.000000	-2.000000
872	.000000	1.000000	-2.000000
80	.000000	.000000	-6.000000
81	.500000	.000000	-6.000000
881	.000000	.500000	-6.000000
90	.000000	.000000	-10.000000
10	1.550000	.000000	.000000
210	1.432013	.593159	.000000
410	1.096016	1.096016	.000000
610	.593159	1.432013	.000000
810	.000000	1.550000	.000000
43	1.550000	.000000	-.300000
843	.000000	1.550000	-.300000
11	2.000000	.000000	.000000
111	1.961571	.390181	.000000
211	1.847759	.765367	.000000
311	1.662939	1.111140	.000000
411	1.414213	1.414213	.000000
511	1.111140	1.662940	.000000
611	.765367	1.847759	.000000
711	.390180	1.961571	.000000
811	.000000	2.000000	.000000
44	2.000000	.000000	-.250000
844	.000000	2.000000	-.250000
56	2.000000	.000000	-.500000
856	.000000	2.000000	-.500000
64	3.000000	.000000	-1.250000
864	.000000	3.000000	-1.250000
12	3.500000	.000000	.000000
212	3.233578	1.339392	.000000
412	2.474874	2.474874	.000000
612	1.339392	3.233578	.000000
812	.000000	3.500000	.000000
57	3.500000	.000000	-.500000
857	.000000	3.500000	-.500000
13	5.000000	.000000	.000000
113	4.903927	.975453	.000000
213	4.619398	1.913417	.000000
313	4.157347	2.777850	.000000
413	3.535535	3.535535	.000000
513	2.777850	4.157347	.000000
613	1.913417	4.619398	.000000
713	.975450	4.903927	.000000
813	.000000	5.000000	.000000
45	5.000000	.000000	-.250000
845	.000000	5.000000	-.250000
58	5.000000	.000000	-.500000
858	.000000	5.000000	-.500000
14	7.500000	.000000	.000000
214	6.929095	2.870126	.000000
414	5.303301	5.303301	.000000
614	2.870126	6.929095	.000000
814	.000000	7.500000	.000000
82	7.500000	.000000	-5.250000
882	.000000	7.500000	-5.250000
15	10.000000	.000000	.000000

```
      115        9.807852        1.950904         .000000
      215        9.238796        3.826834         .000000
      315        8.314696        5.555702         .000000
      415        7.071068        7.071068         .000000
      515        5.555702        8.314696         .000000
      615        3.826834        9.238796         .000000
      715        1.950902        9.807852         .000000
      815         .000000       10.000000         .000000
       83       10.000000         .000000       -5.000000
      283        9.238796        3.826834       -5.000000
      483        7.071068        7.071068       -5.000000
      683        3.826834        9.238796       -5.000000
      883         .000000       10.000000       -5.000000
       92       10.000000         .000000      -10.000000
      192        9.807852        1.950904      -10.000000
      292        9.238796        3.826834      -10.000000
      392        8.314696        5.555702      -10.000000
      492        7.071068        7.071068      -10.000000
      592        5.555702        8.314696      -10.000000
      692        3.826834        9.238796      -10.000000
      792        1.950902        9.807852      -10.000000
      892         .000000       10.000000      -10.000000
       55        1.500000         .000000        -.500000
      855         .000000        1.500000        -.500000
       63        1.500000         .000000       -1.250000
      863         .000000        1.500000       -1.250000
       91        5.000000         .000000      -10.000000
      291        4.619398        1.913400      -10.000000
      491        3.535534        3.535534      -10.000000
      691        1.913400        4.619398      -10.000000
      891         .000000        5.000000      -10.000000
SURFace SURF1
  TYPE-surface QUAD
  ELEMents-surface
     1    1    2    3  103  203  202
    11    1  202  203  303  403  402
    21    1  402  403  503  603  602
    31    1  602  603  703  803  802
     2    3    4    5  105  205  204  203  103
    12  203  204  205  305  405  404  403  303
    22  403  404  405  505  605  604  603  503
    32  603  604  605  705  805  804  803  703
D    3    7  107  207  206  205  105    5    6
D   13  205  305  405  406  407  307  207  206
D   23  407  507  607  606  605  505  405  406
D   33  607  707  807  806  805  705  605  606
T    4  207  107    7    8    9  109  209  208
T   14  209  309  409  408  407  307  207  208
T   24  607  608  609  509  409  408  407  507
T   34  807  707  607  608  609  709  809  808
     5    9   10   11  111  211  210  209  109
    15  209  210  211  311  411  410  409  309
    25  409  410  411  511  611  610  609  509
    35  609  610  611  711  811  810  809  709
     6   11   12   13  113  213  212  211  111
    16  211  212  213  313  413  412  411  311
    26  411  412  413  513  613  612  611  511
    36  611  612  613  713  813  812  811  711
```

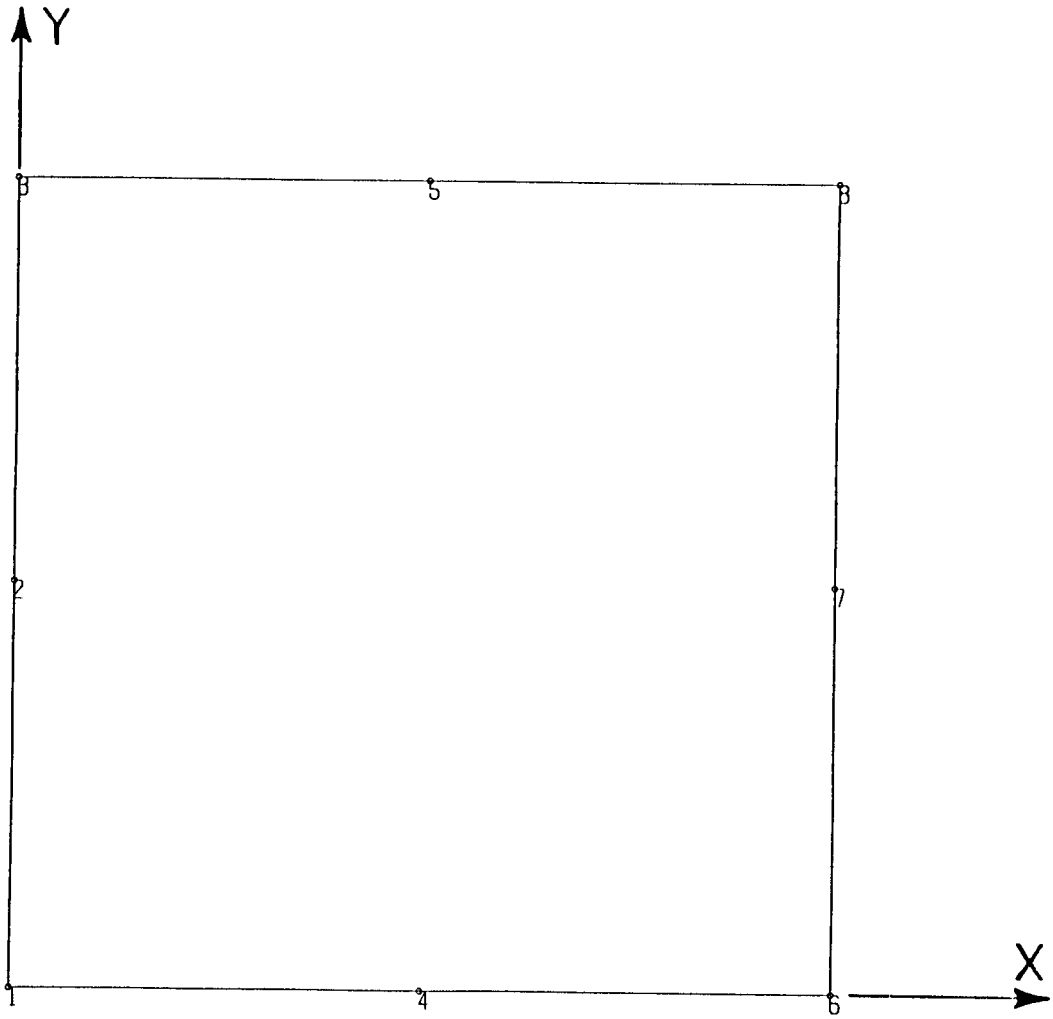
```
        7   13   14   15  115  215  214  213  113
       17  213  214  215  315  415  414  413  313
       27  413  414  415  515  615  614  613  513
       37  613  614  615  715  815  814  813  713
       50   90   91   92  192  292  291
       51   90  291  292  392  492  491
       52   90  491  492  592  692  691
       53   90  691  692  792  892  891
       60   92  192  292  283  215  115   15   83
       61  292  392  492  483  415  315  215  283
       62  492  592  692  683  615  515  415  483
       63  692  792  892  883  815  715  615  683
      101   50   51   52   41    3    2    1   40
      201   50  851  852  841  803  802    1   40
      102   52   42   30   20    5    4    3   41
      202  852  842  830  820  805  804  803  841
      103   30   31   32   21    7    6    5   20
      203  830  831  832  821  807  806  805  820
      104   32   33   34   22    9    8    7   21
      204  832  833  834  822  809  808  807  821
      105   34   43   56   44   11   10    9   22
      205  834  843  856  844  811  810  809  822
      106   56   57   58   45   13   12   11   44
      206  856  857  858  845  813  812  811  844
      107   52   53   54   46   32   31   30   42
      207  852  853  854  846  832  831  830  842
      108   54   55   56   43   34   33   32   46
      208  854  855  856  843  834  833  832  846
      109   70   71   72   61   52   51   50   60
      209   70  871  872  861  852  851   50   60
      110   72   62   54   53   52   61
      210  872  862  854  853  852  861
      111   72   63   56   55   54   62
      211  872  863  856  855  854  862
      112   72   64   58   57   56   63
      212  872  864  858  857  856  863
      113   90   81   72   71   70   80
      213   90  881  872  871   70   80
      114   90   91   92   82   58   64   72   81
      214   90  891  892  882  858  864  872  881
      115   92   83   15   14   13   45   58   82
      215  892  883  815  814  813  845  858  882
  NORMal 1 +
  CRACk 7 107 207 307 407 507 607 707 807

**BCSET
  ID-Bcset DISP1
  VALUe
  GMR-bc REG1
  SURFace-bc SURF1
    ELEMents-bc  201 202 203 204 205 206 207 208
    ELEMents-bc  209 210 211 212 213 214 215
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset DISP2
```

```
    VALUe
    GMR-bc REG1
    SURFace-bc SURF1
      ELEMents-bc  101 102 103 104 105 106 107 108
      ELEMents-bc  109 110 111 112 113 114 115
    DISPlacement-bc 2
      SPLIst ALL
      T-VAlue 1 0.0

  **BCSET
    ID-Bcset DISP3
    VALUe
    GMR-bc REG1
    SURFace-bc SURF1
      ELEMents-bc  4 5 6 7 14 15 16 17 24 25 26 27 34 35 36 37
      DISPlacement-bc 3
      SPLIst ALL
      T-VAlue 1 0.0

  **BCSET
    ID-Bcset TRAC3
    VALUe
    GMR-bc REG1
    SURFace-bc SURF1
      ELEMents-bc  50 51 52 53
      TRACtion-bc 3
      SPLIst ALL
      T-VAlue 1  -886.02
  $
  $ END OF DATA
```

```
 JOB TITLE:  BURIED CIRCULAR CRACK - NO SYMMETRY - a/w=0.1
 STRESS INTENSITY FACTORS AT TIME =   1.00000000       FOR REGION = REG1


 NODE        MODE-I          MODE-II        MODE-III
-------------------------------------------------------


  6     0.10103630E+04  0.00000000E+00  0.00000000E+00
206     0.10275806E+04  0.00000000E+00  0.00000000E+00
406     0.10175180E+04  0.00000000E+00  0.00000000E+00
606     0.10346392E+04  0.00000000E+00  0.00000000E+00
806     0.10095572E+04  0.00000000E+00  0.00000000E+00
```

6.3.7 HEAT TRANSFER EXAMPLES

```
EXAMPLE PROBLEM: HEAT601

ANALYSIS TYPE: TRANSIENT HEAT CONDUCTION
  2-D ANALYSIS

PROBLEM DESCRIPTION:
  THERMAL DIFFUSION IN A SQUARE, WITH ONE SIDE SUBJECTED TO A
  SUDDEN RISE IN TEMPERATURE. INITIALLY, THE SQUARE HAS A UNIFORM
  TEMPERATURE OF 70.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR WITH FOUR ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS,
    PP102-104.

SOLUTION POINTS TO VERIFY:
  GMR     NODE      TIME            (TEMPERATURE)
                              ANALYTICAL        GPBEST
  GMR1      1        1.0        79.10           78.26
                     2.0       101.46          100.66

RUN TIME:
    0.2 X BASE PROBLEM

MISCELLANEOUS:
  ONLY ONE QUADRATIC ELEMENT IS USED ALONG THE DIFFUSIVE LENGTH.
  THEREFORE, SOME ERROR EXISTS AT SMALL TIMES BECAUSE THE MESH
  CANNOT REPRODUCE THE TEMPERATURE PROFILE EXACTLY.
  EXTRANEOUS THERMOELASTIC MATERIAL PARAMETERS ARE INCLUDED, BUT
  HAVE NO EFFECT ON THIS ANALYSIS.
```



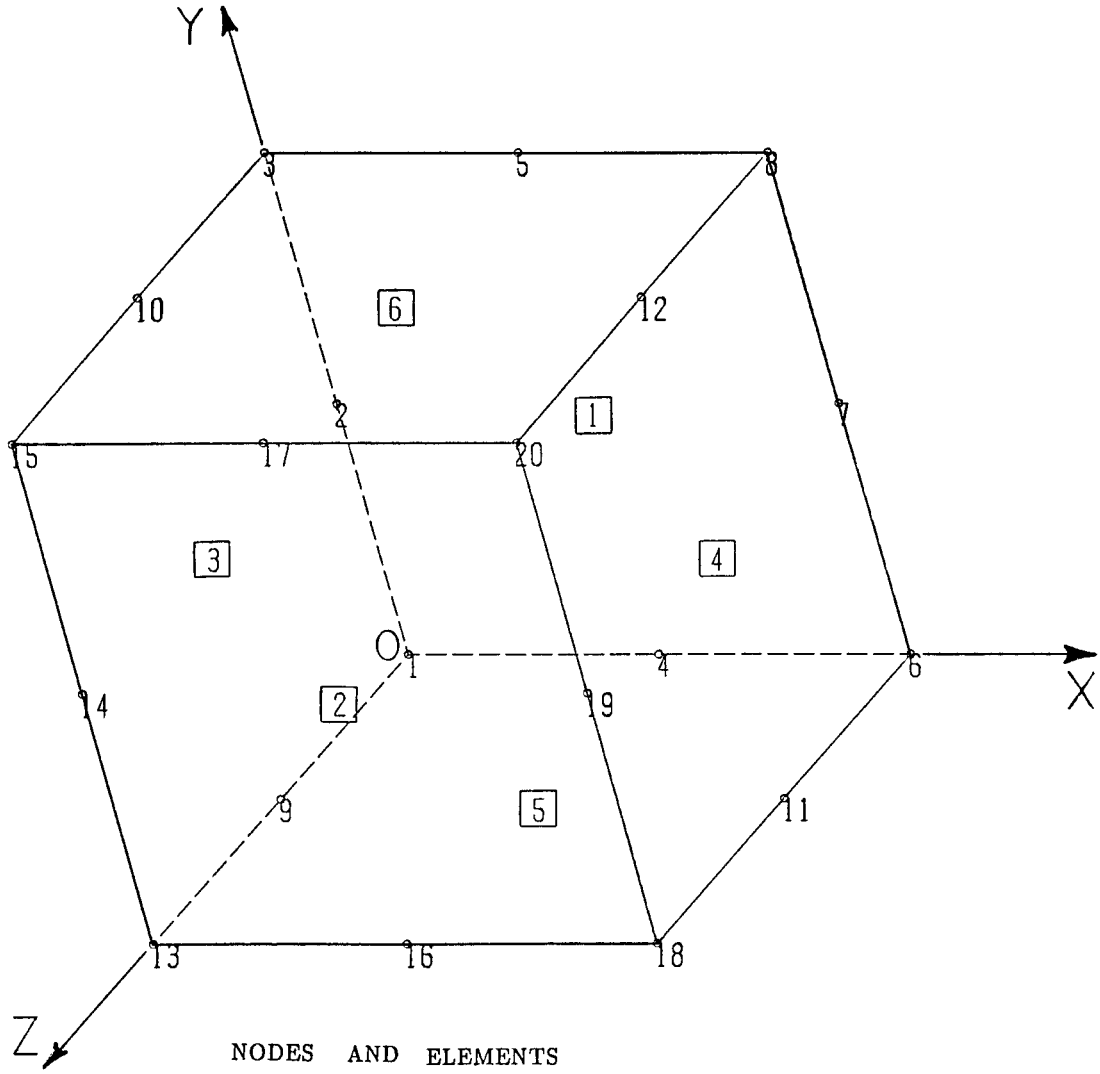
NODES

```
**CASE
  TITLe   TRANSIENT HEAT CONDUCTION IN A SQUARE
  PLANe
  HEAT TRANS
  TIMEs STEP  10  0.20
  PRINtout-control BOUNDARY


**MATERIAL
  ID-Material MAT1
  TEMPeratures  70.0

    EMODulus 10.0E+06    $   ASSUMED TEMPERATURE-INDEPENDENT
    POISson 0.25
    ALPHa 10.0E-06

    DENSity 0.1
    SPECific 2000.
    CONDuctivity 25.


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration  70.0     $   TEMPERATURE AT WHICH MATERIAL PROPERTIES
                 $     ARE DETERMINED FOR REGION
  TREFerence  70.0     $   INITIAL UNIFORM TEMPERATURE OF REGION
  POINts
   1       0.0      0.0
   2       0.0      0.5
   3       0.0      1.0
   4       0.5      0.0
   5       0.5      1.0
   6       1.0      0.0
   7       1.0      0.5
   8       1.0      1.0
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
  1    3  2  1
  2    6  7  8
  3    8  5  3
  4    1  4  6
  NORMal  +


**BCSET
  ID-Bcset TRAC16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 3
```

```
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1   170.0    $  SUDDEN INCREASE IN TEMPERATURE OF 100.0

$   ZERO FLUX BOUNDARY CONDITION ASSUMED ON ELEMENTS  1, 2, AND 4

$ END OF DATA
```

```
JOB TITLE:    TRANSIENT HEAT CONDUCTION IN A SQUARE
BOUNDARY SOLUTION AT TIME =      2.000000         FOR REGION = GMR1

ELEMENT  NODE NO.  TEMPERATURE     FLUX

    1       3     0.17000E+03  0.00000E+00
    1       2     0.12107E+03  0.00000E+00
    1       1     0.10066E+03  0.00000E+00

    2       6     0.10066E+03  0.00000E+00
    2       7     0.12107E+03  0.00000E+00
    2       8     0.17000E+03  0.00000E+00

    3       8     0.17000E+03 -0.29333E+04
    3       5     0.17000E+03 -0.28187E+04
    3       3     0.17000E+03 -0.29333E+04

    4       1     0.10066E+03  0.00000E+00
    4       4     0.10160E+03  0.00000E+00
    4       6     0.10066E+03  0.00000E+00
```

```
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 JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:
  GMR     NODE         (TEMPERATURE)
                  ANALYTICAL        GPBEST
  GMR1     22       25.00           25.00

  GMR     NODE           (FLUX-Y)
                  ANALYTICAL        GPBEST
  GMR1     22      -2500.0         -2500.2

RUN TIME:
   0.6 X BASE PROBLEM

MISCELLANEOUS:
  INTEGRATION FILES ARE SAVED FOR RESTART.
```



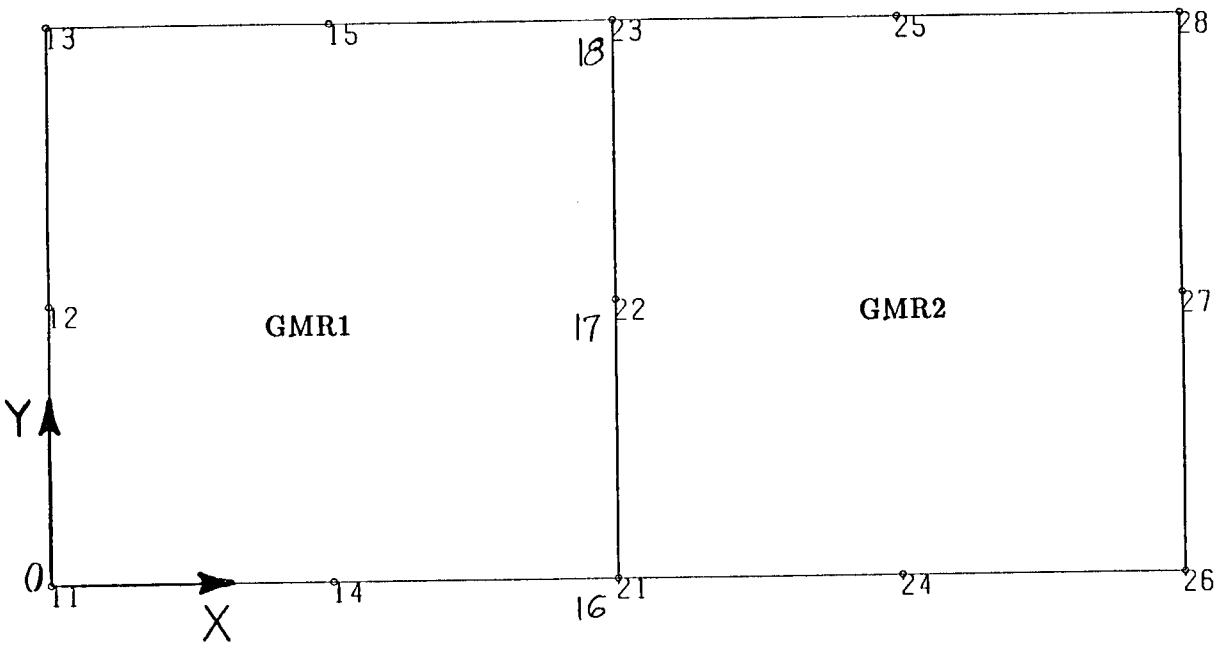
NODES AND ELEMENTS

```
**CASE
  TITLe   ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
  HEAT
  RESTart WRITE


**MATERIAL
  ID-Material MAT1
  TEMPeratures  70.0
    CONDuctivity 25.


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration  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-surface QUAD
    ELEMents-surface
  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 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-Bcset TRAC15
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 5
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset TRAC16
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 6
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1   100.0

$   ZERO FLUX BOUNDARY CONDITION ASSUMED ON ELEMENTS  1 THRU 4

$ END OF DATA
```

```
JOB TITLE:    ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
HEAT RATE CALCULATION AT TIME =      1.000000

HEAT RATE FOR REGION  GMR1

          ELEMENT     HEAT RATE
             1         0.00000E+00
             2         0.00000E+00
             3         0.00000E+00
             4         0.00000E+00
             5         0.25000E+04
             6        -0.25000E+04

  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 = GMR1


NODE       TEMPERATURE

 21        0.000000E+00
 22        0.249999E+02
 23        0.500009E+02
 24        0.750002E+02
 25        0.100000E+03


JOB TITLE:    ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
INTERIOR FLUX AT TIME =      1.000000           FOR REGION = GMR1


NODE        FLUX-X            FLUX-Y            FLUX-Z

 21      0.000000E+00     -0.249994E+04      0.000000E+00
 22      0.142300E-05     -0.250018E+04      0.142300E-05
 23      0.150466E-10     -0.250005E+04      0.149631E-10
 24     -0.142297E-05     -0.250013E+04     -0.142297E-05
 25      0.000000E+00     -0.249994E+04      0.000000E+00


 END OF ANALYSIS
```

```
EXAMPLE PROBLEM: HEAT603

ANALYSIS TYPE: STEADY-STATE HEAT CONDUCTION
  2-D ANALYSIS

PROBLEM DESCRIPTION:
  TEMPERATURE DISTRIBUTION WITH THERMAL RESISTANCE SPECIFIED
  ACROSS A GMR-TO-GMR INTERFACE. EACH REGION IS A UNIT SQUARE.
  TEMPERATURE IS SPECIFIED AS 0.0 AT X=0.0 AND AS 200.0 AT X=2.0.
  CONSEQUENTLY, THE TEMPERATURE PROFILE IS LINEAR IN THE X-DIRECTION
  WITHIN EACH REGION.

BOUNDARY ELEMENT MODEL:
  TWO GMR MODEL, 4 BOUNDARY ELEMENTS IN EACH GMR.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92-93.

SOLUTION POINTS TO VERIFY:
  GMR     NODE         (TEMPERATURE)
                  ANALYTICAL        GPBEST
  GMR1     16       66.67           66.65
  GMR2     21      133.33          133.35

RUN TIME:
   0.07 X BASE PROBLEM

MISCELLANEOUS:
  THE REGIONS HAVE DIFFERENT INITIAL (REFERENCE) TEMPERATURES. THUS,
  TDIF MUST BE INCLUDED IN THE INTERFACE DATA SET TO ACCOUNT FOR
  THIS DIFFERENCE IN THE COMPATIBILITY RELATIONSHIPS.

  INTEGRATION FILES ARE SAVED FOR RESTART.
```



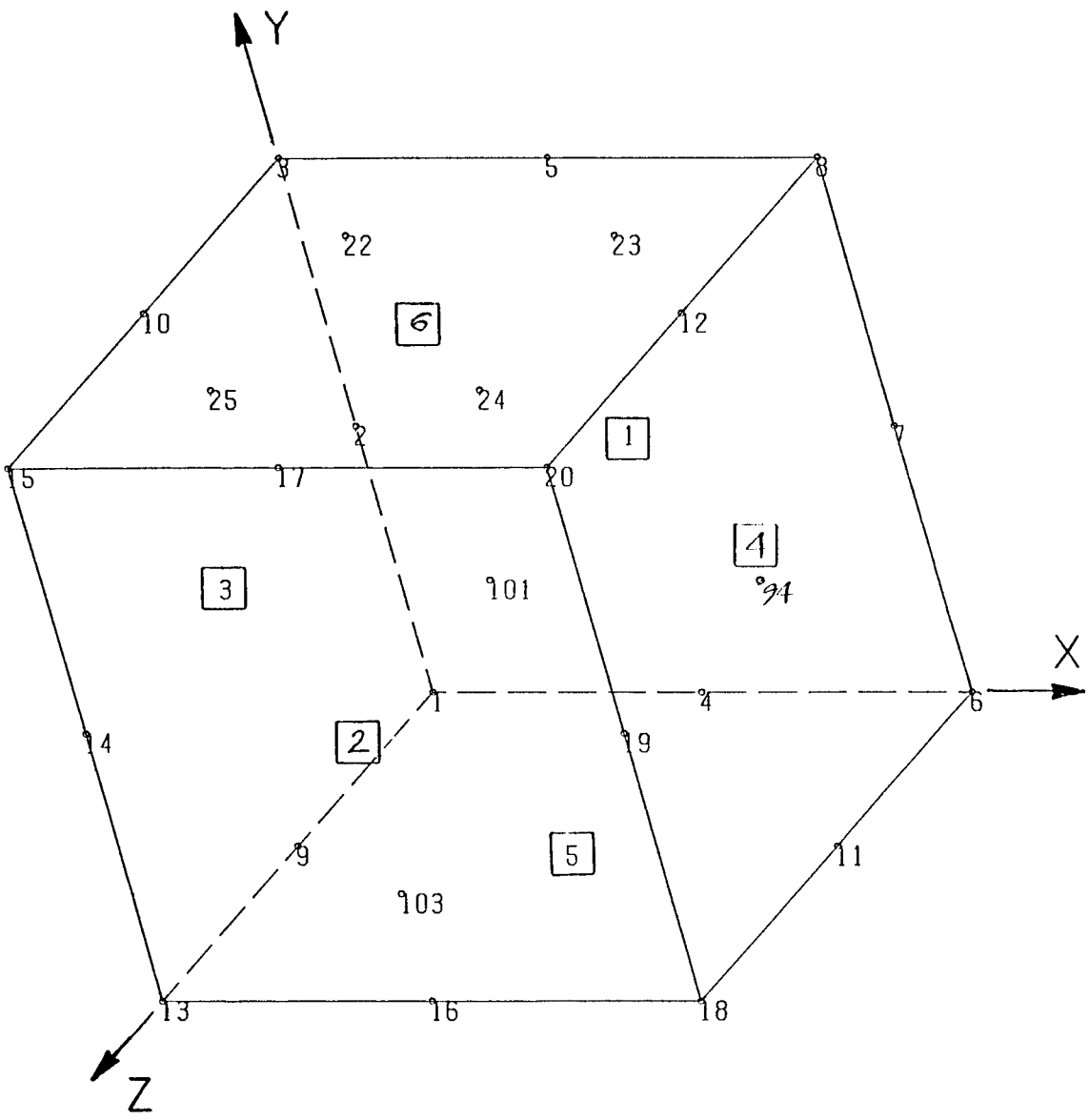
NODES

```
**CASE
  TITLe   THERMAL RESISTANCE BETWEEN GMR'S
  PLANe
  HEAT
  RESTart WRITE


**MATERIAL
  ID-Material MAT1
  TEMPeratures   0.0
    CONDuctivity  1.


**MATERIAL
  ID MAT2
  TEMPeratures   0.0
    CONDuctivity  1.


**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration   0.0
  TREFerence   0.0
  POINts
   11       0.0      0.0
   12       0.0      0.5
   13       0.0      1.0
   14       0.5      0.0
   15       0.5      1.0
   16       1.0      0.0
   17       1.0      0.5
   18       1.0      1.0
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
   11    13  12  11
   12    16  17  18
   13    18  15  13
   14    11  14  16
  NORMal  +


**GMR
  ID-Gmr GMR2
  MATErial MAT2
  TINTegration   0.0
  TREFerence 100.0
  POINts
   21       1.0      0.0
   22       1.0      0.5
   23       1.0      1.0
```

```
   24       1.5      0.0
   25       1.5      1.0
   26       2.0      0.0
   27       2.0      0.5
   28       2.0      1.0
  SURFace SURF2
    TYPE-surface QUAD
    ELEMents-surface
   21    23  22  21
   22    26  27  28
   23    28  25  23
   24    21  24  26
  NORMal  +


**INTERFACE
   GMR-1 GMR1
    SURFace-gmr-1 SURF1
    ELEMents-gmr-1    12
   GMR-2 GMR2
    SURFace-gmr-2 SURF2
    ELEMents-gmr-2    21
$
   RESIstance-value  1.0      $      THERMAL RESISTANCE

**BCSET
  ID-Bcset TRAC11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  11
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset TRAC22
  VALUe
  GMR-bc GMR2
  SURFace-bc SURF2
    ELEMents-bc  22
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1   200.0

$   ZERO FLUX BOUNDARY CONDITION ASSUMED ON REMAINING ELEMENTS

$ END OF DATA
```

```
JOB TITLE:    THERMAL RESISTANCE BETWEEN GMR'S
BOUNDARY SOLUTION AT TIME =      1.000000         FOR REGION = GMR1

ELEMENT  NODE NO.  TEMPERATURE     FLUX

   11      13     0.00000E+00  0.66659E+02
   11      12     0.00000E+00  0.66729E+02
   11      11     0.00000E+00  0.66659E+02


   12      16     0.66652E+02 -0.66696E+02
   12      17     0.66644E+02 -0.66711E+02
   12      18     0.66652E+02 -0.66696E+02


   13      18     0.66652E+02  0.00000E+00
   13      15     0.33324E+02  0.00000E+00
   13      13     0.00000E+00  0.00000E+00


   14      11     0.00000E+00  0.00000E+00
   14      14     0.33324E+02  0.00000E+00
   14      16     0.66652E+02  0.00000E+00
```

```
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 JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:
  GMR     NODE         (TEMPERATURE)
                  ANALYTICAL        GPBEST
  GMR1    101       0.500           0.500

  GMR     NODE            (FLUX-Z)
                  ANALYTICAL        GPBEST
  GMR1    102      -2.000          -2.000

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


**MATERIAL
  ID-Material MAT1
$    REFERENCE TEMPERATURE
  TEMPeratures   0.0
$    THERMAL CONDUCTIVITY
    CONDuctivity  1.


**MATERIAL
  ID-Material MAT2
$    REFERENCE TEMPERATURE
  TEMPeratures   0.0
$    THERMAL CONDUCTIVITY
    CONDuctivity  1.


**GMR
  ID-Gmr GMR1
$    USE MATERIAL MAT1 FOR REGION
  MATErial MAT1
$    REFERENCE TEMPERATURE FOR REGION
  TINTegration   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
  19    .50000   .25000   .50000
  20    .50000   .50000   .50000
  21    .25000   .50000   .25000
  22    .12500   .50000   .12500
  23    .37500   .50000   .12500
  24    .37500   .50000   .37500
  25    .12500   .50000   .37500
  92    .25000   .25000   .50000
  94    .50000   .25000   .25000
$
  SURFace SURF1
$    USE LINEAR FUNCTIONAL VARIATION
    TYPE-surface LINEAR
$
    ELEMents-surface
$  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-surface QUADRATIC
$
    ELEMents-surface
$  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 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-Bcset TEMP11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF2
    ELEMents-bc 1
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     0.0
$
$     SPECIFY TEMPERATURE = 1.0 AT Z=0.5 (ELEMENT 2, GMR1)
$
```

```
**BCSET
  ID-Bcset TEMP12
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 2
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     1.0
$
$     REMAINING ELEMENTS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)
$
$
$  END OF DATA
```

```
          JOB TITLE:    HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
          INTERIOR TEMPERATURE AT TIME =      1.000000           FOR REGION = GMR1


NODE       TEMPERATURE

101        0.500011E+00
102        0.100000E+01
103        0.100000E+01



          JOB TITLE:    HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
          INTERIOR FLUX AT TIME =      1.000000           FOR REGION = GMR1


NODE        FLUX-X            FLUX-Y            FLUX-Z

101      0.670418E-07      0.187336E-08     -0.200004E+01
102      0.000000E+00      0.000000E+00     -0.200004E+01
103      0.000000E+00      0.000000E+00     -0.200004E+01


 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.

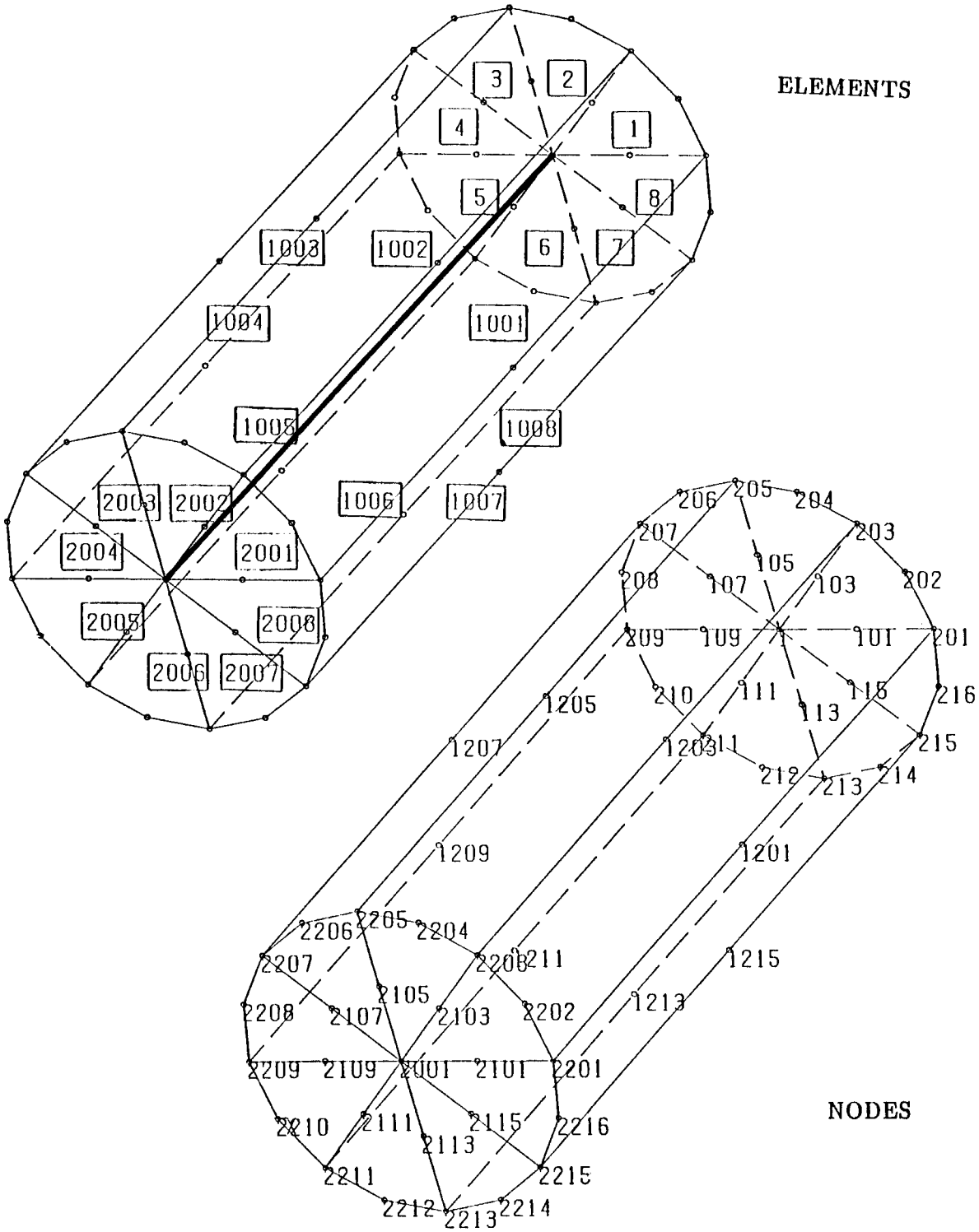
REFERENCE FOR ANALYTICAL SOLUTION:
CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP205-207.

SOLUTION POINTS TO VERIFY:

GMR	NODE	TIME	(TEMPERATURE) ANALYTICAL	GPBEST
GMR1	5005	4.00	0.1105	0.1067
		8.00	0.1971	0.1948

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.


ELEMENTS
3
2
4
1
5
8
1003
1002
6
7
1004
1001
1005
1008
2003
2002
1006
1007
2004
2001
2005
2008
2006
2007
206
205
204
207
203
105
208
107
103
202
209
109
101
201
210
111
115
216
1205
113
1207
1203
211
215
212
213
214
1209
1201
2206
2205
2204
2207
2208
211
1215
2105
2208
2107
2103
2202
1213
2209
2109
2001
2101
2201
NODES
2210
2111
2115
2216
2113
2211
2215
2212
2213
2214

```
**CASE
  TITLe   TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
$
$    THREE-DIMENSIONAL GEOMETRY (DEFAULT)
$    TRANSIENT HEAT TRANSFER
  HEAT TRANSIENT
  TIMEs STEP  8    1.0


**MATERIAL
  ID-Material MAT1
$    REFERENCE TEMPERATURE
  TEMPeratures   0.0
$    THERMAL CONDUCTIVITY
    CONDuctivity 1.0
$    DENSity
    DENSity 1.0
$    SPECIFIC HEAT
    SPECific 1.0


**GMR
  ID-Gmr GMR1
$    USE MATERIAL MAT1 FOR REGION
  MATErial MAT1
$    REFERENCE TEMPERATURE FOR REGION
  TINTegration   0.0
$
  POINts
$       ID          X            Y        Z
$
         1       0.00000      0.00000    0.0
       101       4.00000      0.00000    0.0
       103       2.82843      2.82843    0.0
       105       0.00000      4.00000    0.0
       107      -2.82843      2.82843    0.0
       109      -4.00000      0.00000    0.0
       111      -2.82843     -2.82843    0.0
       113       0.00000     -4.00000    0.0
       115       2.82843     -2.82843    0.0
       201       8.00000      0.00000    0.0
       202       7.39104      3.06147    0.0
       203       5.65685      5.65685    0.0
       204       3.06147      7.39104    0.0
       205       0.00000      8.00000    0.0
       206      -3.06147      7.39104    0.0
       207      -5.65685      5.65685    0.0
       208      -7.39104      3.06147    0.0
       209      -8.00000      0.00000    0.0
       210      -7.39104     -3.06147    0.0
       211      -5.65685     -5.65685    0.0
       212      -3.06147     -7.39104    0.0
```

```
        213       0.00000     -8.00000    0.0
        214       3.06147     -7.39104    0.0
        215       5.65685     -5.65685    0.0
        216       7.39104     -3.06147    0.0
$
       1201       8.00000      0.00000   20.0
       1203       5.65685      5.65685   20.0
       1205       0.00000      8.00000   20.0
       1207      -5.65685      5.65685   20.0
       1209      -8.00000      0.00000   20.0
       1211      -5.65685     -5.65685   20.0
       1213       0.00000     -8.00000   20.0
       1215       5.65685     -5.65685   20.0
$
       2001       0.00000      0.00000   40.0
       2101       4.00000      0.00000   40.0
       2103       2.82843      2.82843   40.0
       2105       0.00000      4.00000   40.0
       2107      -2.82843      2.82843   40.0
       2109      -4.00000      0.00000   40.0
       2111      -2.82843     -2.82843   40.0
       2113       0.00000     -4.00000   40.0
       2115       2.82843     -2.82843   40.0
       2201       8.00000      0.00000   40.0
       2202       7.39104      3.06147   40.0
       2203       5.65685      5.65685   40.0
       2204       3.06147      7.39104   40.0
       2205       0.00000      8.00000   40.0
       2206      -3.06147      7.39104   40.0
       2207      -5.65685      5.65685   40.0
       2208      -7.39104      3.06147   40.0
       2209      -8.00000      0.00000   40.0
       2210      -7.39104     -3.06147   40.0
       2211      -5.65685     -5.65685   40.0
       2212      -3.06147     -7.39104   40.0
       2213       0.00000     -8.00000   40.0
       2214       3.06147     -7.39104   40.0
       2215       5.65685     -5.65685   40.0
       2216       7.39104     -3.06147   40.0
$
$
  SURFace SURF1
$    USE QUADRATIC FUNCTIONAL VARIATION
    TYPE-surface QUAD
$
$    DISC AT NEAR END OF CYLINDER
$
  ELEMents-surface
$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
$
```

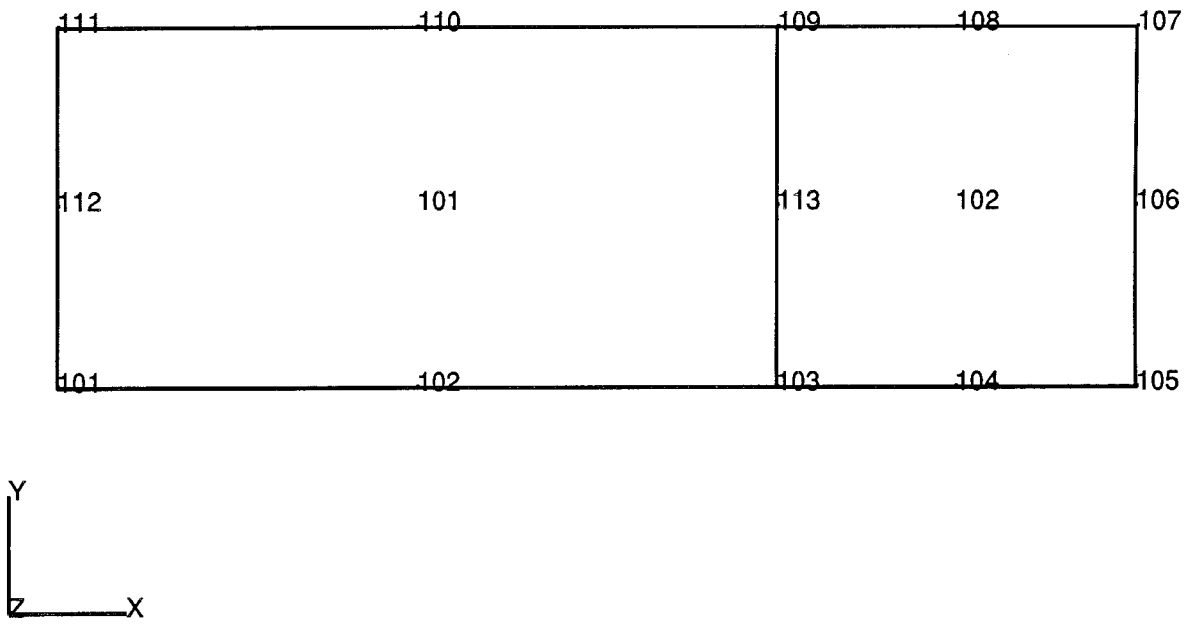
```
$    SURFACE OF OUTER CYLINDER
$
1001     201  202  203 1203 2203 2202 2201 1201
1002     203  204  205 1205 2205 2204 2203 1203
1003     205  206  207 1207 2207 2206 2205 1205
1004     207  208  209 1209 2209 2208 2207 1207
1005     209  210  211 1211 2211 2210 2209 1209
1006     211  212  213 1213 2213 2212 2211 1211
1007     213  214  215 1215 2215 2214 2213 1213
1008     215  216  201 1201 2201 2216 2215 1215
$
$    DISC AT FAR END OF CYLINDER
$
2001    2001 2101 2201 2202 2203 2103
2002    2001 2103 2203 2204 2205 2105
2003    2001 2105 2205 2206 2207 2107
2004    2001 2107 2207 2208 2209 2109
2005    2001 2109 2209 2210 2211 2111
2006    2001 2111 2211 2212 2213 2113
2007    2001 2113 2213 2214 2215 2115
2008    2001 2115 2215 2216 2201 2101
$
$    OUTWARD (RIGHT-HAND) NORMAL OF ELEMENT    1 IS NEGATIVE
  NORMal     1 -
$
$    SURFACE OF INNER CYLINDER
$
 HOLE
   POINts
9001       0.0      0.0      5.0
9002       0.0      0.0      7.5
9003       0.0      0.0     10.0
9004       0.0      0.0     15.0
9005       0.0      0.0     20.0
9006       0.0      0.0     25.0
9007       0.0      0.0     30.0
9008       0.0      0.0     32.5
9009       0.0      0.0     35.0
$
$    USE QUADRATIC FUNCTIONAL VARIATION FOR 3-NODED HOLE ELEMENTS
    TYPE-surface QUAD
$
    ELEMents-surface
$      ID    RAD    G1   G2   G3
     9001   1.00  9001 9002 9003
     9002   1.00  9003 9004 9005
     9003   1.00  9005 9006 9007
     9004   1.00  9007 9008 9009
$

  SAMPling-pointss POINTS
5001       1.7      0.0     20.0
5002       2.4      0.0     20.0
5003       3.1      0.0     20.0
5004       3.8      0.0     20.0
5005       4.5      0.0     20.0
5006       5.2      0.0     20.0
5007       5.9      0.0     20.0
```

```
5008       6.6      0.0     20.0
5009       7.3      0.0     20.0
$
$     SPECIFY TEMPERATURE = 1.0 ON INNER CYLINDER
$
**BCSET
  ID-Bcset TEMP1
$       VALUE BOUNDARY CONDITION SET
  VALUe
  GMR-line-bc GMR1
  HOLE-line-bc
    ELEMents-bc  9001  9002  9003  9004
  TEMPeratures-line-bc
    SPLIst ALL
    T-VAlue 1     1.0
$
$     SPECIFY TEMPERATURE = 0.0 ON OUTER CYLINDER
$
**BCSET
  ID-Bcset TEMP2
$       VALUE BOUNDARY CONDITION SET
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc  1001  1002  1003  1004
    ELEMents-bc  1005  1006  1007  1008
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1     0.0
$
$
$     ENDS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)


$ END OF DATA
```

```
          JOB TITLE:    TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
          HOLE ELEMENT SOLUTION AT TIME =      8.000000         FOR REGION = GMR1


ELEMENT  NODE NO.  TEMPERATURE     FLUX

 9001    9001     0.10000E+01 -0.85685E+00
 9001    9002     0.10000E+01 -0.64049E+00
 9001    9003     0.10000E+01 -0.58113E+00


 9002    9003     0.10000E+01 -0.58113E+00
 9002    9004     0.10000E+01 -0.57257E+00
 9002    9005     0.10000E+01 -0.57366E+00


 9003    9005     0.10000E+01 -0.57366E+00
 9003    9006     0.10000E+01 -0.57257E+00
 9003    9007     0.10000E+01 -0.58113E+00


 9004    9007     0.10000E+01 -0.58113E+00
 9004    9008     0.10000E+01 -0.64049E+00
 9004    9009     0.10000E+01 -0.85685E+00




          JOB TITLE:    TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
          INTERIOR TEMPERATURE AT TIME =      8.000000           FOR REGION = GMR1


 NODE       TEMPERATURE

5001        0.700817E+00
5002        0.509723E+00
5003        0.373628E+00
5004        0.272088E+00
5005        0.194836E+00
5006        0.135517E+00
5007        0.896075E-01
5008        0.535336E-01
5009        0.000000E+00



  END OF ANALYSIS
```

6.3.8 ELASTOPLASTICITY EXAMPLES

```
EXAMPLE PROBLEM: plas022.dat

ANALYSIS TYPE: ELASTO-PLASTICITY, PLANE STRESS

PROBLEM DESCRIPTION:
  A 2-D PLATE SUBJECTING TO SIMPLE TENSION
  PLANE STRESS
  NEWTON-RAPHSON METHOD
  THE INITIAL YIELD STRENGTH IS 1000.0 WITH HARDENING

BOUNDARY ELEMENT MODEL:
   1 GMR
   6 (3-NODE) SURFACE ELEMENTS, 2 (8-NODE) CELLS, 13 NODES

SOLUTION POINTS TO VERIFY:
  GMR   NODE         X-DISPLACEMENT AT TENSION=1400.0
                   ANALYTICAL                GPBEST
   1    105        1.62000                   1.62047
   1    106        1.62000                   1.62047
   1    107        1.62000                   1.62047
```


111
110
109
108
107
112
101
113
102
106
101
102
103
104
105
Y
Z
X

```
**CASE
  TITLe  PLASTIC ANALYSIS, PLANE STRESS, NEWTON-RAPHSON
  TIMEs  1.0  2.0  3.0
              $ (SPECIFIES TIMES AT WHICH LOADS ARE APPLIED.)
  PLANe STRESS
  PLASticity   $ ELASTO-PLASTIC ANALYSIS
  NEWTon-raphson       $ NEWTON-RAPHSON METHOD
  NEUTral-file
  ECHO-input

**MATERIAL
  ID-Material MAT1
    TEMPeratures  70.0
    EMODulus  10000.0
    POISson  0.2

  INELastic
    VON-mises  $ VON MISES PLASTICITY MATERIAL MODEL
    YIELd  1000.0
    CURVe      $ DEFINES HARDENING OF MATERIAL
           1100.0  0.1

**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TINTegration   70.0
  POINts
     101   0.0  0.0
     102   1.0  0.0
     103   2.0  0.0
     104   2.5  0.0
     105   3.0  0.0
     106   3.0  0.5
     107   3.0  1.0
     108   2.5  1.0
     109   2.0  1.0
     110   1.0  1.0
     111   0.0  1.0
     112   0.0  0.5
     113   2.0  0.5

  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
          101    101  102  103
          102    103  104  105
          103    105  106  107
          104    107  108  109
          105    109  110  111
          106    111  112  101
    NORMal  +

 VOLUme
```

```
    TYPE-volume QUAD
    CELLs
       101   101  102  103  113  109  110  111  112
       102   103  104  105  106  107  108  109  113
    FULL   $ (SPECIFIES THAT THIS GMR IS COMPLETELY FILLED WITH CELLS
           $ AND INITIAL STRESS EXPANSION IS USED TO ACCURATELY
           $ EVALUATE JUMP TERMS)

**BCSET
  ID-Bcset DISP11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 106
  DISPlacement-bc  1
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset DISP12
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 101 102
  DISPlacement-bc  2
    SPLIst ALL
    T-VAlue 1     0.0
**BCSET
  ID-Bcset TRAC11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 103
    TIMEs-bc 1.0  2.0  3.0
  TRACtion-bc  1
    SPLIst ALL
    T-VAlue 1      800.0
    T-VAlue 2     1200.0
    T-VAlue 3     1400.0

$ END OF DATA
```

```
   JOB TITLE:   PLASTIC ANALYSIS, PLANE STRESS, NEWTON-RAPHSON

   INTERIOR DISPLACEMENT AT TIME =    3.000000     FOR REGION = GMR1
```

NODE	X DISPLACEMENT	Y DISPLACEMENT
101	.000000E+00	.000000E+00
102	.540156E+00	.000000E+00
103	.108031E+01	.000000E+00
104	.135039E+01	.000000E+00
105	.162047E+01	.000000E+00
106	.162047E+01	-.119112E+00
107	.162047E+01	-.238225E+00
108	.135039E+01	-.238224E+00
109	.108031E+01	-.238224E+00
110	.540156E+00	-.238225E+00
111	.000000E+00	-.238225E+00
112	.000000E+00	-.119112E+00
113	.108031E+01	-.119112E+00

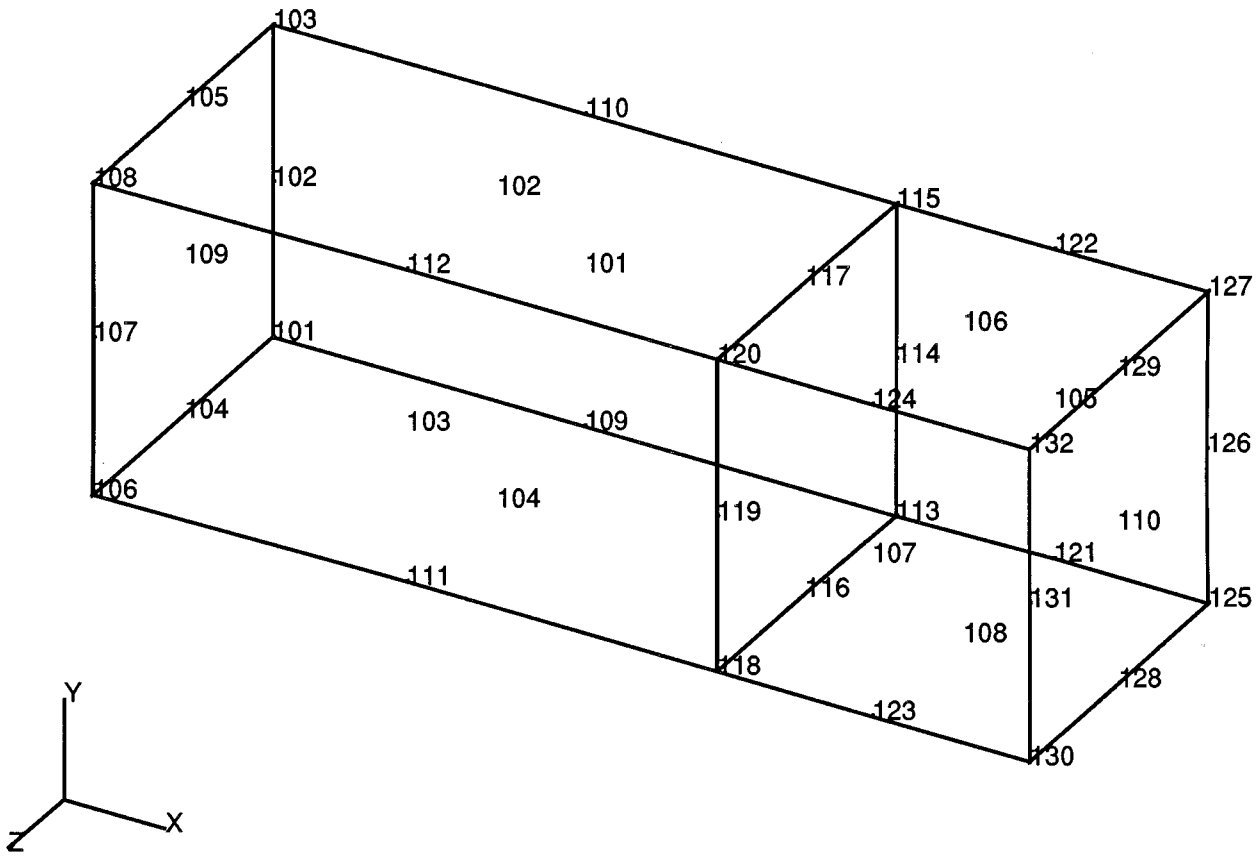
```
   JOB TITLE:   PLASTIC ANALYSIS, PLANE STRESS, NEWTON-RAPHSON

   INTERIOR STRESS AT TIME =    3.000000       FOR REGION = GMR1
```

NODE	SIGMAXX	SIGMAYY	TAUXY	SIGMAZZ
101	.139987E+04	-.230131E+00	.000000E+00	.344034E-14
102	.139987E+04	-.229794E+00	.000000E+00	-.313768E-14
103	.139987E+04	-.230193E+00	.000000E+00	.328918E-14
104	.139987E+04	-.229919E+00	.000000E+00	.119733E-13
105	.139987E+04	-.229713E+00	.000000E+00	-.139129E-13
106	.139987E+04	-.229568E+00	.000000E+00	.183314E-13
107	.139987E+04	-.229570E+00	.000000E+00	-.635733E-14
108	.139987E+04	-.229459E+00	.000000E+00	.233768E-14
109	.139987E+04	-.229460E+00	.000000E+00	-.477879E-14
110	.139987E+04	-.229463E+00	.000000E+00	.722613E-14
111	.139987E+04	-.230019E+00	.000000E+00	.581177E-15
112	.139987E+04	-.229573E+00	.000000E+00	.397902E-14
113	.139987E+04	-.229721E+00	-.193335E-04	-.781695E-14

```
END OF GPBEST ANALYSIS
```

6.3.9 CONCURRENT-THERMOPLASTICITY EXAMPLES

```
EXAMPLE PROBLEM: cpls017.dat

ANALYSIS TYPE: THERMAL PLASTICITY, 3-D, STEADY STATE

PROBLEM DESCRIPTION:
  A 3-D BLOCK ROLLER CONSTRAINED AT BOTH ENDS
  SUBJECTING TO A UNIFORM TEMPERATURE INCREASE FROM 0.0
  STEADY STATE ANALYSIS
  TEST MELTING EFFECT

BOUNDARY ELEMENT MODEL:
   1 GMR
  10 (8-NODE) SURFACE ELEMENTS
   2 (20-NODE) CELLS
  32 NODES

SOLUTION POINTS TO VERIFY:
                      SIGMA-XX AT TEMPERATURE = 160
   GMR   NODE      ANALYTICAL                 GPBEST
    1    101       -656.60                    -655.83
    1    102       -656.60                    -655.35
    1    103       -656.60                    -655.21
    1    104       -656.60                    -655.35
```


103
105
110
108
102
102
115
122
109
112
101
117
127
106
107
101
120
114
129
124
105
104
103
109
132
126
106
104
119
113
110
107
121
111
116
131
125
108
118
128
123
130
Y
X
Z

```
**CASE
  TITLe  CTHERMAL PLASTICITY, 3-D, STEADY-STATE, NEWTON-RAPHSON
  TIMEs-output  1.0  2.0  3.0  4.0  5.0  6.0  7.0  8.0
  CTHErmal PLASTICITY  $ STEADY-STATE THERMOPLASTIC ANALYSIS
  NEWTon-raphson
  NEUTral-file
  ECHO-input

**MATERIAL
  ID-Material MAT1
    TEMPeratures  0.0
    EMODulus  10000.0
    POISson  0.2
    ALPHa 0.001
    SPECific  1.0
    CONDuctivity  1.0
    TPEAk 0.0     $ THE YIELD STRENGTH REMAINS CONSTANT TO THIS
TEMP.
    TMELt 200.0   $ THE MELTING TEMPERATURE

  INELastic
    VON-mises       $ TEMPERATURE-DEPENDENT VON MISES MODEL
    YIELd 1000.0
    CURVe           $ DEFINE HARDENING
          1100.0  0.1

**GMR
  ID-Gmr GMR1
  MATErial MAT1
  TREFerence   0.0
  TINTegration   0.0

  POINts
     101   0.0   0.0   0.0
     102   0.0   0.5   0.0
     103   0.0   1.0   0.0
     104   0.0   0.0   0.5
     105   0.0   1.0   0.5
     106   0.0   0.0   1.0
     107   0.0   0.5   1.0
     108   0.0   1.0   1.0
     109   1.0   0.0   0.0
     110   1.0   1.0   0.0
     111   1.0   0.0   1.0
     112   1.0   1.0   1.0
     113   2.0   0.0   0.0
     114   2.0   0.5   0.0
     115   2.0   1.0   0.0
     116   2.0   0.0   0.5
     117   2.0   1.0   0.5
     118   2.0   0.0   1.0
     119   2.0   0.5   1.0
     120   2.0   1.0   1.0
```

```
      121   2.5   0.0   0.0
      122   2.5   1.0   0.0
      123   2.5   0.0   1.0
      124   2.5   1.0   1.0
      125   3.0   0.0   0.0
      126   3.0   0.5   0.0
      127   3.0   1.0   0.0
      128   3.0   0.0   0.5
      129   3.0   1.0   0.5
      130   3.0   0.0   1.0
      131   3.0   0.5   1.0
      132   3.0   1.0   1.0
  SURFace SURF1
    TYPE-surface QUAD
    ELEMents-surface
      101      101  102  103  110  115  114  113  109
      102      103  110  115  117  120  112  108  105
      103      106  111  118  119  120  112  108  107
      104      101  109  113  116  118  111  106  104
      105      113  121  125  126  127  122  115  114
      106      115  122  127  129  132  124  120  117
      107      118  123  130  131  132  124  120  119
      108      113  121  125  128  130  123  118  116
      109      101  102  103  105  108  107  106  104
      110      125  126  127  129  132  131  130  128
    NORMal  101 +

  VOLUme
    TYPE-volume QUAD
    CELLs
      101    101 109 113 114 115 110 103 102 104 116 117 105
      101    106 111 118 119 120 112 108 107
      102    113 121 125 126 127 122 115 114 116 128 129 117
      102    118 123 130 131 132 124 120 119
    FULL

$
$   GMR1 BOUNDARY CONDITIONS
$
**BCSET
  ID-Bcset DISP13
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 101 105
  DISPlacement-bc 3
    SPLIst ALL
    T-VAlue 1 0.0

**BCSET
  ID-Bcset DISP12
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 104 108
  DISPlacement-bc 2
    SPLIst ALL
    T-VAlue 1 0.0
```

```
**BCSET
  ID-Bcset DISP11
  VALUe
  GMR-bc GMR1
  SURFace-bc SURF1
    ELEMents-bc 109 110
    TIMEs-bc  1.0  2.0  3.0  4.0  5.0  6.0  7.0  8.0
  DISPlacement-bc 1
    SPLIst ALL
    T-VAlue 1   0.0
    T-VAlue 2   0.0
    T-VAlue 3   0.0
    T-VAlue 4   0.0
    T-VAlue 5   0.0
    T-VAlue 6   0.0
    T-VAlue 7   0.0
    T-VAlue 8   0.0
  TEMPeratures-bc
    SPLIst ALL
    T-VAlue 1    50.0
    T-VAlue 2   100.0
    T-VAlue 3   110.0
    T-VAlue 4   120.0
    T-VAlue 5   130.0
    T-VAlue 6   140.0
    T-VAlue 7   150.0
    T-VAlue 8   160.0

$   END OF DATA
```

```
   JOB TITLE:   CTHERMAL PLASTICITY, 3-D, STEADY-STATE, NEWTON-RAPHSON
   INTERIOR DISPLACEMENT AT TIME =      8.000000    FOR REGION = GMR1
```

NODE	X DISPLACEMENT	Y DISPLACEMENT	Z DISPLACEMENT	TEMPERATURE
101	.000000E+00	.000000E+00	.000000E+00	.160000E+03
102	.000000E+00	.110151E+00	.000000E+00	.160000E+03
103	.000000E+00	.220328E+00	.000000E+00	.160000E+03
104	.000000E+00	.000000E+00	.110151E+00	.160000E+03
105	.000000E+00	.220291E+00	.110149E+00	.160000E+03
106	.000000E+00	.000000E+00	.220328E+00	.160000E+03
107	.000000E+00	.110149E+00	.220291E+00	.160000E+03
108	.000000E+00	.220311E+00	.220311E+00	.160000E+03
109	.311124E-04	.000000E+00	.000000E+00	.160000E+03

```
   JOB TITLE:   CTHERMAL PLASTICITY, 3-D, STEADY-STATE, NEWTON-RAPHSON
   INTERIOR STRESS AT TIME =      8.000000           FOR REGION = GMR1
```

NODE	SIGMAXX	SIGMAYY	SIGMAZZ	TAUXY	TAUXZ	TAUYZ
101	-.655830E+03	-.499862E+00	-.499498E+00	.000000E+00	.000000E+00	.000000E+00
102	-.655358E+03	.991398E-01	-.137276E+00	-.396278E-01	.000000E+00	-.921068E-01
103	-.655213E+03	.353642E+00	-.853008E-01	-.674528E-01	-.242316E-08	-.147317E+00
104	-.655358E+03	-.137012E+00	.993586E-01	.000000E+00	-.396891E-01	-.922254E-01
105	-.655240E+03	.139273E+00	.725914E-01	.000000E+00	-.333579E-01	.565986E-01
106	-.655213E+03	-.848310E-01	.353547E+00	-.239269E-08	-.675036E-01	-.147403E+00
107	-.655240E+03	.725224E-01	.139355E+00	-.333603E-01	.000000E+00	.566028E-01
108	-.655064E+03	.283084E+00	.283654E+00	-.519270E-01	-.519570E-01	.217001E+00
109	-.655832E+03	-.487781E+00	-.487869E+00	.560412E-01	.560256E-01	.000000E+00

```
    END OF GPBEST ANALYSIS
```

7.0 APPLICATIONS

The analyses developed in **GPBEST** have been applied to a vast array of realistic engineering problems in references listed in Section 9. A few representative examples are described in the following sections:

7.1 Elastic Analysis

7.2 Heat transfer Analysis

7.3 Periodic Acoustic Analysis

7.4 Periodic Dynamic Analysis

7.5 Poroelastic Analysis

7.6 Thermo-Elastic Analysis

7.7 Electromagnetic Analysis

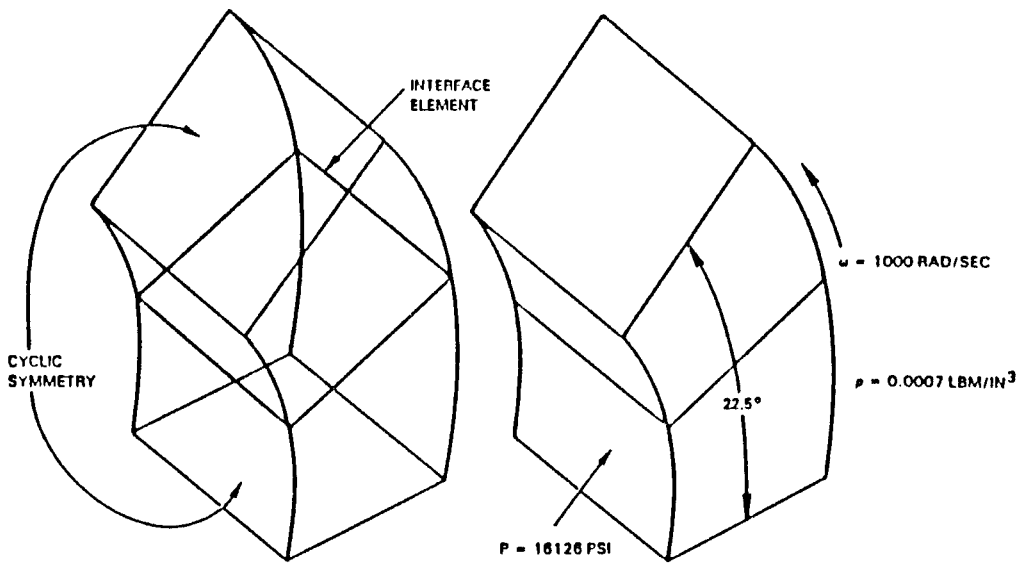
7.8 Fracture Analysis

7.9 Elasto-Plastic Analysis

7.1 ELASTIC ANALYSIS

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).



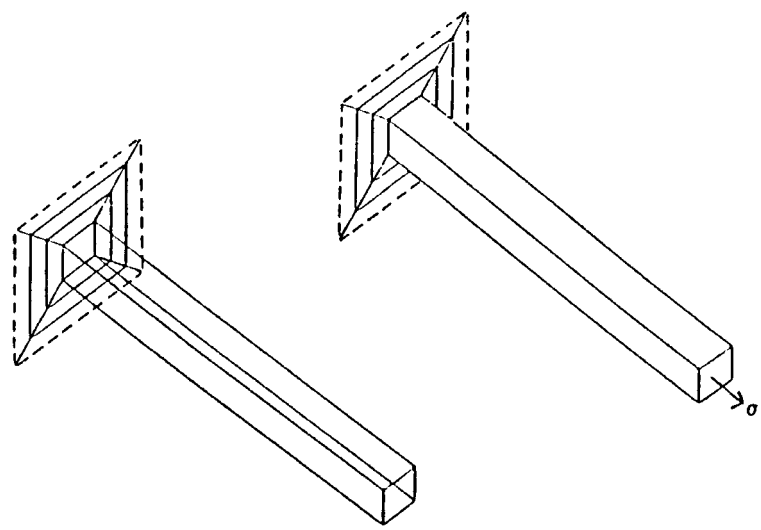
		PRESSURE		SPEED	
		ANALYTICAL	BEST	ANALYTICAL	BEST
ID	U_R	5.90×10^{-3}	5.91×10^{-3}	7.13×10^{-4}	7.13×10^{-4}
	σ_R	−16126	−16126	21	0
	σ_θ	41928	41997	5752	5751
OD	U_R	4.80×10^{-3}	4.81×10^{-3}	6.29×10^{-4}	6.29×10^{-4}
	σ_R	0	0	21	0
	σ_θ	25802	25865	3390	3371

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).



- - - - - BOUNDARY AT INFINITY

	Displacement at Tip of Beam
Analytical	1.13×10^{-2}
One Region, Linear Variation	5.00×10^{-4}
One Region, Quadratic Variation	6.91×10^{-4}
Two Regions, Linear Variation	1.05×10^{-2}
Two Regions, Quadratic Variation	1.12×10^{-2}

Fig. 7.1.2 A column attached to an elastic half-space

Turbine Component Analysis [34,35]:

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.

Subregion	Elements	Linear (Nodes)	Quadratic (Nodes)
1	60	61	180
2	86	85	256
3	107	98	303
4	106	96	298
5	80	76	232
TOTAL	439	416	1269

The system equations for a fully linear analysis contain 1248 equations.

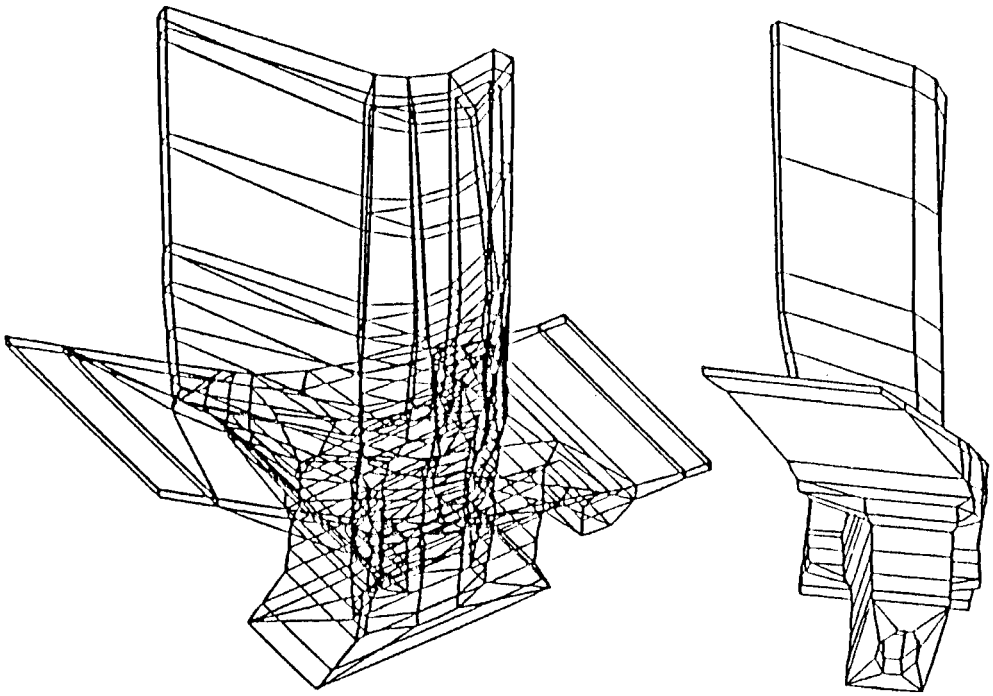


Fig. 7.1.3a **GPBEST** Model of Cooled Turbine Blade

Elastic Analysis

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).

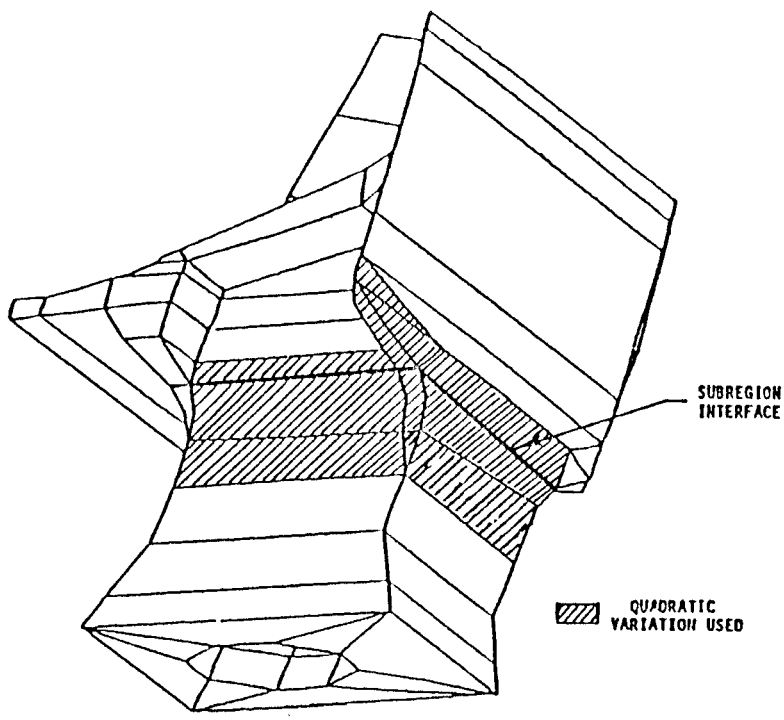


Fig. 7.1.3b Local Quadratic Surface Elements for the **GPBEST** Model of a Cooled Turbine Blade

Stresses in Rotating Sphere [38]:

This is an example of the stresses in a solid elastic sphere centered at the origin of a Cartesian coordinate system rotating about the vertical axis. Due to symmetry, only half of the body is modeled. Four elements with a total of 9 nodes are used. Note that the centerline is not discretized. The nodes along the r axis are constrained against displacement in the z direction. The only loading applied is that due to a rotation about the z axis. The results are compared to theoretical and finite element results in Fig. 7.1.4 Again, excellent agreement is obtained.

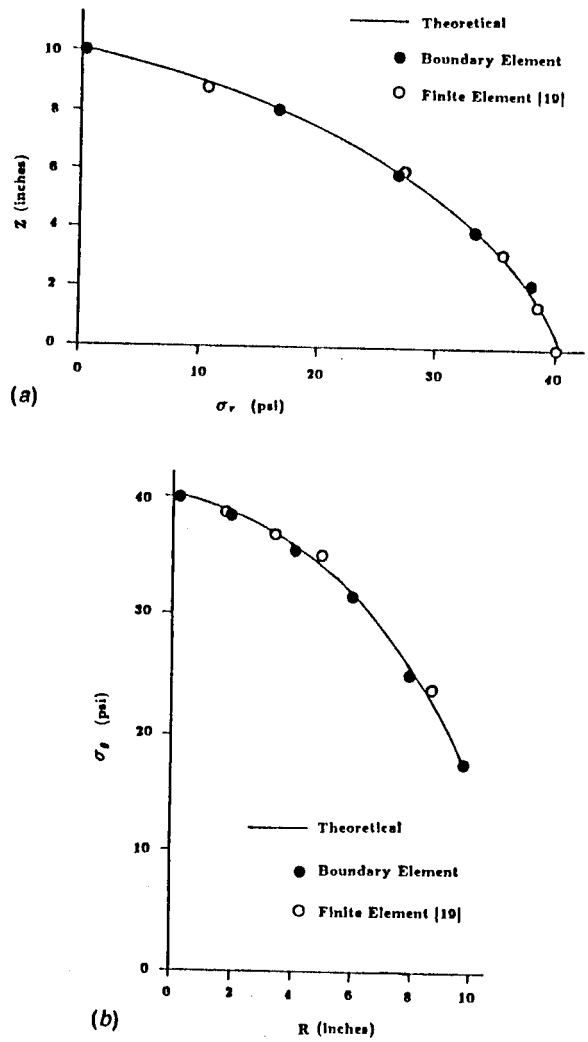


Fig. 7.1.4 (a) Radial Stress at R=0; (b) Tangential stress at Z=0

Rotating Disk[38]:

The previous problems showed the accuracy of the method for two simple geometries. This example is chosen to test the method for a more geometrically complex situation. Because analytic solutions for irregular geometries are scarce and since we would like to be able to assess the validity of our results, a rotating disk, previously analyzed in Ref. 16 using the finite element method, is used.

The disk is modeled for boundary element analysis with 14 elements and 29 nodes, for a total of 58 degrees of freedom. Additionally, eight internal points are located within the body. We note that the finite element analysis summarized in Ref. 16 uses 18 cubic elements having a total of 238 degrees of freedom.

Fig. 7.1.5a,b shows plots of tangential and equivalent stresses. Insufficient data are given in Ref. 6 for a detailed comparison of results; however, the trends are obviously similar.

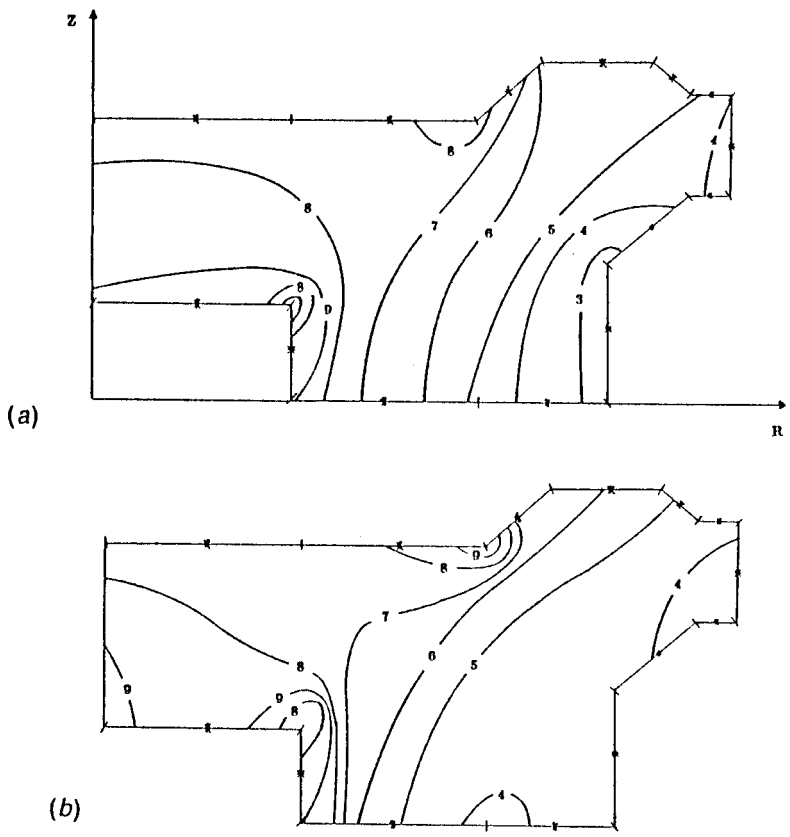


Fig. 7.1.5 (a) Contours of tangential stress in rotating disk
Fig. 7.1.5 (b) Contours of equivalent stress in rotating disk

Conical Water Tank [38]:

An analysis of a water tank, shown in Fig. 7.1.6a, is carried out using BEM. The hoop stress on the inside surface of the tank subjected to hydrostatic pressure is shown in Fig. 7.1.6a along with the result from Chapter 9 of Ref. 16. The two results are similar although an exact comparison is not possible since geometry data and material constants are not given for the later case. In Fig. 7.1.6b, a comparison of BEM solutions is presented for the hoop stress on both the inner and outer surface (along A-B) of the tank for two different loadings. The first loading is hydrostatic pressure only and the second is a combination of the hydrostatic pressure and self-weight. Note that in the lower section the hoop stress at the outer surface is compressive, i.e., the bending effect is dominant there.

One hundred isoparametric quadratic boundary elements are used to model the body. The tank is assumed to be constructed of concrete with ρ = 4.65 lbm /ft^3, E = 3.472 $\times$ 10^6 psi, and ν = 0.18. The thickness of the uniform section is taken to be 1 ft. The time required for integration is reduced by dividing the tank into 6 subregions. Two or more elements are used at every interface to insure an accurate representation of the variables across the interface. The computation time for a single region mesh is 50% greater than that of the 6 subregion model, although the computed results are almost identical. Solution to within 5% of the one shown is achievable with half the number of elements.

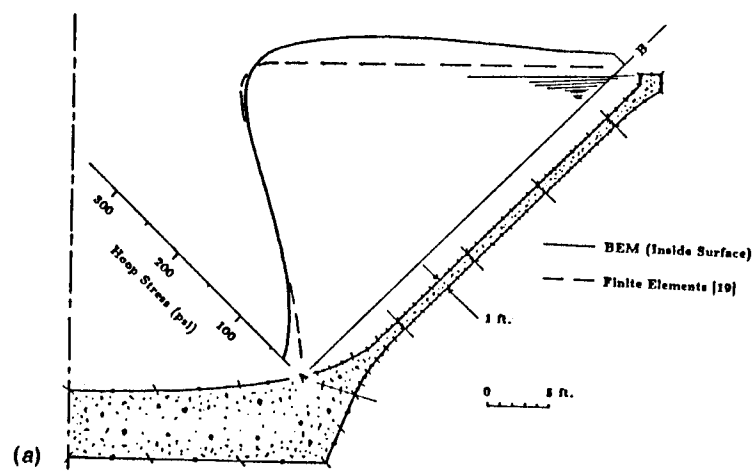


Fig. 7.1.6a Conical water tank subjected to hydrostatic pressure

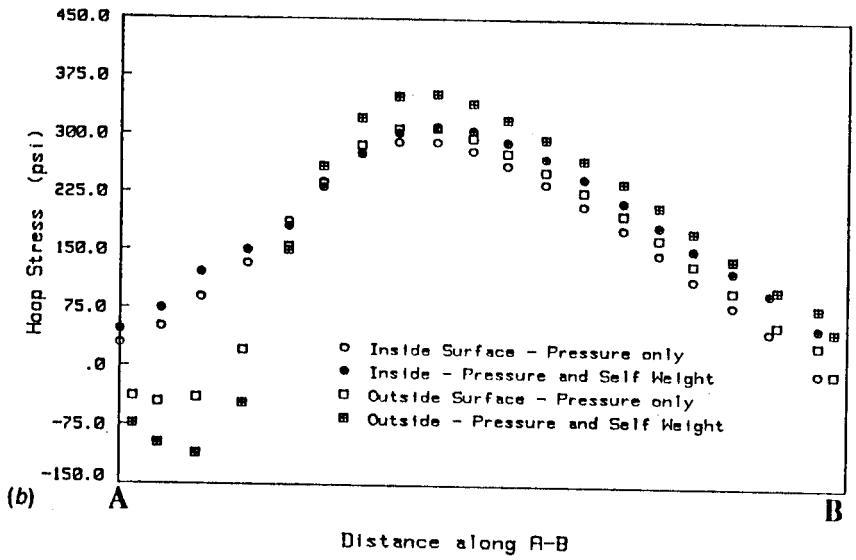


Fig. 7.1.6b Conical water tank subjected to hydrostatic pressure and self weight

Toriodal Shell under Uniform Internal Pressure [49]:

This problem is a purely axisymmetric one which involves the circumferential order $n = 0$ only. The thickness of the toriodal shell is $0.5 inch$ and its average radius is $15\ inches$ (Figure 7.1.7a). Poisson's ratio is 0.3 and Young's modulus is $10^7\ psi$. The only applied loading is the uniform internal pressure $p = 1.0\psi$. Because of symmetry with respect to $X - Y$ plane, only upper half of the toroidal shell is analyzed. A 6 region mesh with 12 elements in each region is used (Figure 7.1.7b) for this upper half toroidal shell and a roller boundary condition is applied on the interface between the upper half and the lower half of the toroidal shell. The average radial displacement on the mid-surface of toriodal shell by the present analysis is plotted in Figure 7.1.7c and compared with the result by Zienkiewicz et al [67], Delpak [68], Giannini and Miles [69] and Mukherjee and Poddar [70]. Besides, the tangential stresses and meridian stress resultants calculated by averaging the stress on internal and external surfaces are plotted and compared with results by FEM in Figures 7.1.7d and 7.1.7e. The maximum radial displacement and the maximum meridian stress resultant from the present analysis is slightly higher than the previously published results by shell theory. It is interesting to note that the Finite Element results differ by 10% depending on the type of the shell elements used. The previous BEM results for this problem is due to Mukherjee and Poddar [70] who speciallized the solids formulation into a thin shell theory. In the present analysis, the toriodal shell is treated as a three-dimensional solid and very good result is seen to have been obtained.

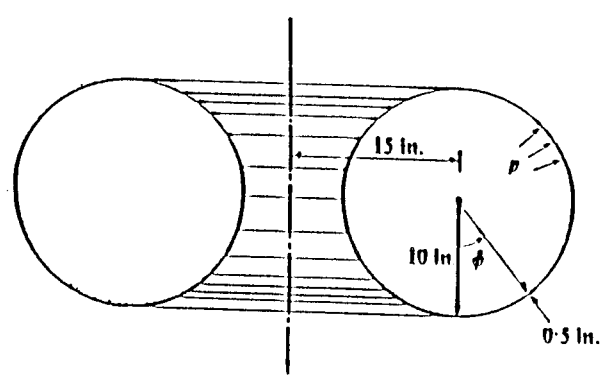


Fig. 7.1.7a Toroidal Shell under Uniform Internal Pressure $p = 1.0\psi$

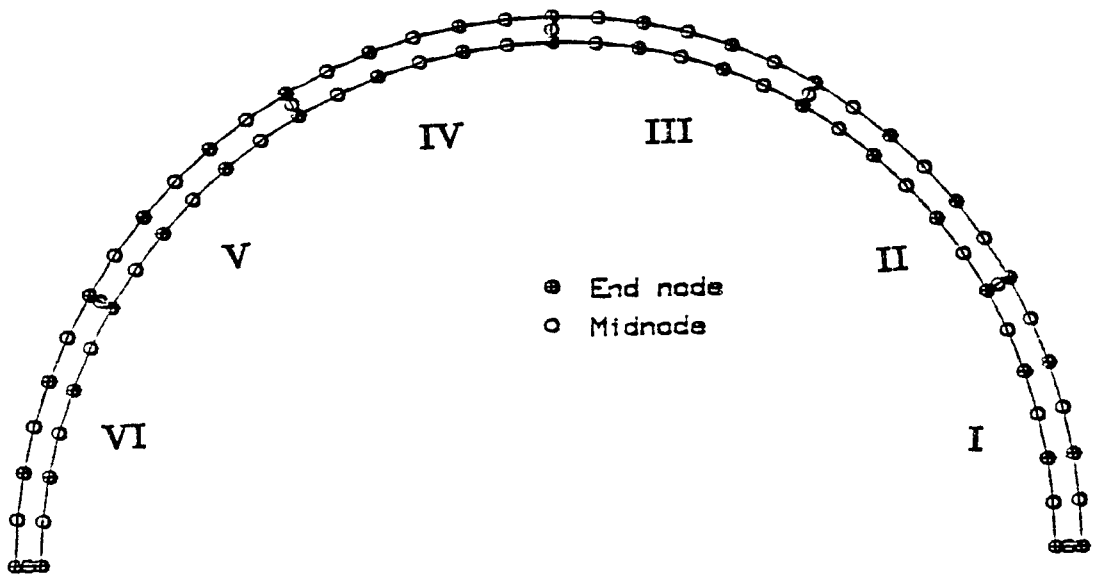


Fig. 7.1.7b Six Region Boundary Element Model of Toroidal Shell

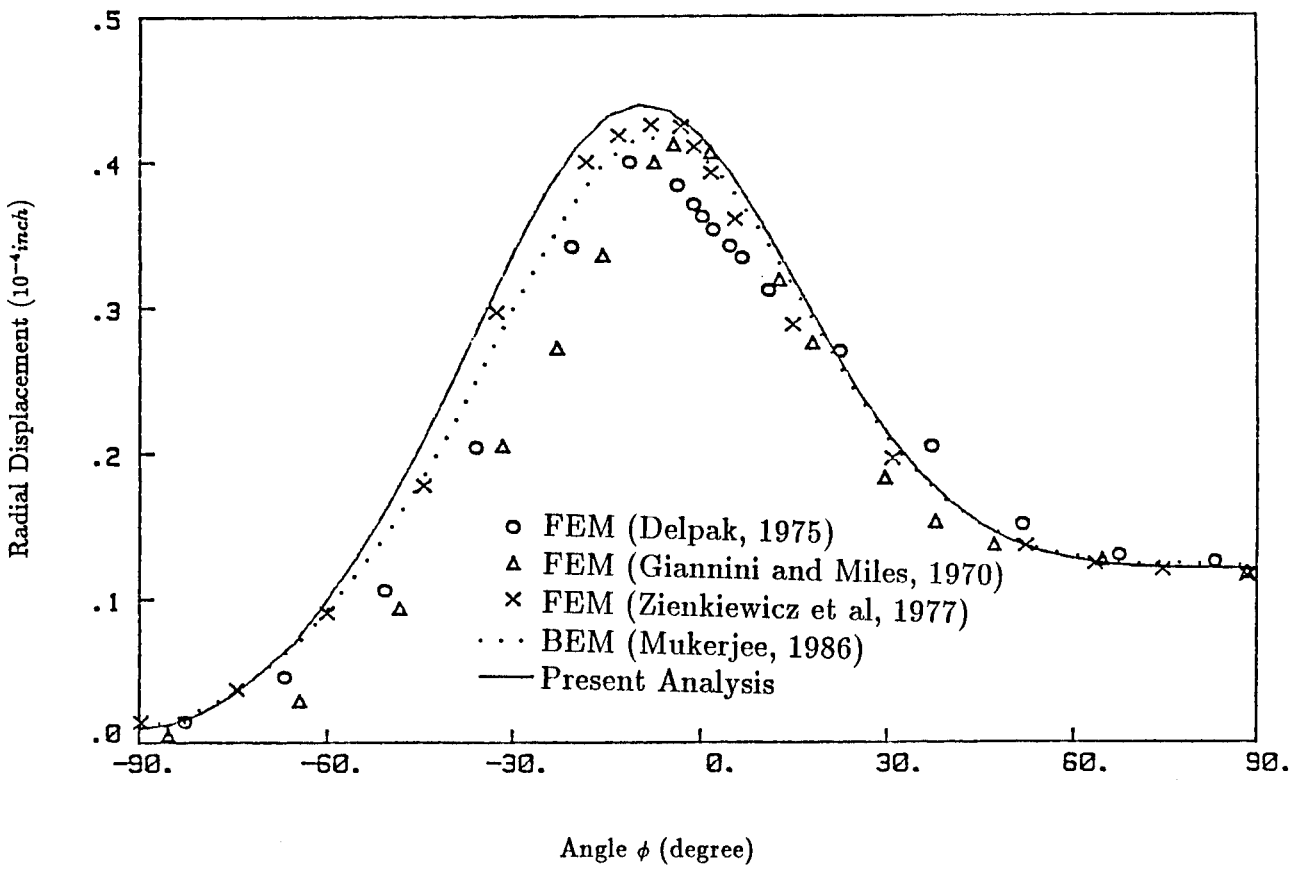


Fig. 7.1.7c Radial Displacement of Toroidal Shell

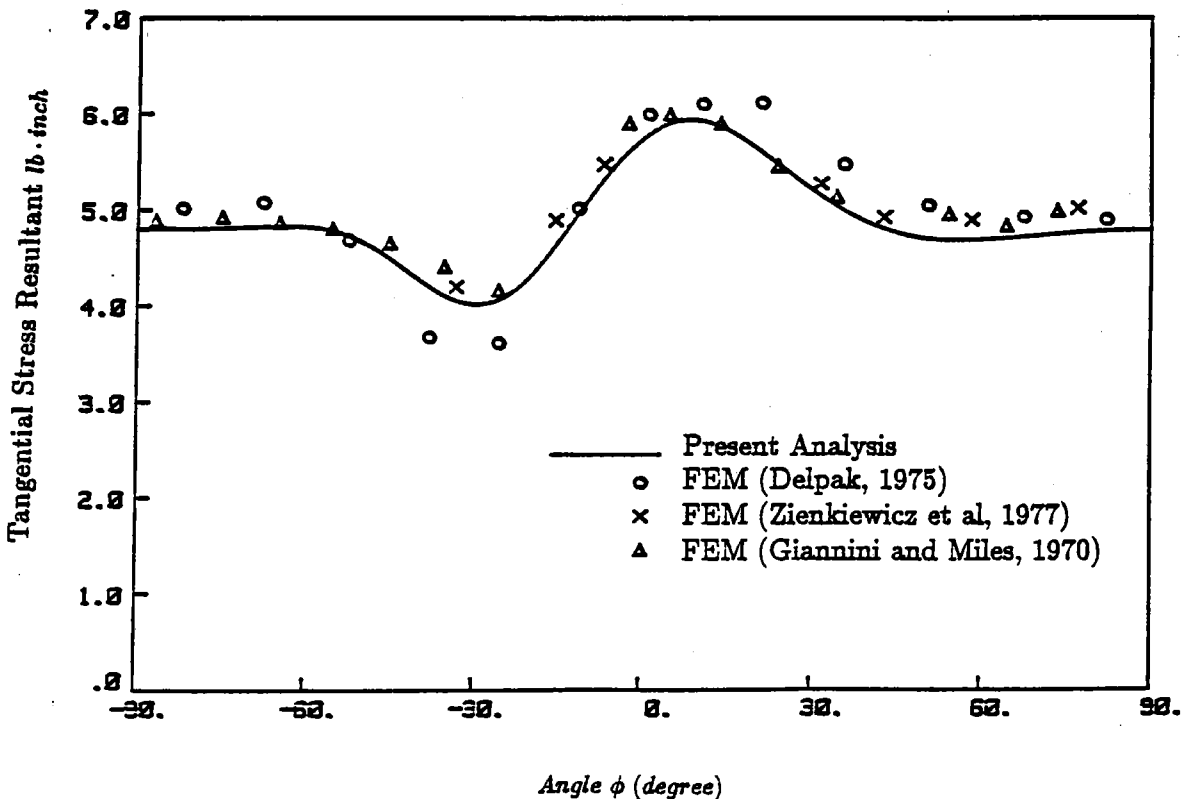


Fig. 7.1.7d Tangential Stress Resultant of Toroidal Shell

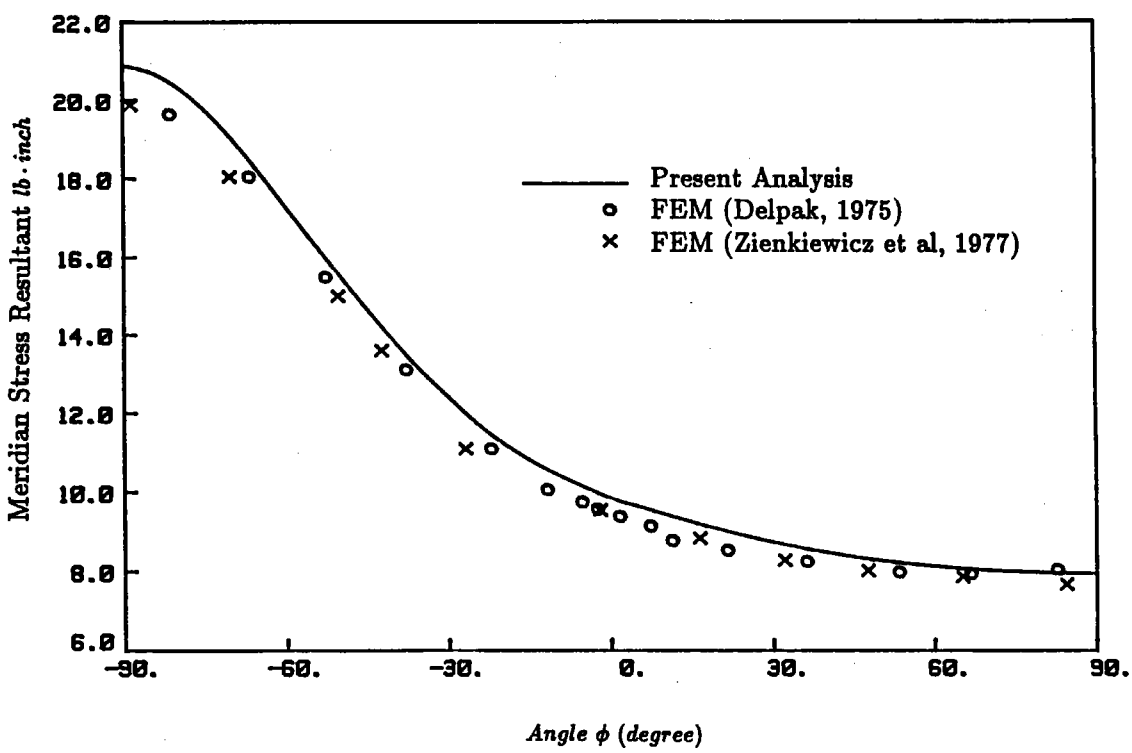


Fig. 7.1.7e Meridian Stress Resultant of Toroidal Shell

An Axisymmetric Gas Turbine Component under Bending [49]:

An axisymmetric gas turbine engine shell structure under a bending moment of $50\ kip \cdot inch$ at one end is shown in Figure 7.1.8a. The bending moment is generated by a rigid shaft which enters its right end and is not shown here. It is assumed that this end moment is distributed in such a way that only the analysis of the circumferential order $n = 1$ is considered. An approximate solution to the problem by membrane theory and experiments have been given in Kraus [71]. To solve it by the membrane theory, the structure was approximated by combining seven segments of different axisymmetric shells and flat plate (Figure 7.1.8a). The thinnest part of the turbine shell is ignored and at this end the nominal bending stress calculated by simple beam theory is applied as external loading to represent the end moment M. The position of the central axis was not given in Kraus [71], accordingly it was determined by comparing the average bending stress of $25.0 \cos\theta\ ksi.$ with that given by the membrane theory for an applied load $50.0\ kip \cdot inch$. The Poisson's ratio is assumed to be 0.30. To analyze the problem by BEM, the 3 region axisymmetric boundary element model shown in Figure 7.1.8b used. Note that the previous approximation of geometry for applying the membrane theory is not necessary for BEM, so the stress calculated by BEM should be more realistic. The comparisons between the stress distributions on both the outside surface and inside surface by experimental method, membrane theory and BEM are plotted in Figure 7.1.8c - 7.1.8f. The results agree very well except that a 40% higher maximum meridian stress on outside surface (see Figure 7.1.8c) is obtained by BEM than that by the membrane theory. The underestimation by the membrane theory for this problem is due to the fact that, in the analysis by membrane theory, a slightly thicker equivalent section of the shell than the original structure was used (see Figure 7.1.8a).

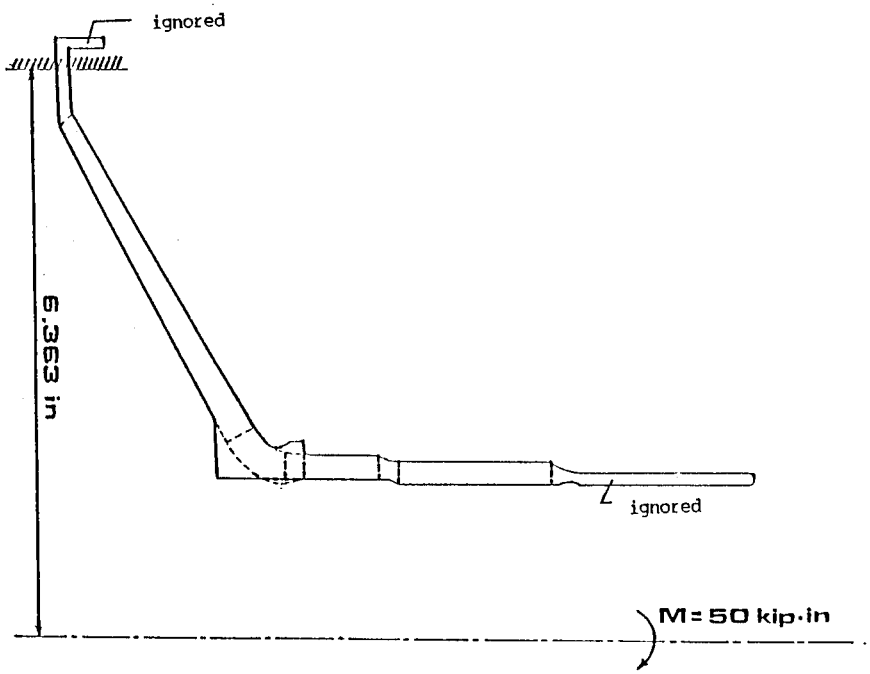


Fig. 7.1.8a Approximated Model of Axisymmetric Gas Turbine Shell in the Analysis by Membrane Theory

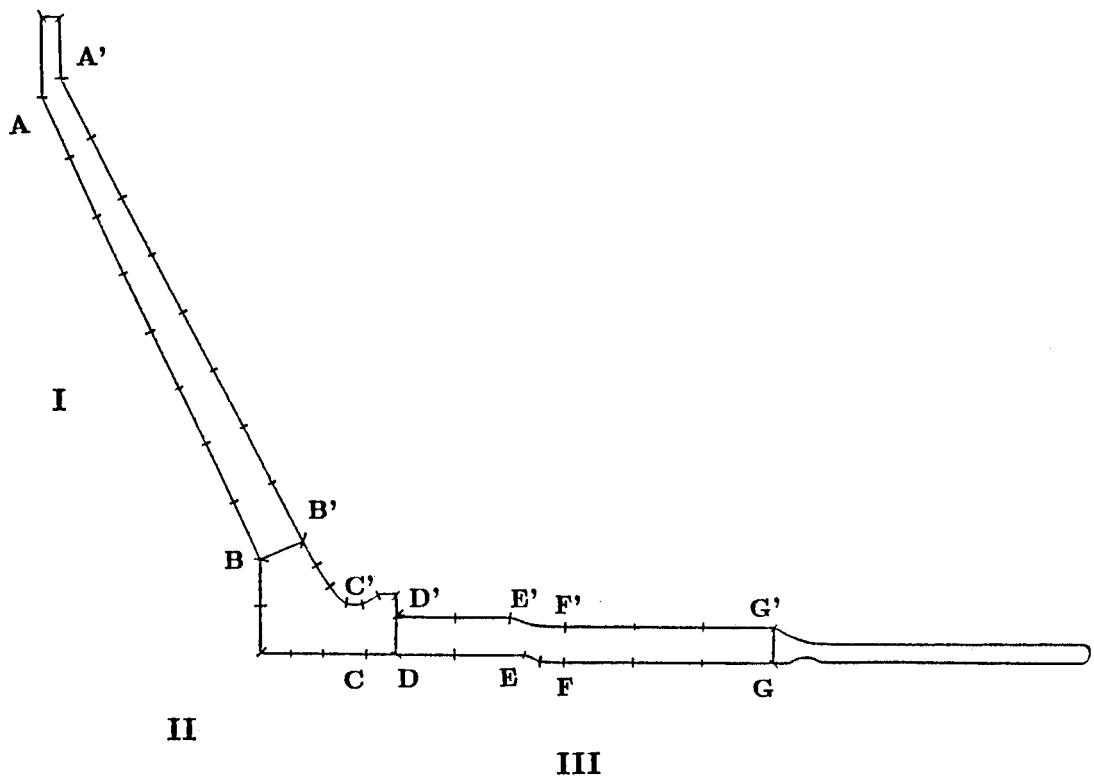


Fig. 7.1.8b Three region Boundary Element Model of Axisymmetric Gas Turbine Shell

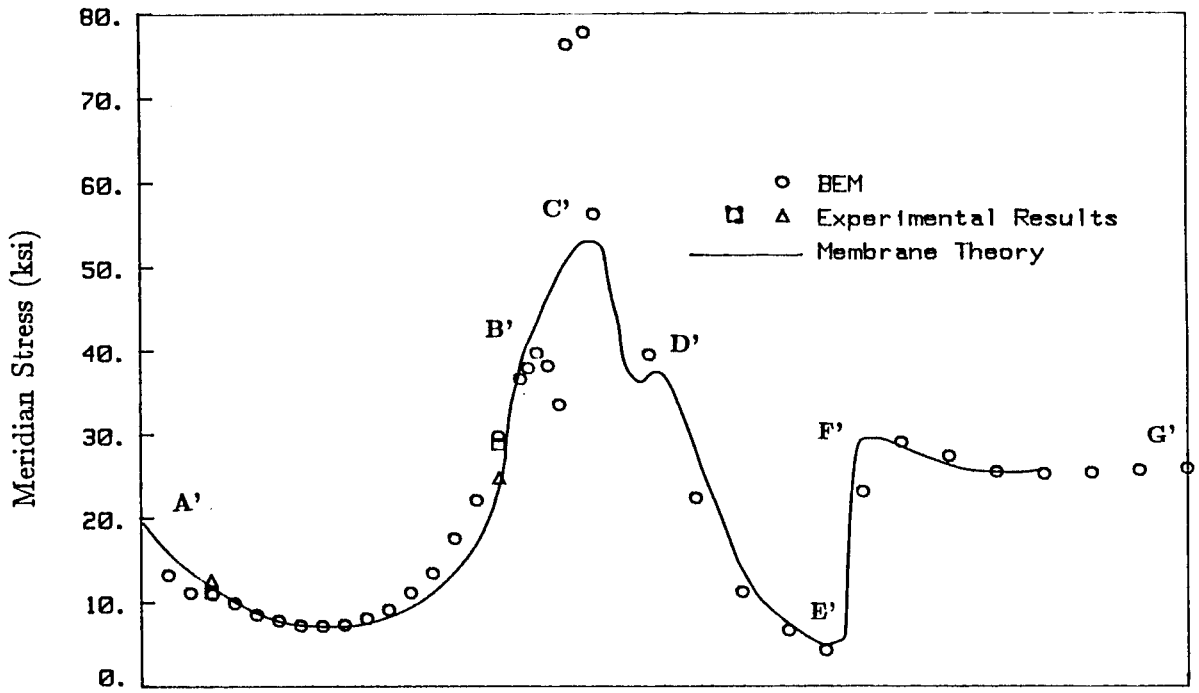


Fig. 7.1.8c Meridian Stress Distribution on the External Surface of Turbine Shell

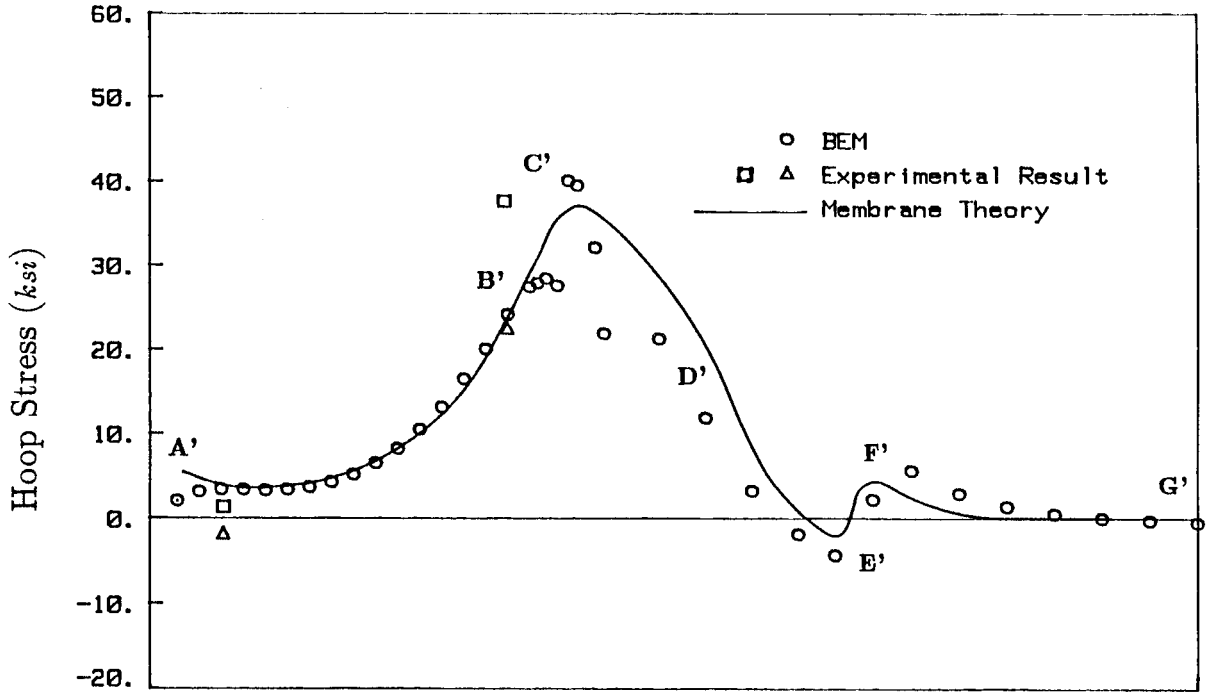


Fig. 7.1.8d Tangential Stress Distribution on the External Surface of Turbine Shell

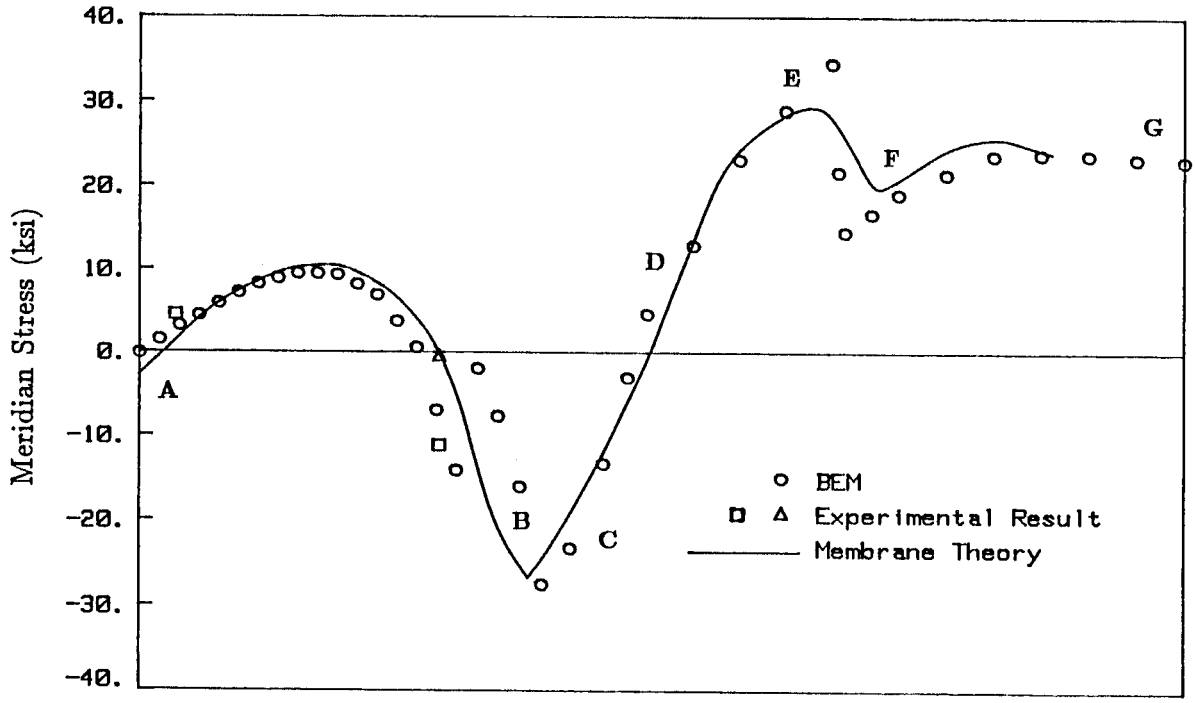


Fig. 7.1.8e Meridian Stress Distribution on the Internal Surface of Turbine Shell

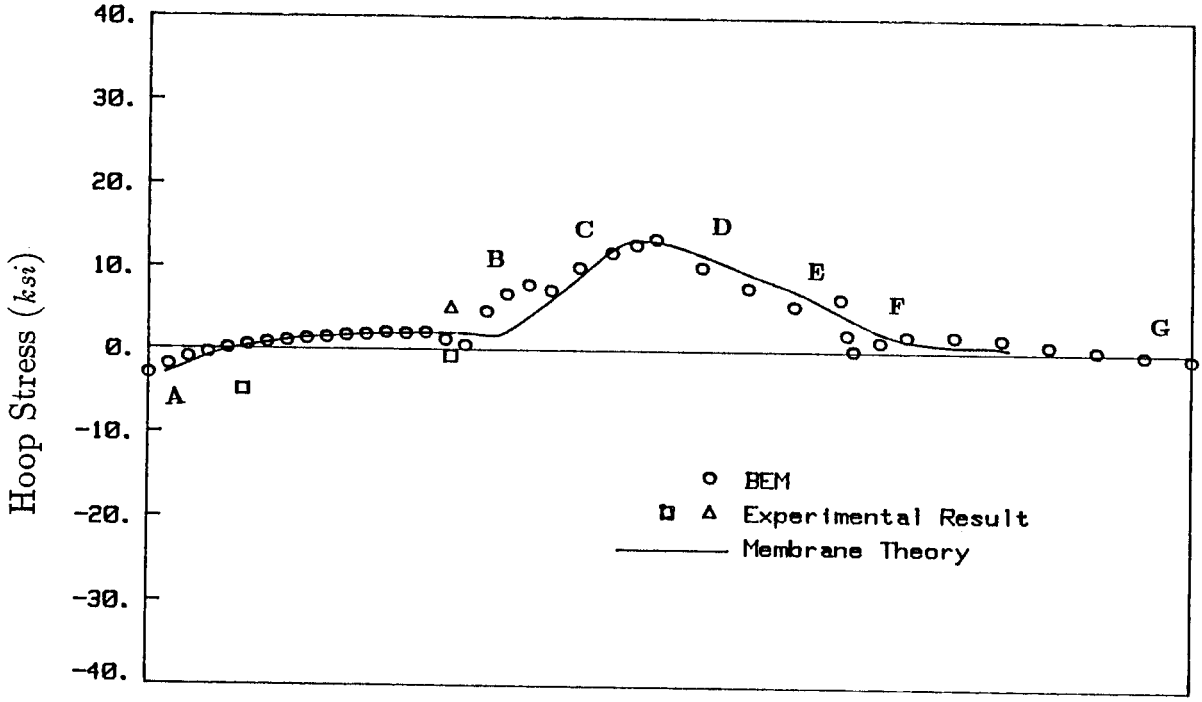


Fig. 7.1.8f Tangential Stress Distribution on the Internal Surface of Turbine Shell

7.2 HEAT TRANSFER ANALYSIS

Tube and Disk Fin Heat Exchanger [42]:

This problem is the heat transfer analysis of a portion of a heat exchanger. Figure 7.2.1a 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 modeled 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 7.2.1b 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.

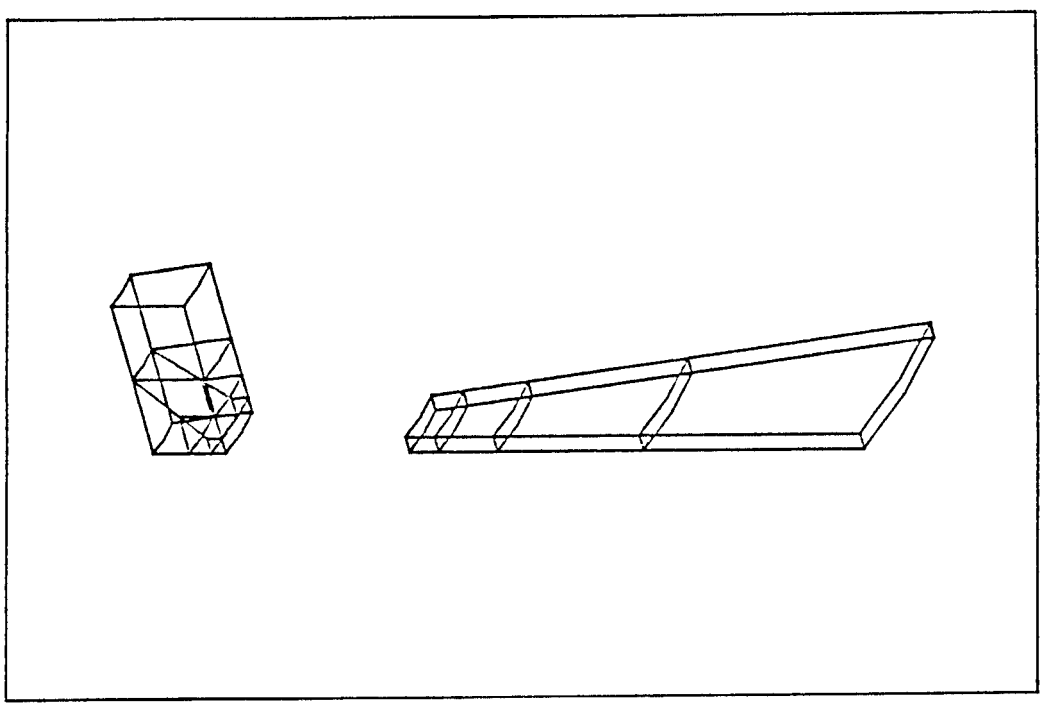


Fig. 7.2.1a Tube and disk fin heat exchanger - 2 GMR
Boundary Element Model

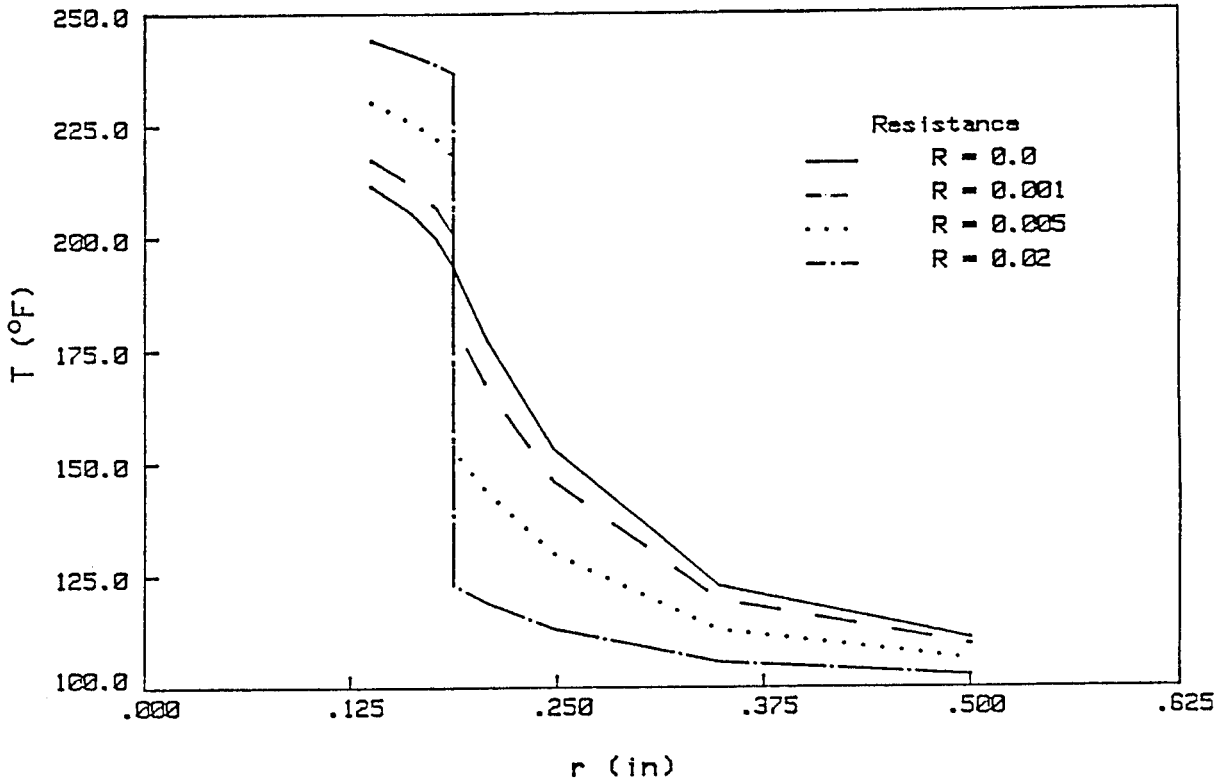


Fig. 7.2.1b Tube and disk fin heat exchanger Results

Heat Transfer with Hole Elements [42]:

A new element class has been developed for the heat transfer version of **GPBEST** . 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 modeling, a model problem consisting of three cylindrical holes in a cube was solved. The tube length and radius are, x respectively, 80% and 1% of the cube dimension. The location of the cavities in the cube is shown in Figure 7.2.2a 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 7.2.2b. The tube element analysis almost exactly reproduces the much more expensive results of explicit modeling. Comparison of results on the surface of the cube shows agreement of unknowns (either temperature or flux) to better than 1%.

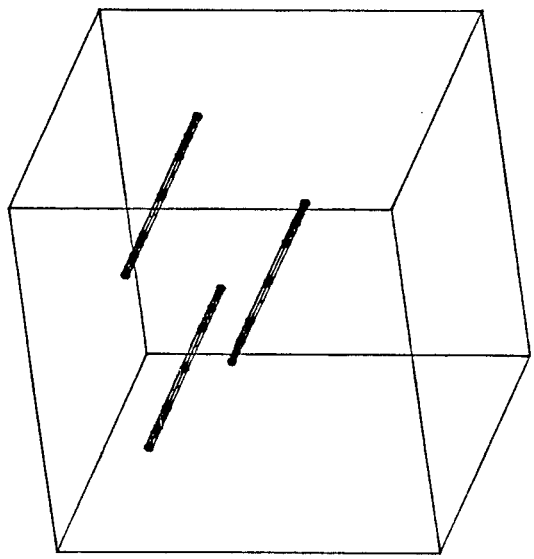


Fig. 7.2.2a Cube with embedded cylindrical holes
Boundary Element Model

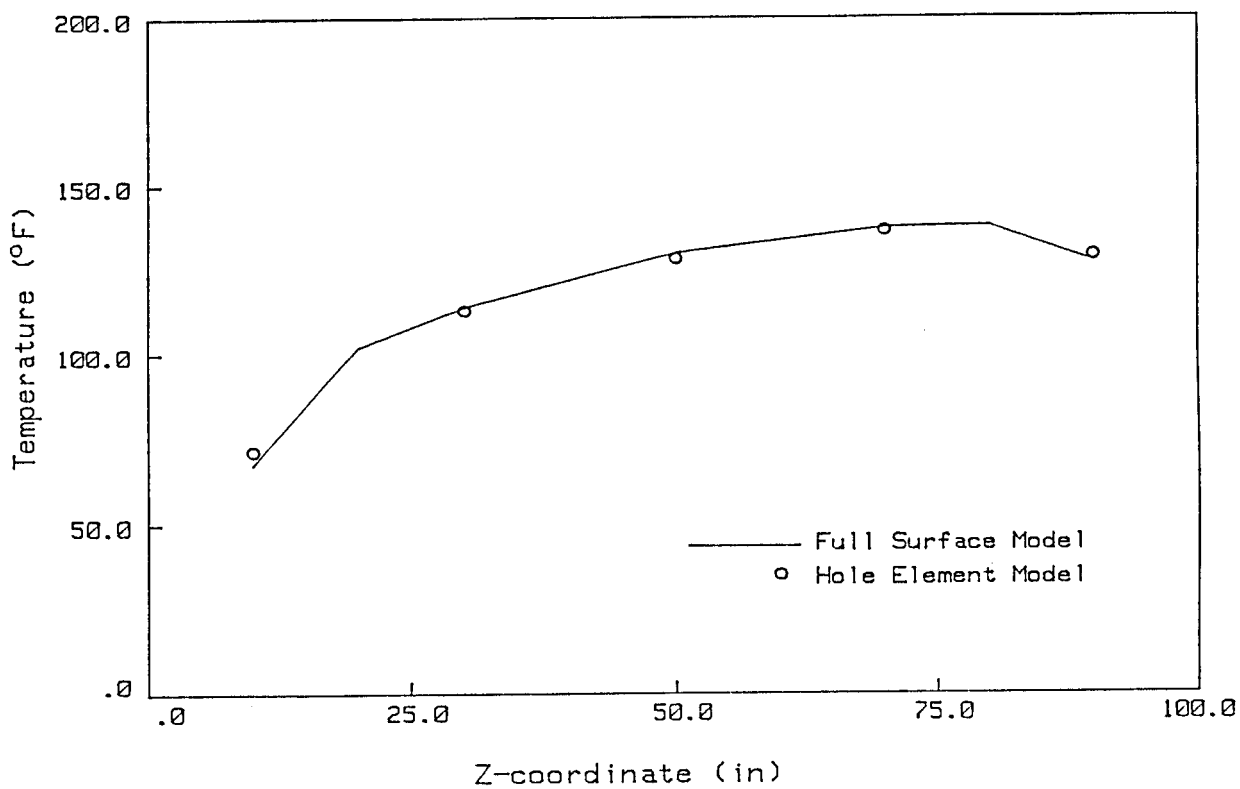


Fig. 7.2.2b Cube with embedded cylindrical holes
Temperature along central exclusion

Cooling of a Steel Sphere [78]:

A solid steel sphere of radius 1.5 in., initially at a uniform temperature of 100°F, is slowly cooled in 0° air by convection. The exact solution to this problem is given by Carslaw and Jaeger (1947) as

$$\theta(r,t) = \frac{2H\theta_o}{r}\sum_{n=1}^{\infty} e^{-c\alpha_n^2 t}\frac{R^2\alpha_n^2 + (RH-1)^2}{\alpha_n^2[r^2\alpha_n^2 + RH(RH-1)]} sin R\alpha_n sin r\alpha_n \tag{7.2.1}$$

where θ_o is the initial temperature, c is the diffusivity, R is the outer radius, and α_n are the roots of

$$R\alpha cot R\alpha + RH - 1 = 0. \tag{7.2.2}$$

Furthermore,

$$H = h/k \tag{7.2.3}$$

in which h is the convection film coefficient and k is the thermal conductivity. For positive H, all the roots of (7.2.2) are real. To complete the problem specification, let

$c = 0.0205\text{in.}^2/\text{sec.},$

$k = 25\text{in.} - \text{lb}/\text{sec.in.}^o\text{F},$

$h = 1.25\text{in.} - \text{lb}/\text{sec.in.}^{o2}\text{F}.$

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 7.2.3a - 7.2.3c 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 **GPBEST** , a quadratic variation of geometry is utilized for each surface element. The results from **GPBEST** , for a time step of 1.25 sec., are compared with the exact solution in Figure 7.2.3d 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.

An axisymmetric analysis was also conducted using the four element, nine model of Figure 7.2.3e. Results for a surface node are shown in Figure 7.2.3f. Again, excellent correlation is obtained.

Notice that abrupt changes in temperature not not imposed on the sphere at time zero. Consequently, the convergence difficulties, that plagued the previous unidirectional diffusion example during the first few time steps, are not apparent for the sphere with convection boundary conditions. However, if the film coefficient is increased to a large value, then the same small time difficulties will return.

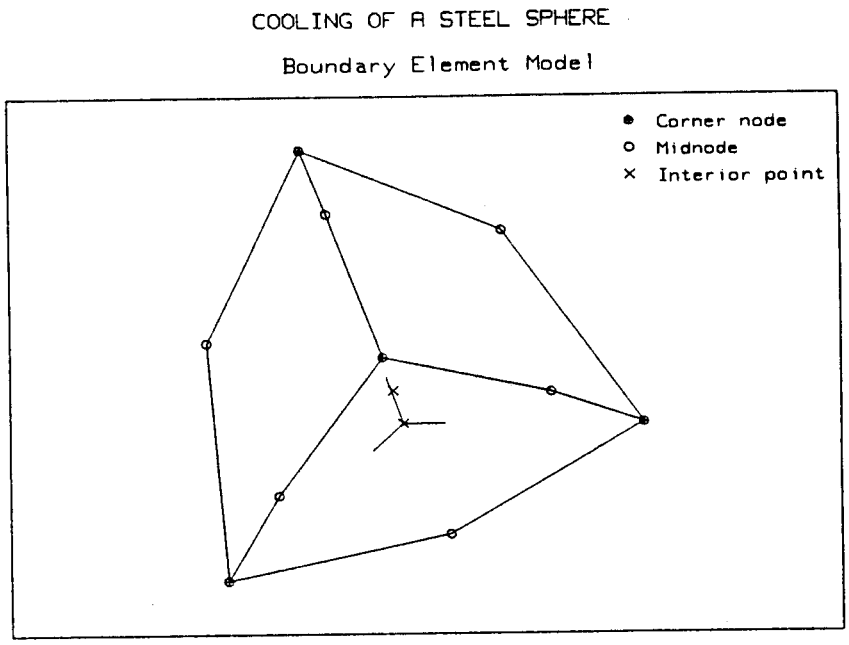


Figure 7.2.3a

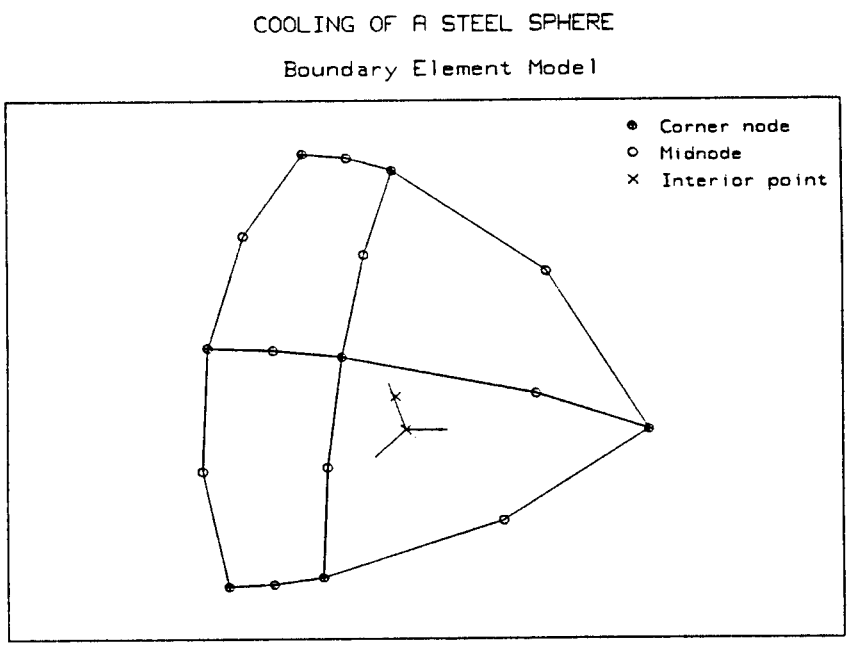


Figure 7.2.3b

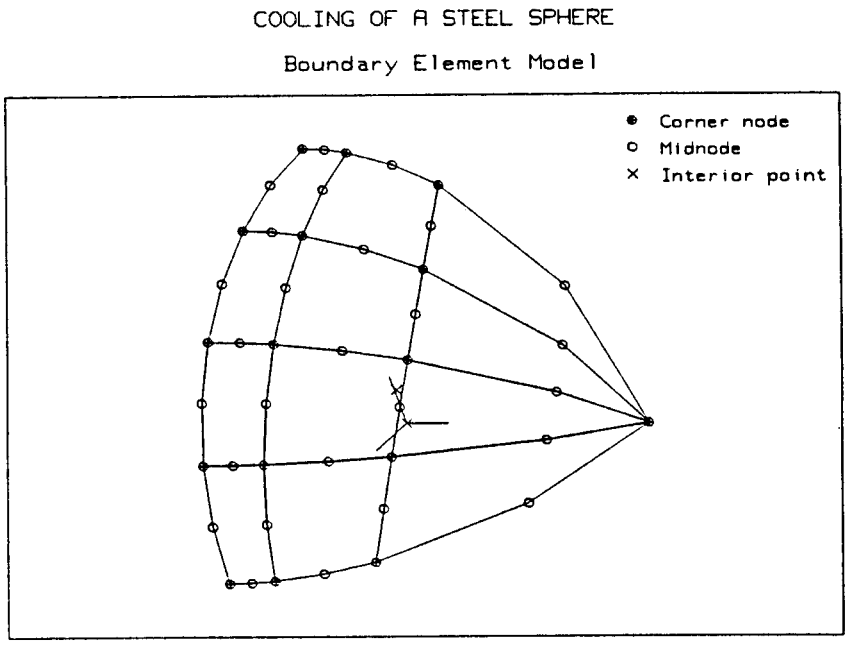


Figure 7.2.3c

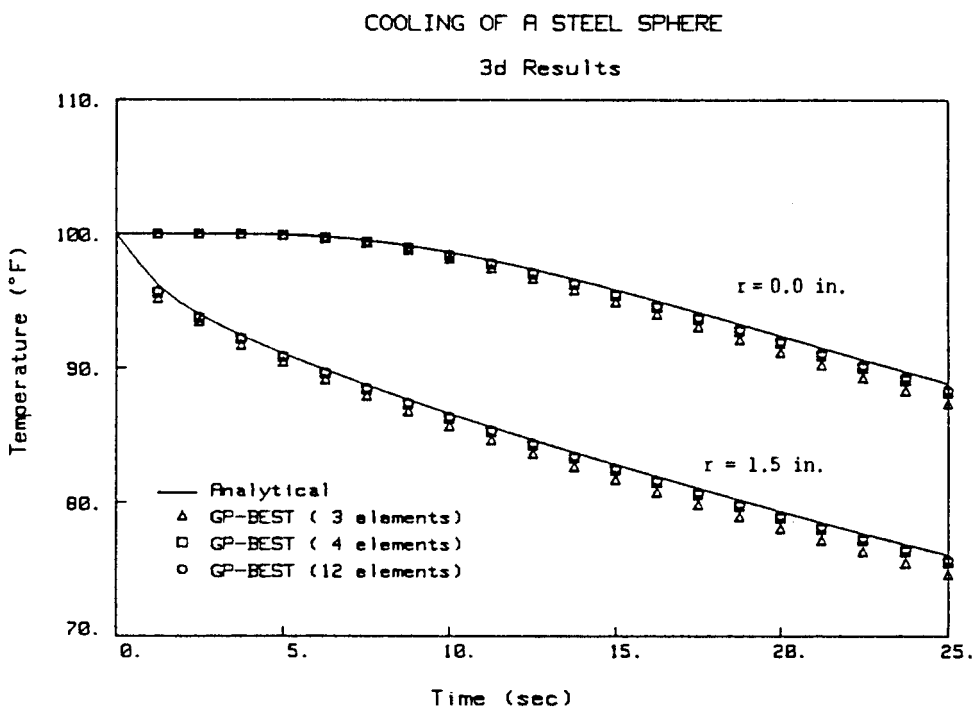


Figure 7.2.3d

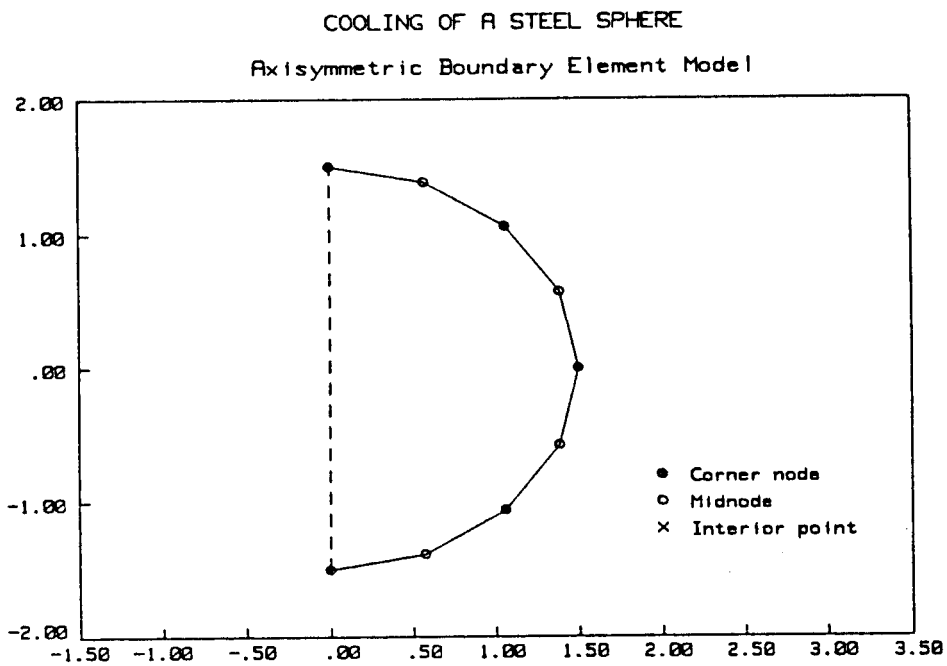


Figure 7.2.3e

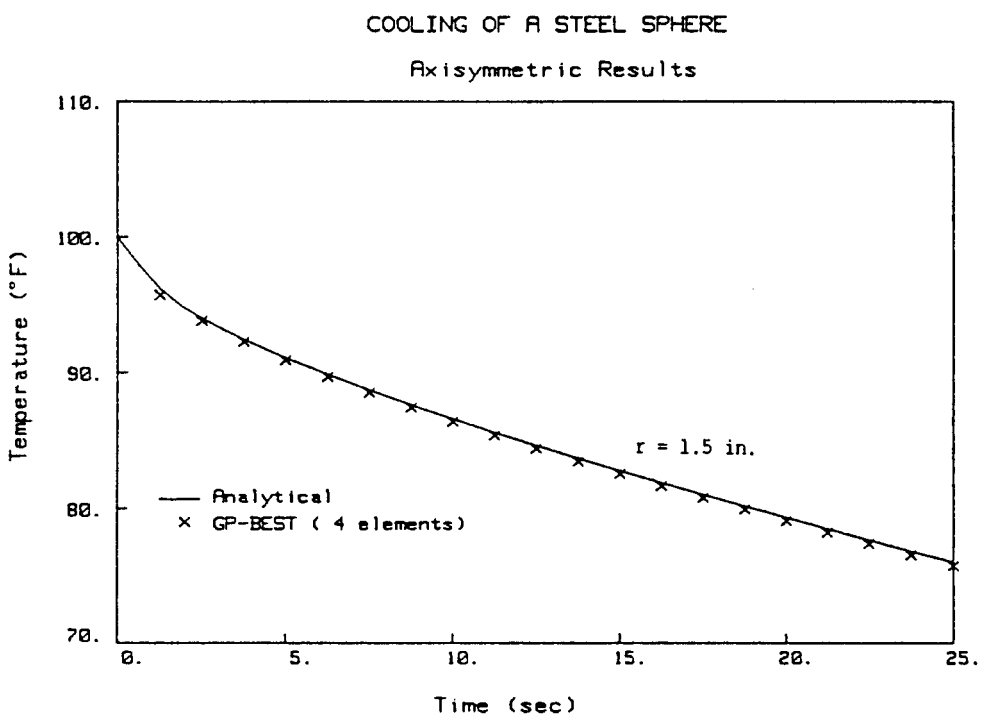


Figure 7.2.3f

Prolate Spheroidal Solid [78]:

As another diffusion example, the transient heating of a prolate spheroidal solid is examined. The analytical solution was obtained by Haji-Sheikh and Sparrow [80], while Zienkiewicz [16] presented the results of a finite element analysis using isoparametric solid elements. The axisymmetric surface geometry is defined by

$$R = L_1 cos\phi$$

$$Z = L_2 sin\phi$$

where $L_1 = 1.0, L_2 = 2.0$ and ϕ is the counterclockwise angle from the r-axis in the r-z plane. Initially, the body is at zero temperature, but then, at time $t - 0^+$, a unit temperature is specified all along the outer surface. Also, unit diffusivity is assumed.

The four element, nine node axisymmetric boundary element mesh is displayed in figure 7.2.4a. Meanwhile, analytical, finite element, and **GPBEST** solutions are compared, in the graph of Figure 7.2.4b, for a point at the origin. The boundary element results, for $\Delta T = 0.05$, are quite good after the initial few time steps. Again, the sudden imposition of temperature causes problems at the beginning of the diffusive process. If more accuracy is desired at these early times, a finer spatial and temporal discretization is in order.

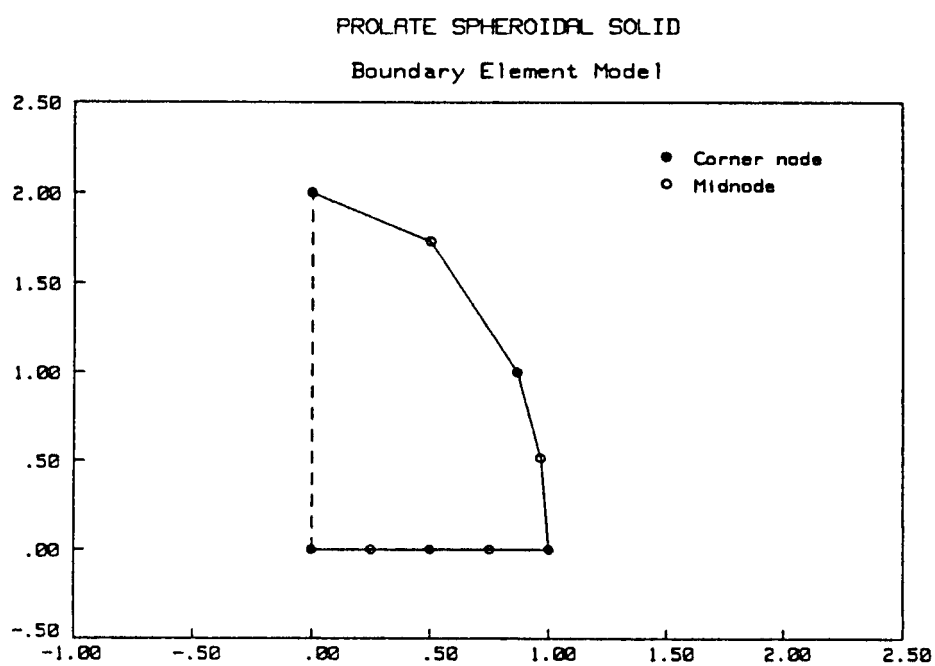


Figure 7.2.4a

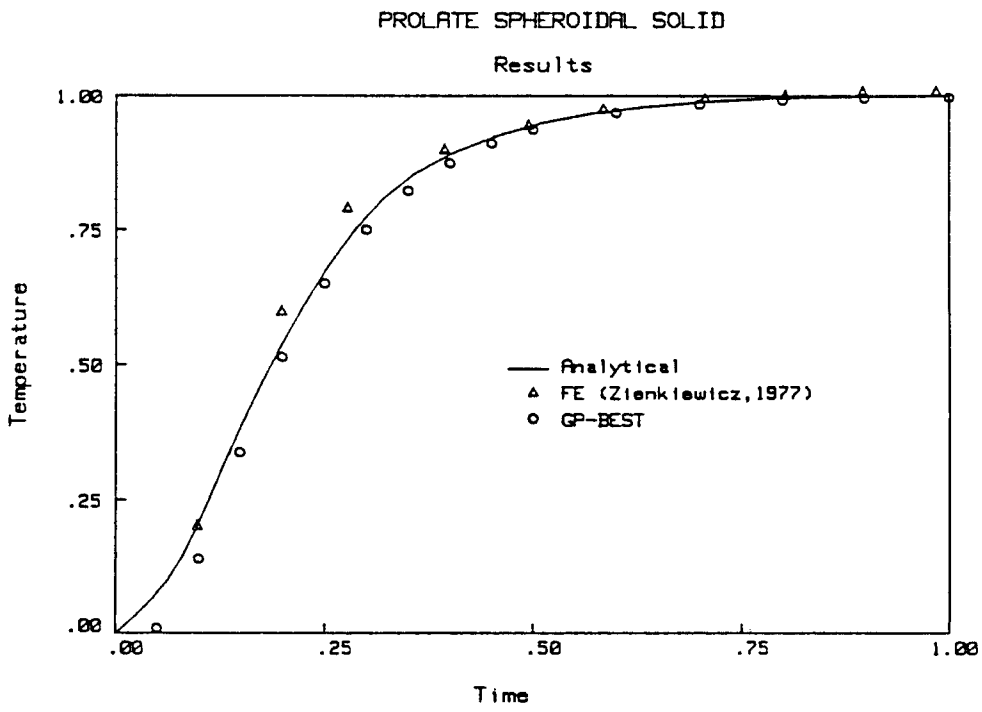

PROLATE SPHEROIDAL SOLID
Results
1.00
.75
.50
.25
.00
Temperature
.00
.25
.50
.75
1.00
Time
Analytical
FE (Zienkiewicz,1977)
GP-BEST

Figure 7.2.4b

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 1.0in., a thickness of 0.25in., and an overall lenght of 5.5in. Meanwhile, the mold itself is quite simple. It consists of two steel components, a core and a cavity, with combined outer dimensions of 15in. $\times$ 20in. $\times$ 20in. Several 0.25in. dimeter cooling lines, running parallel to the lenght of the tank, are provided in both components.

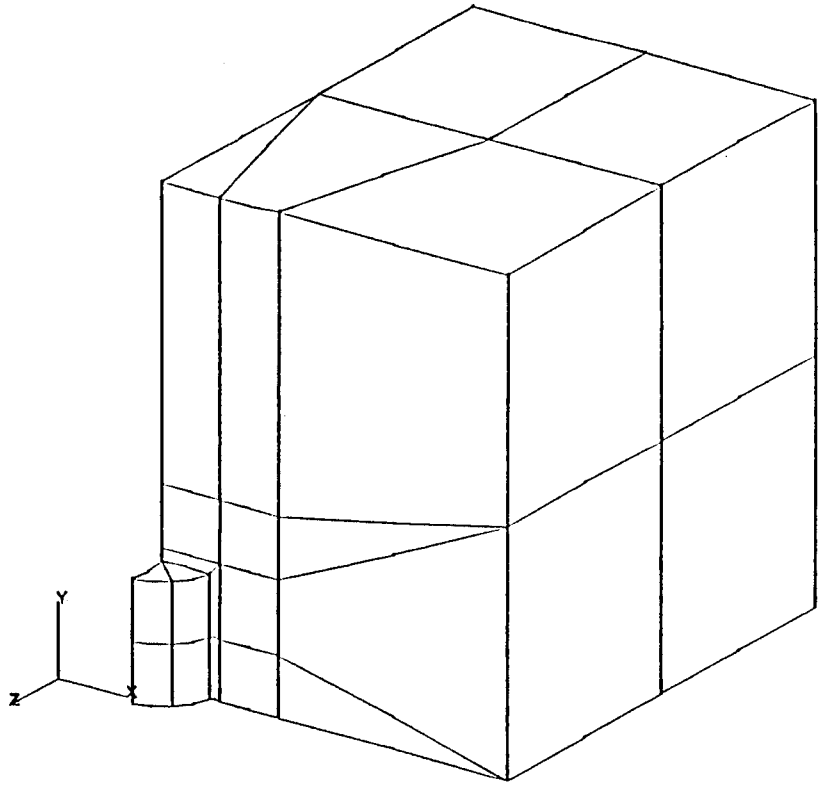
The boundary element model of the positive quadrant of the mold is depicted in Figure 7.2.5a. In that diagram, the core and cavity are shown individually as separate GMR's, which are viewed from different vantage points. Thirty-two quadratic surface elements model the core, while the cavity is discretized 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 7.2.5b.

Convective heat transfer is assumed on all exposed surfaces. Specifically, on the surface of the tank h = 25in.-lb./in.^{2}sec$^\circ$ F and θ_{amb} = 450°, while for the cooling lines h = 50in.-lb./in.^{2}sec$^\circ$ F and θ_{amb} = 120°. A much lower convection coefficient (h = 0.1in.-lb./in.^{2}sec$^\circ$ F) is assumed on the outer mold surfaces which are in contact with 100°F ambient air. Also, the thermal resistance of 0.01in.^{2}sec.$^\circ$F/in.-lb. is imposed across the imperfect core to cavity interface. The conductivity of steel is taken as 5.8in.-lb./sec.in.$^\circ$F.

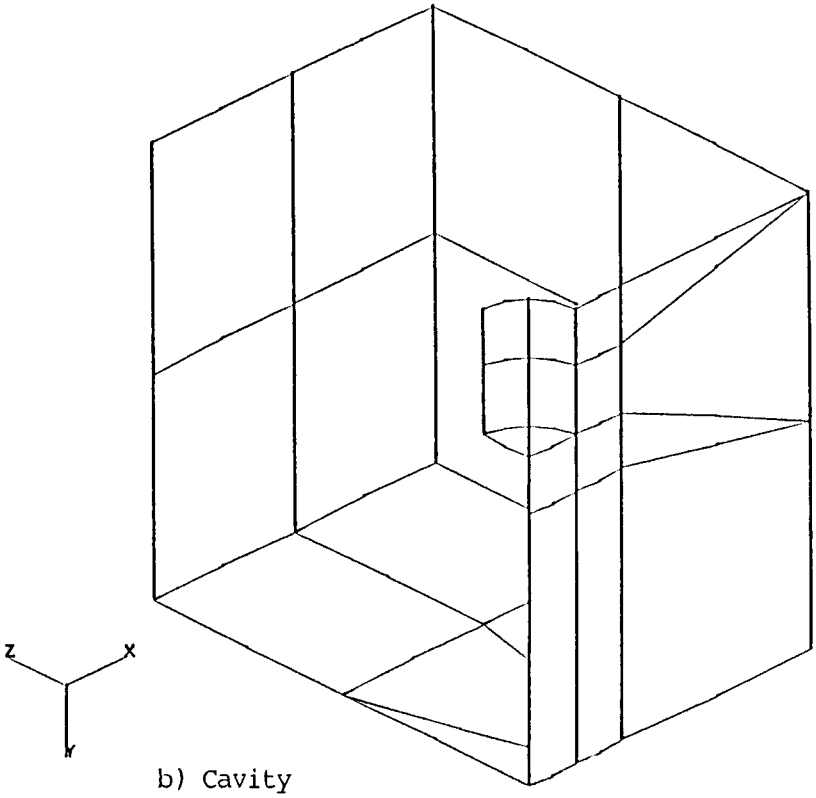
The resulting steady-state temperature contours are displayed in Figure 7.2.5c. Notice that the highest temperatures occur near the crown of the tank, while the largest thermal gradients appear near the foot. Also, as is evident from Figure 7.2.5d, there are significant temperature differences between 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.

Figure 7.2.5a Injection Mold - Boundary Element Model



a) Core



b) Cavity

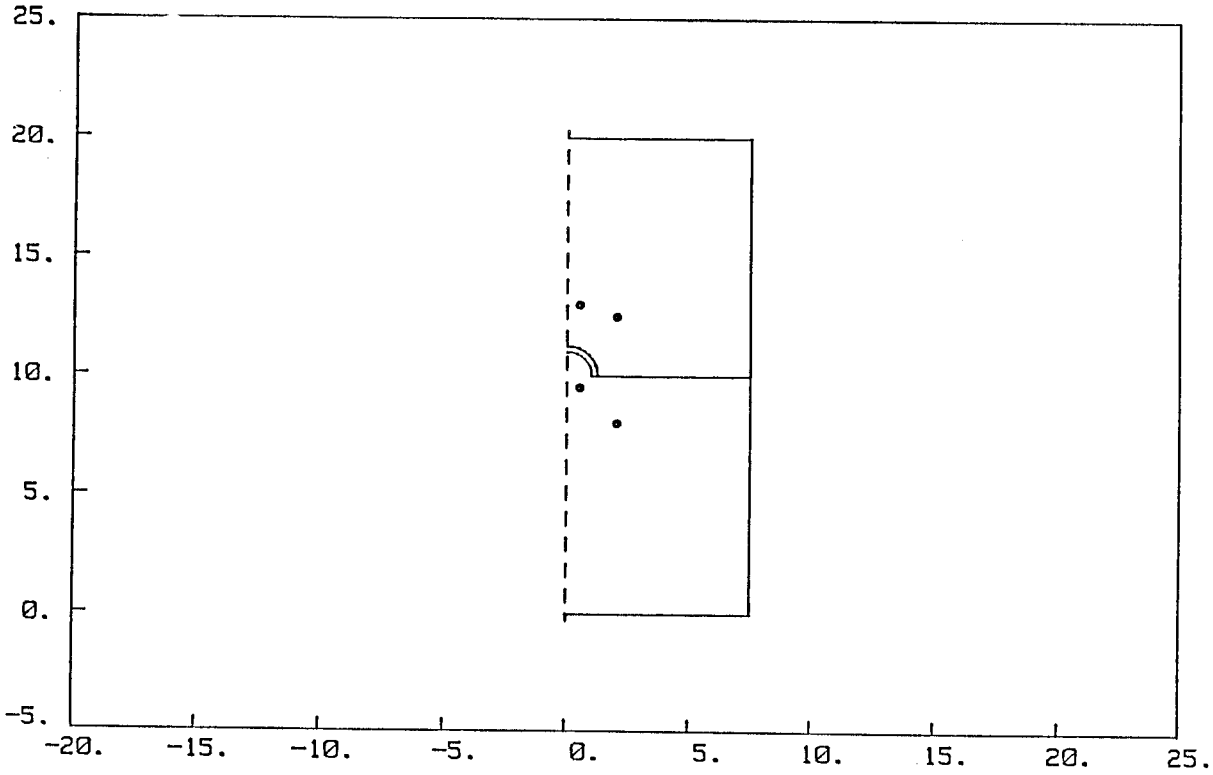


Figure 7.2.5b Cross-section - Injection Mold

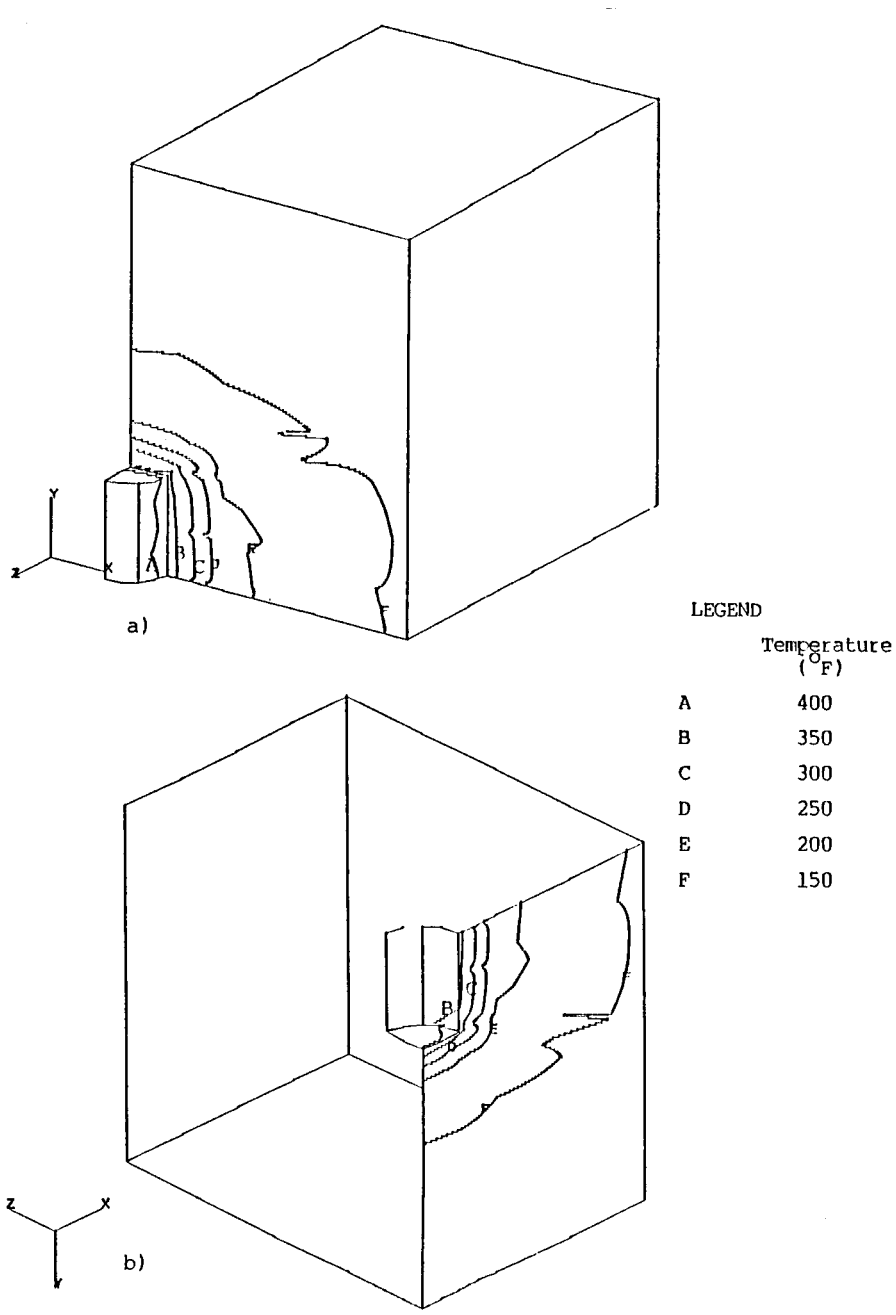


Figure 7.2.5c Temperature Contours - Injection Mold

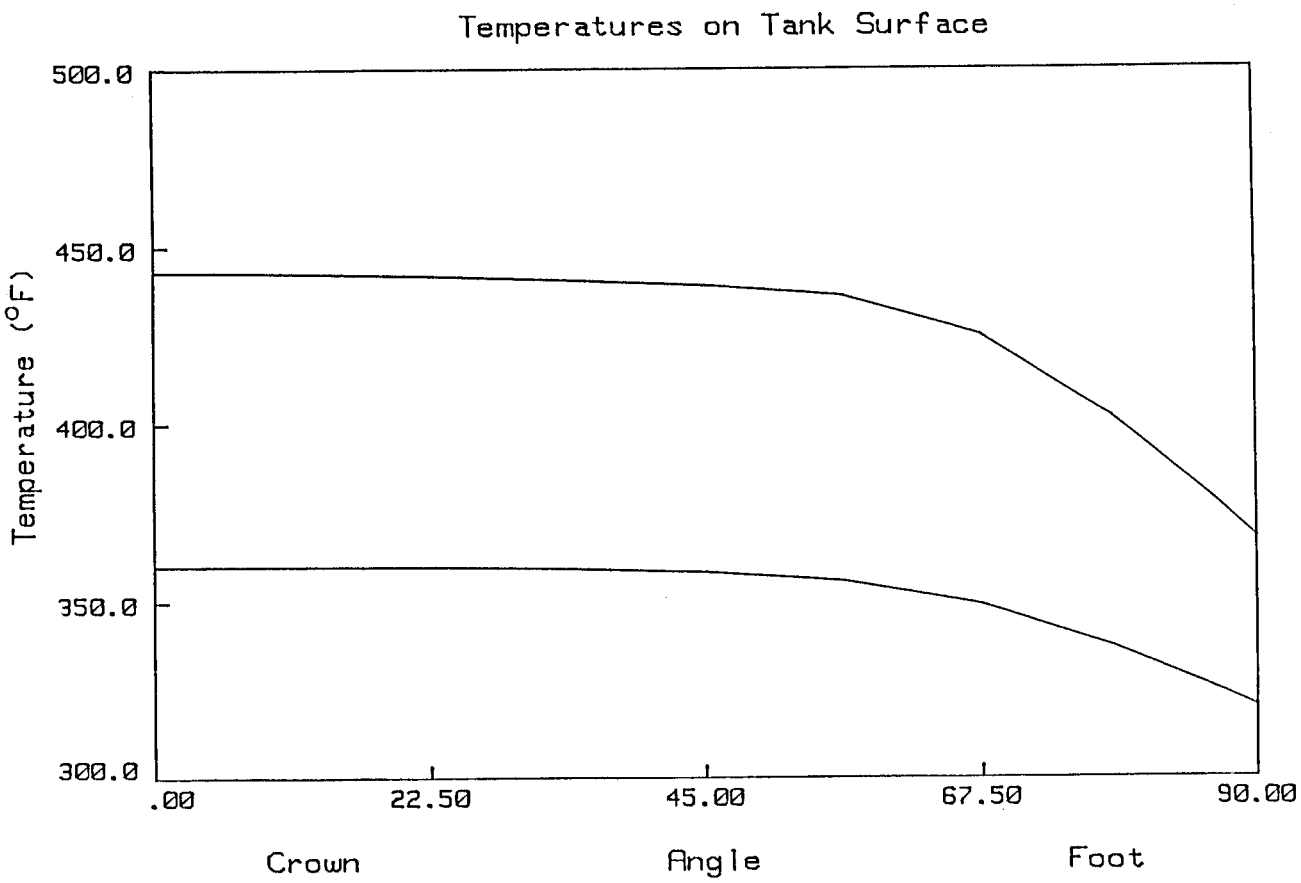


Figure 7.2.5d Temperatures on Tank Surface - Injection Mold

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 7.2.6a. 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 7.2.6a, 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 7.2.6b. 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^{2}C is specified. The tip of the blade is completely insulated. Additionally, the thermal material properties are specified as follows :

K = 0.0216 W/mmC
C = 6.14 mm^2/sec.

The resulting steady-state temperature distribution is presented in Figure 7.2.6c. 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 7.2.6d. 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 7.2.6e-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.

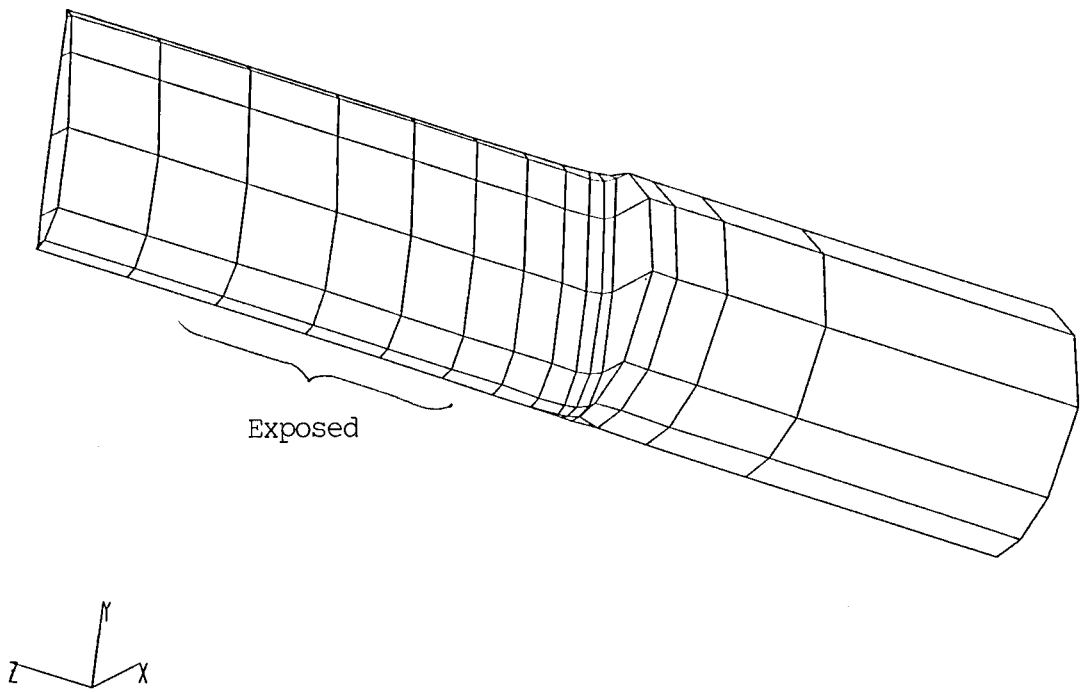


Fig. 7.2.6a Thermal Response of a Turbine Blade
Boundary Element Model

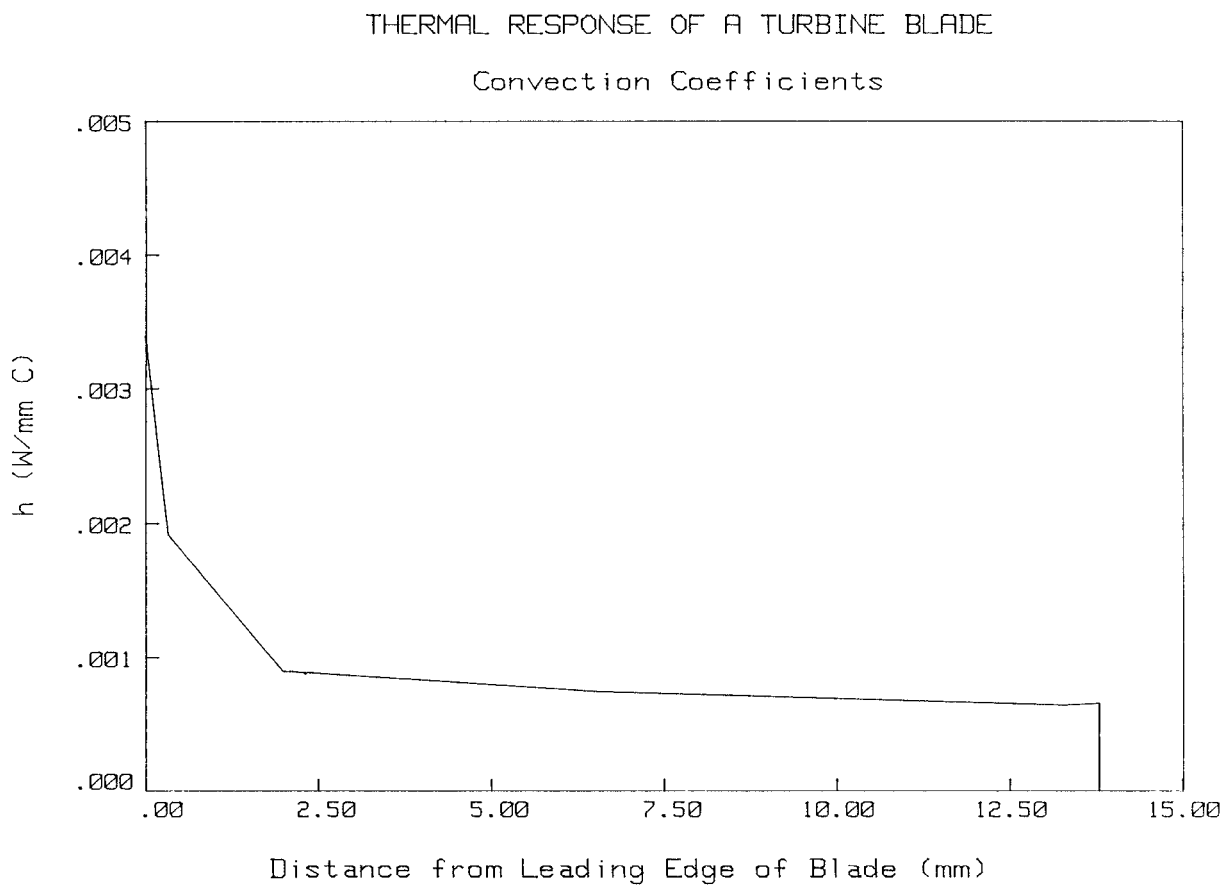


Fig. 7.2.6b Thermal Response of a Turbine Blade
Convective Coefficients

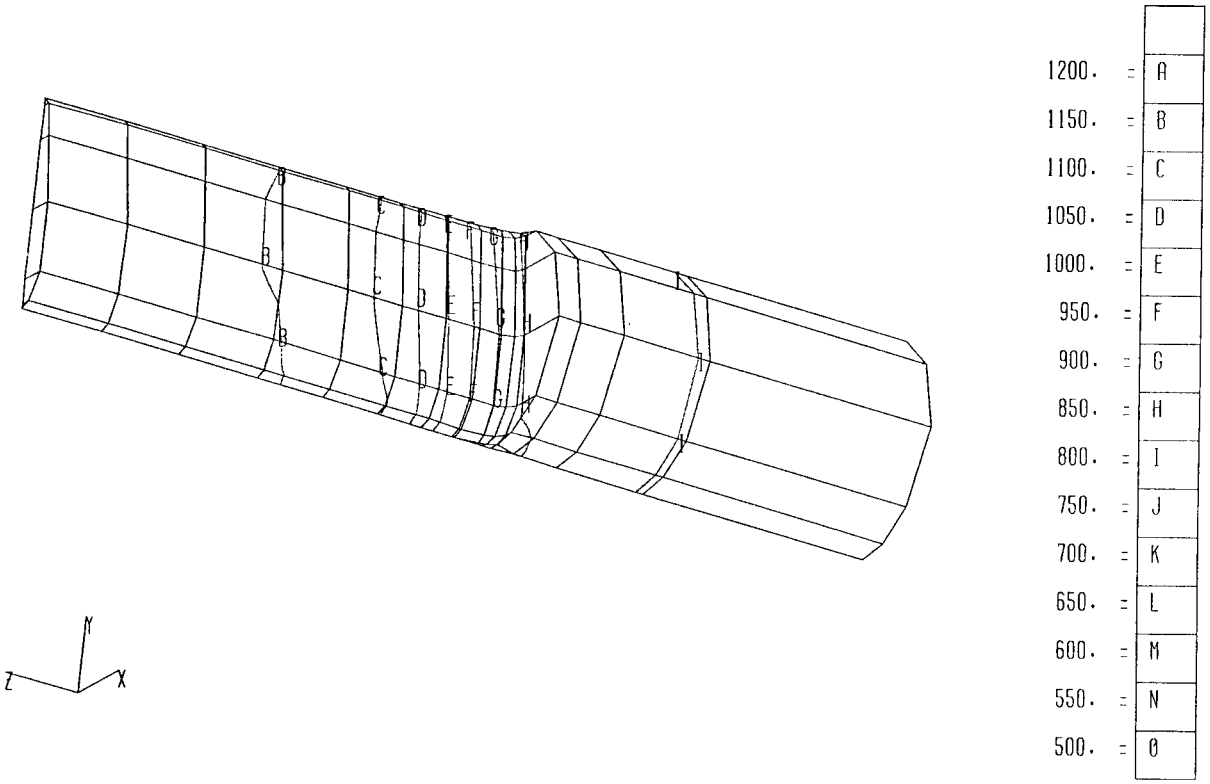


Fig. 7.2.6c Thermal Response of a Turbine Blade
Steady-state Temperatures

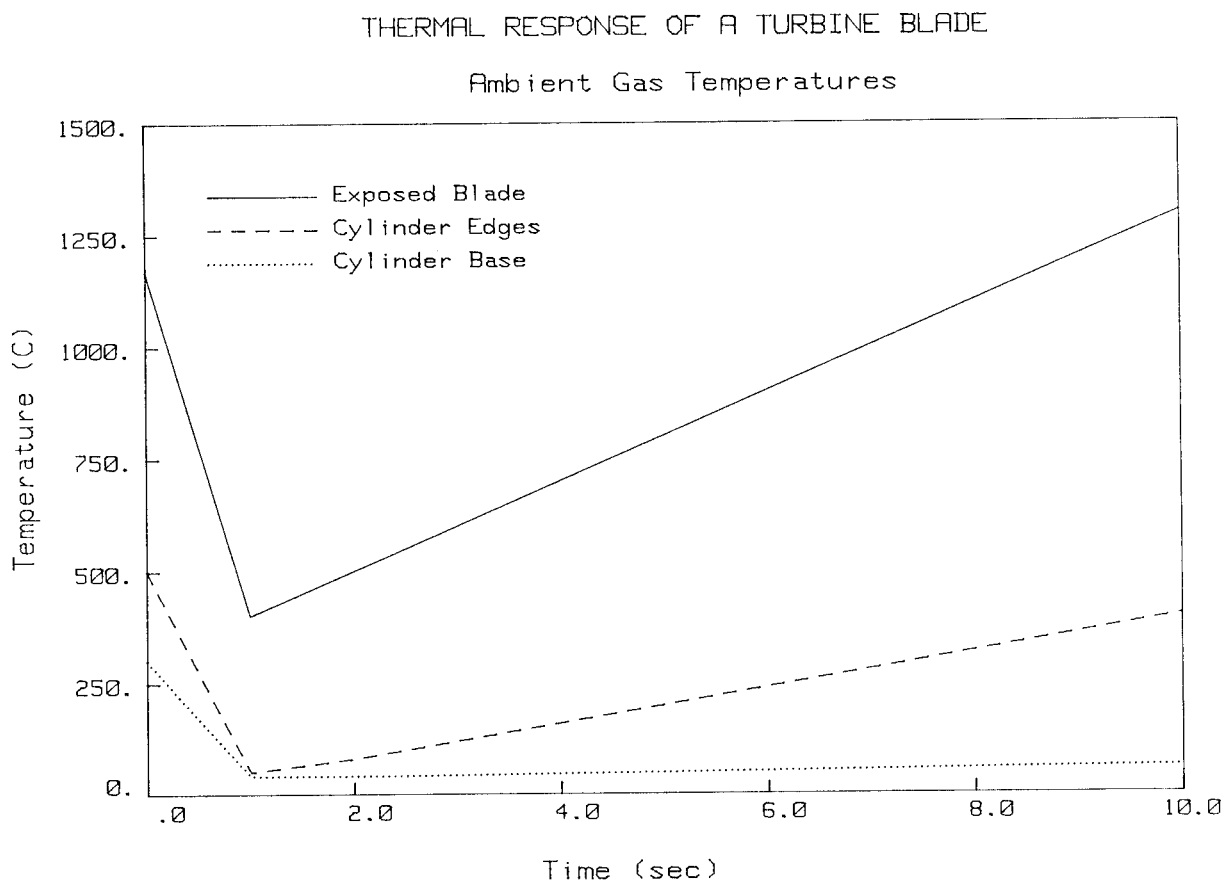


Fig. 7.2.6d Thermal Response of a Turbine Blade
Ambient Gas Temperatures

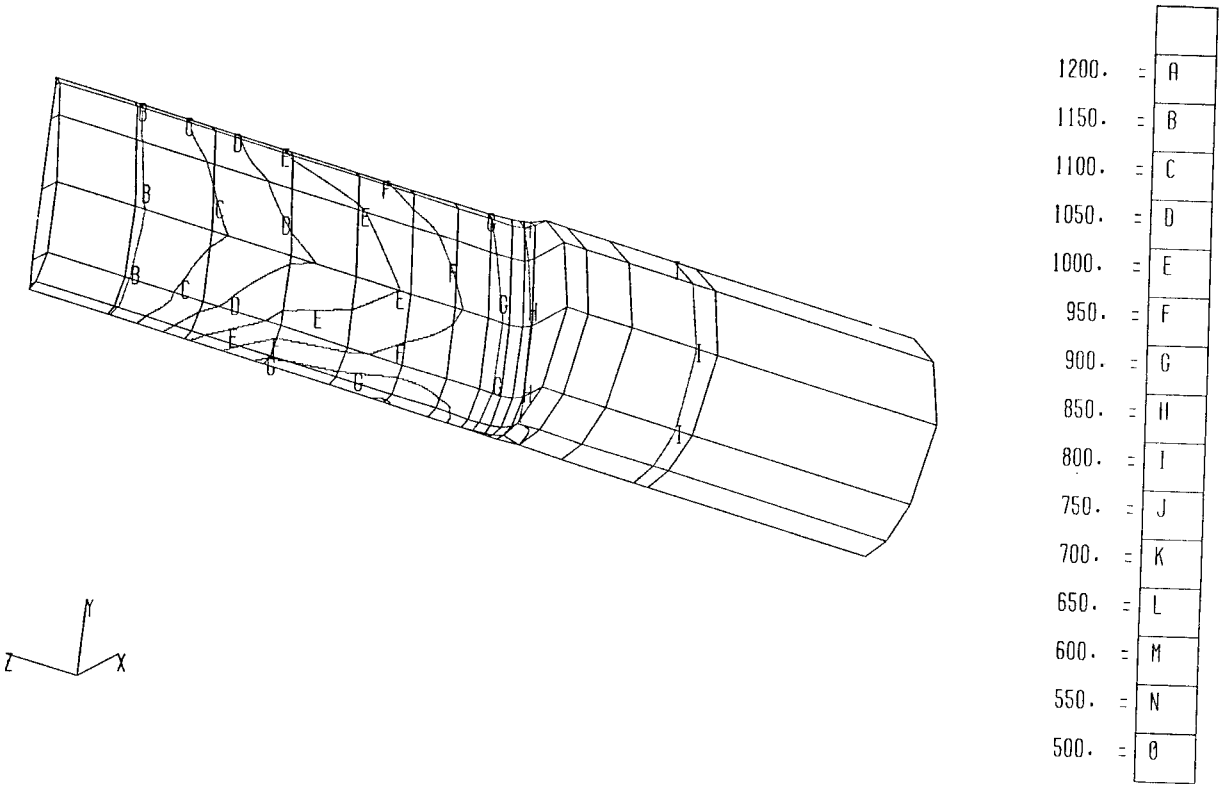


Fig. 7.2.6e Thermal Response of a Turbine Blade
Transient Temperatures at t = 1 sec.

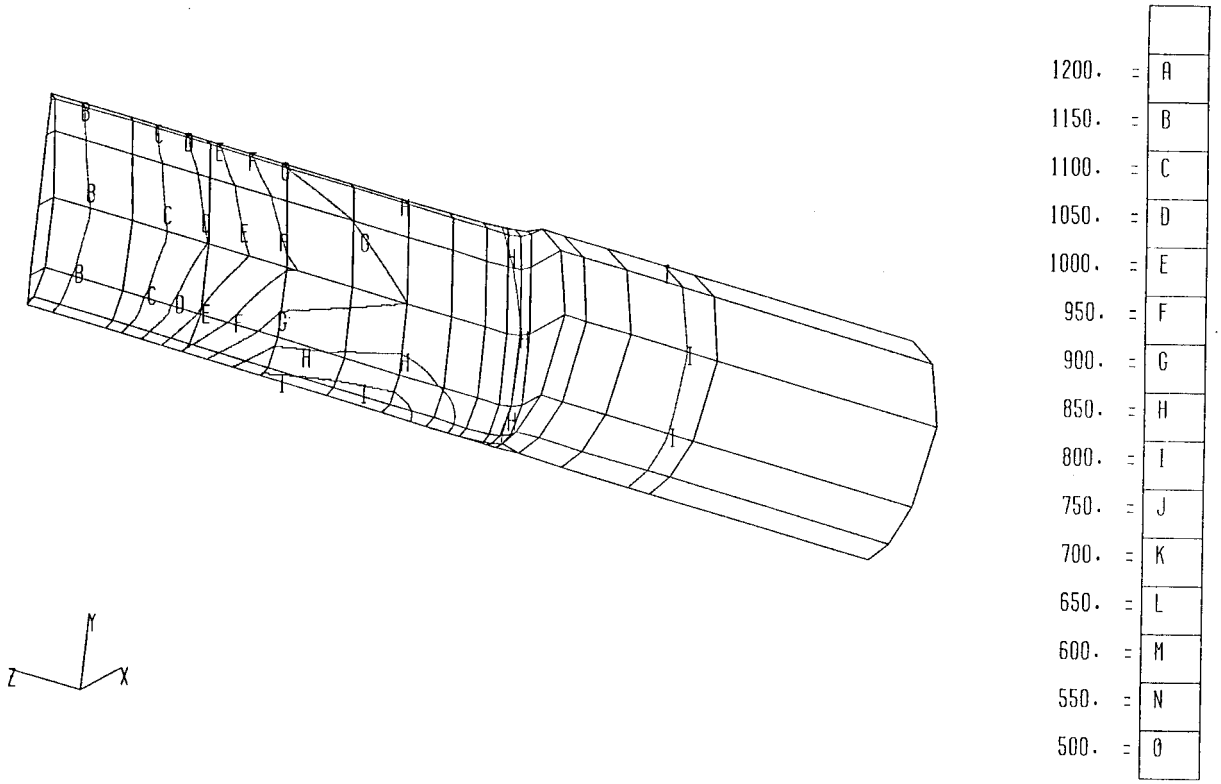


Fig. 7.2.6f Thermal Response of a Turbine Blade
Transient Temperatures at t = 2 sec.

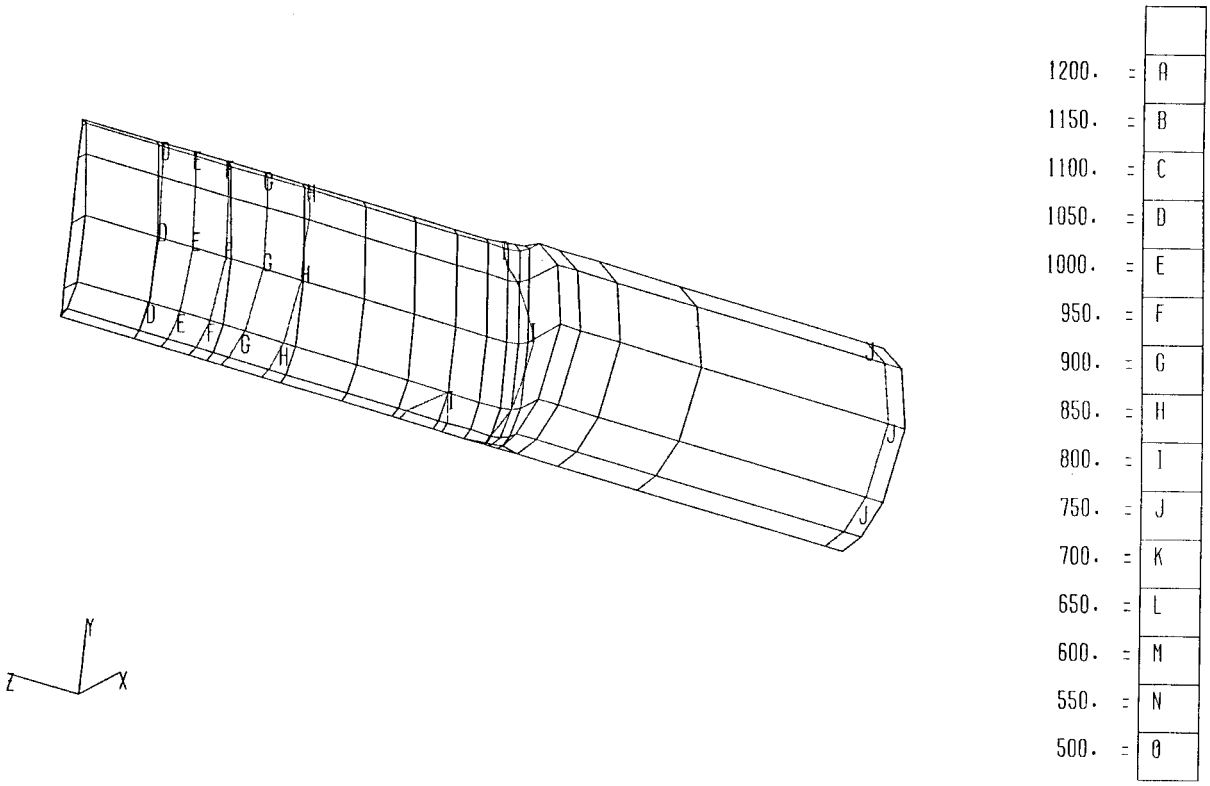


Fig. 7.2.6g Thermal Response of a Turbine Blade
Transient Temperatures at t = 5 sec.

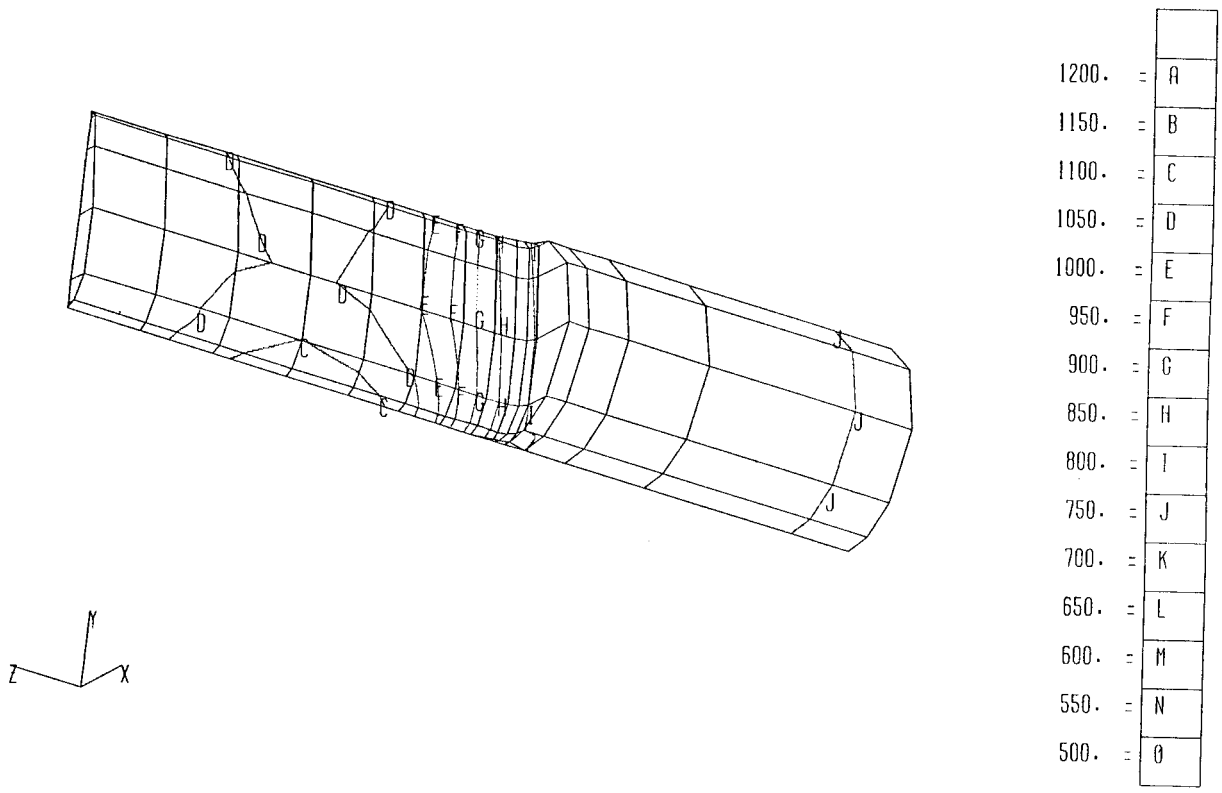


Fig. 7.2.6h Thermal Response of a Turbine Blade
Transient Temperatures at t = 10 sec.

7.3 PERIODIC ACOUSTIC ANALYSIS

Acoustic radiation from a sphere:

Figure 7.3.1a shows the fine BEM mesh for one half of the sphere which contains 12 elements per octant. The coarse mesh used had only 3 elements per octant. Figure 7.3.1b shows the imaginary part of developed acoustic pressure on the surface due to a uniform radiating flux compared with the exact analytical solution. Excellent agreeement can be observed for this mesh even at high frequency.

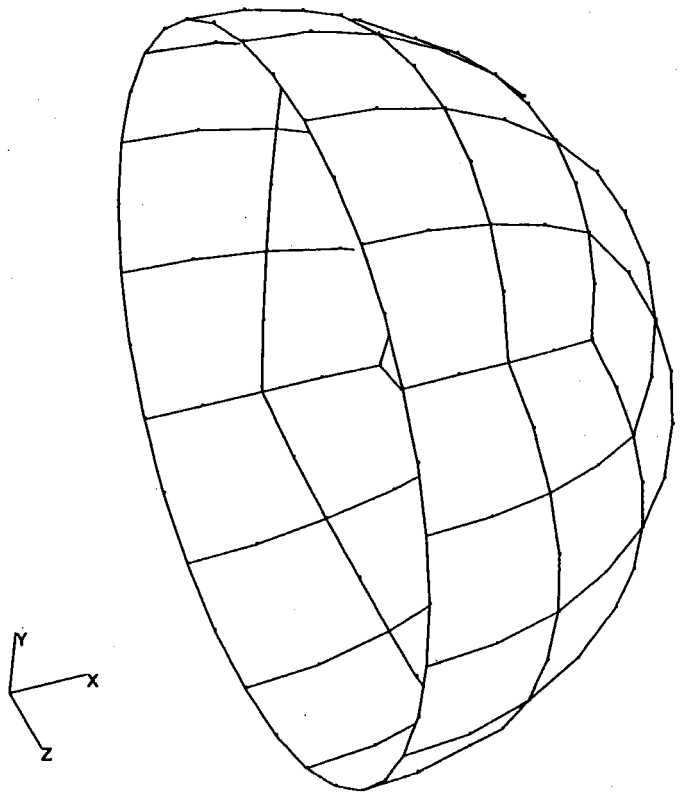


Fig. 7.3.1a BEM fine mesh for hemisphere

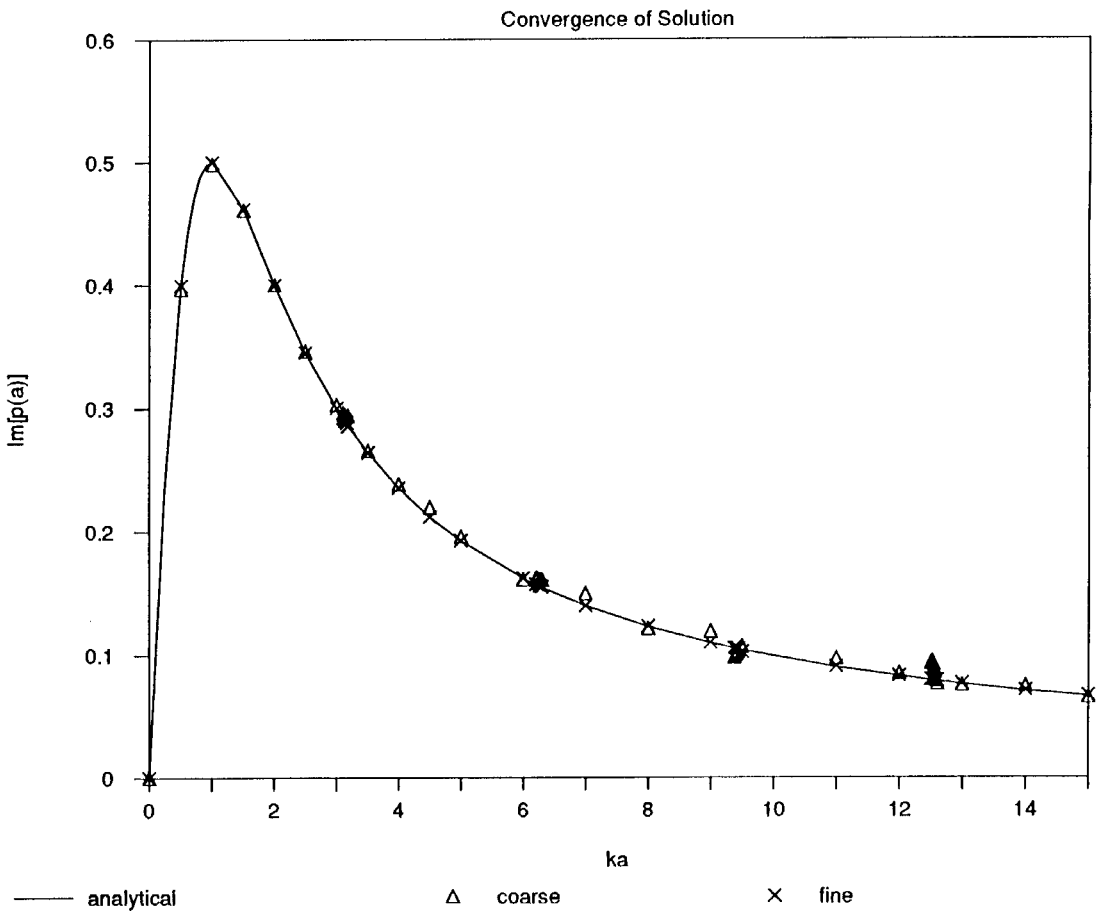


Fig. 7.3.1b Imaginary part of pressure on surface of cylinder

Acoustic radiation from a uniformly pulsating cylinder with rigid ends:

Figure 7.3.2a shows the problem and three meshes used for one quarter of this problem. Figure 7.3.2b shows the far field responses for the three meshes at frequencies of KA = 8, 2 and 5 respectively. These results are shown in a normalised polar plot to show the far field response highlighting the effects of discontinuous boundary conditions at the edge of the cylinder. It is interesting to note that at higher frequencies the finer mesh produce the convergent results.

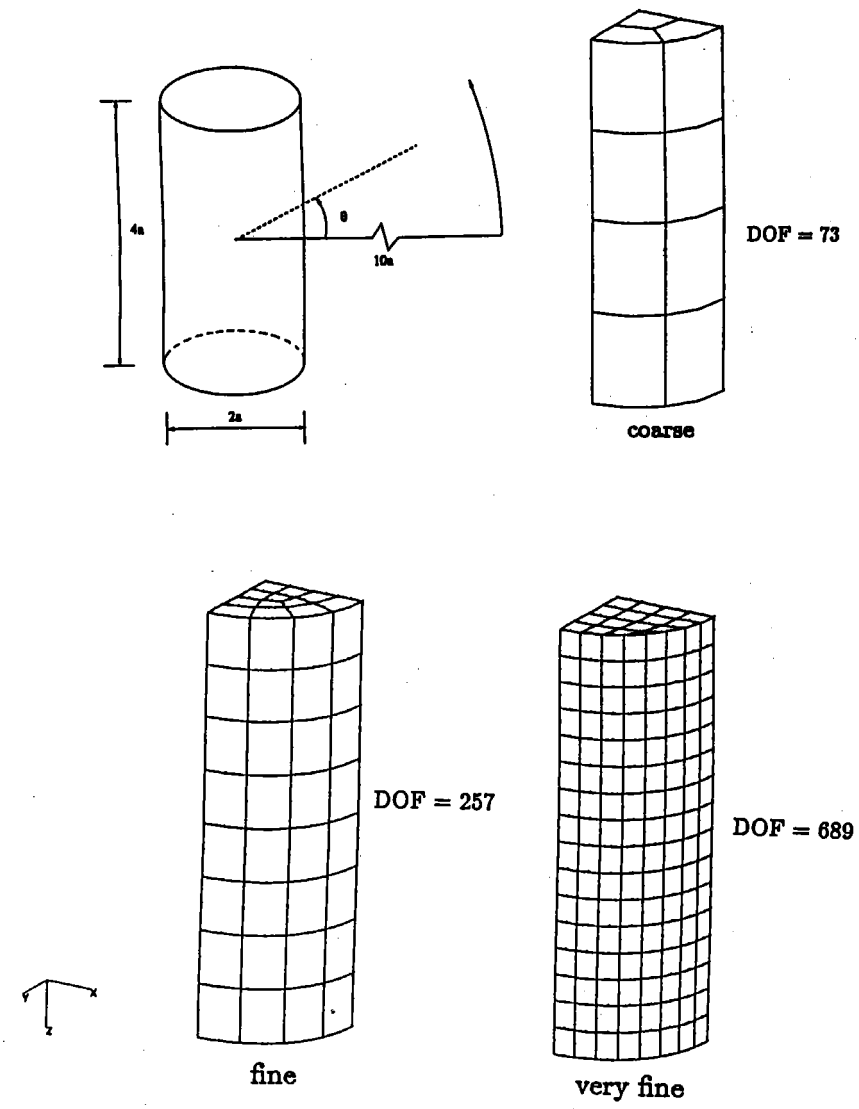


Fig. 7.3.2a Pulsating cylinder with stationary edges

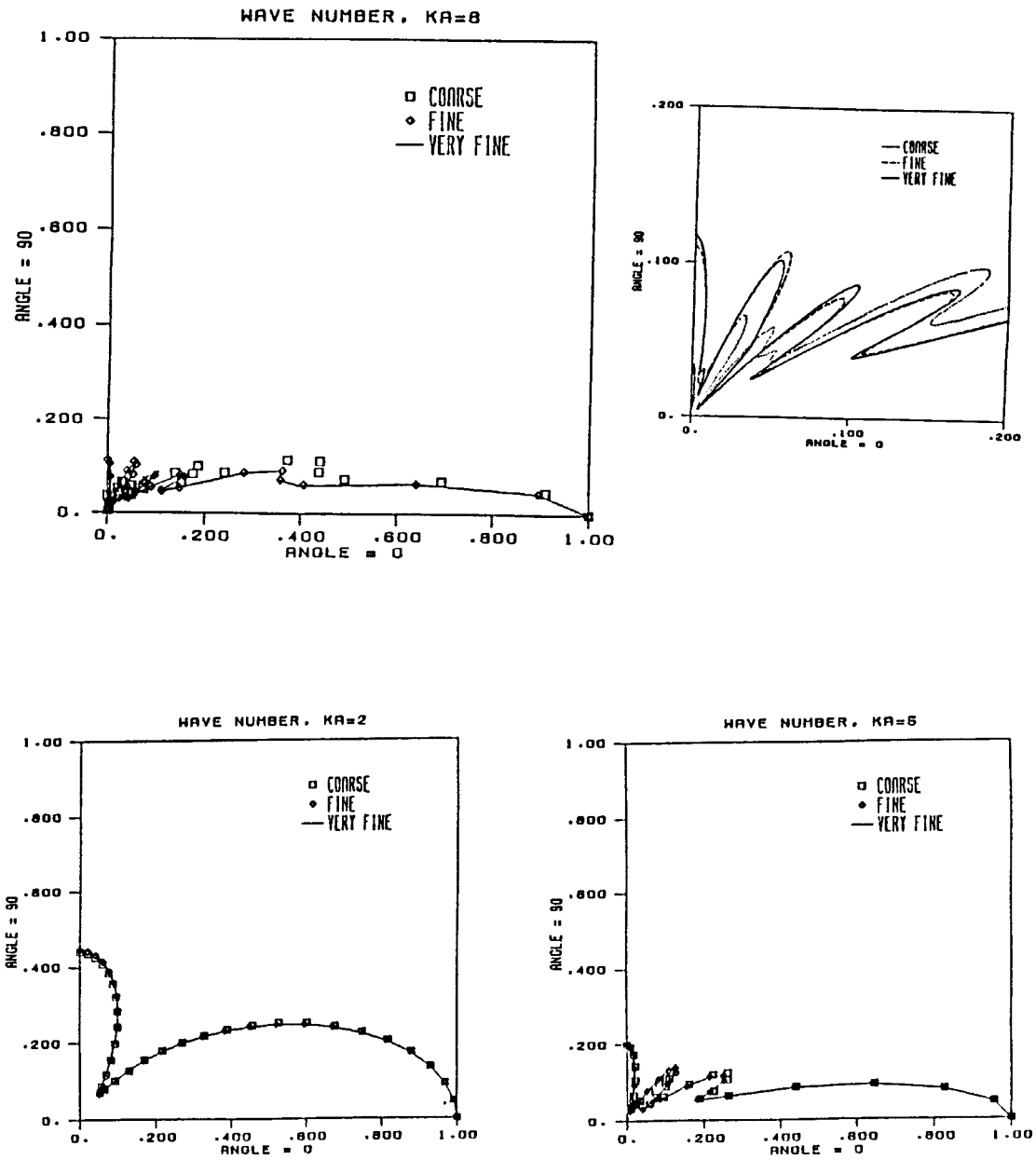


Fig. 7.3.2b Polar plot of normalized far field pressure

7.4 PERIODIC DYNAMIC ANALYSIS

Dynamic analysis of a buried foundation [84]:

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.

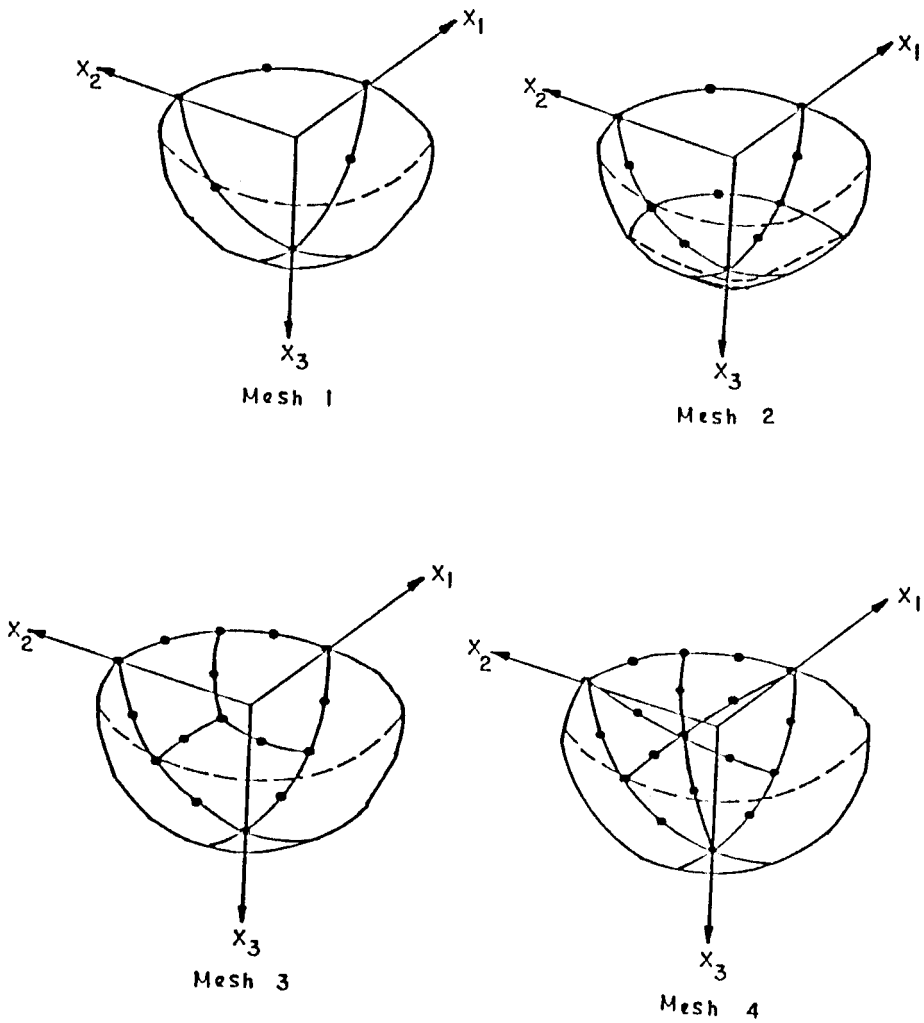


Fig. 7.4.3a Discretizations of hemispherical foundation-soil interfaces

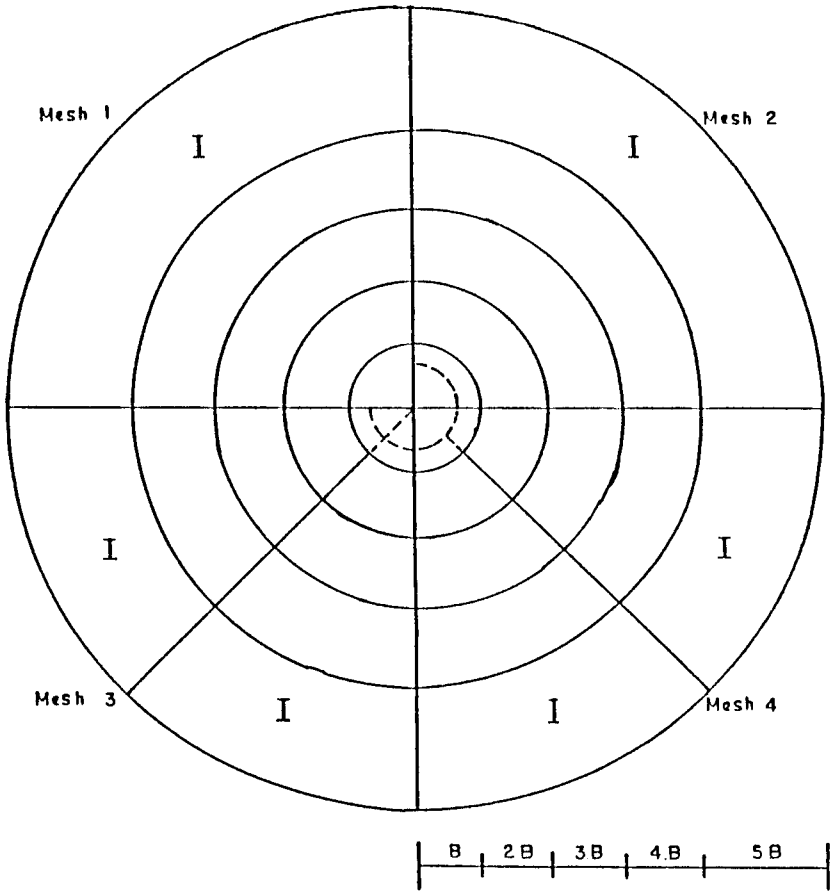


Fig. 7.4.3b Discretizations of the free field for a hemispherical foundation

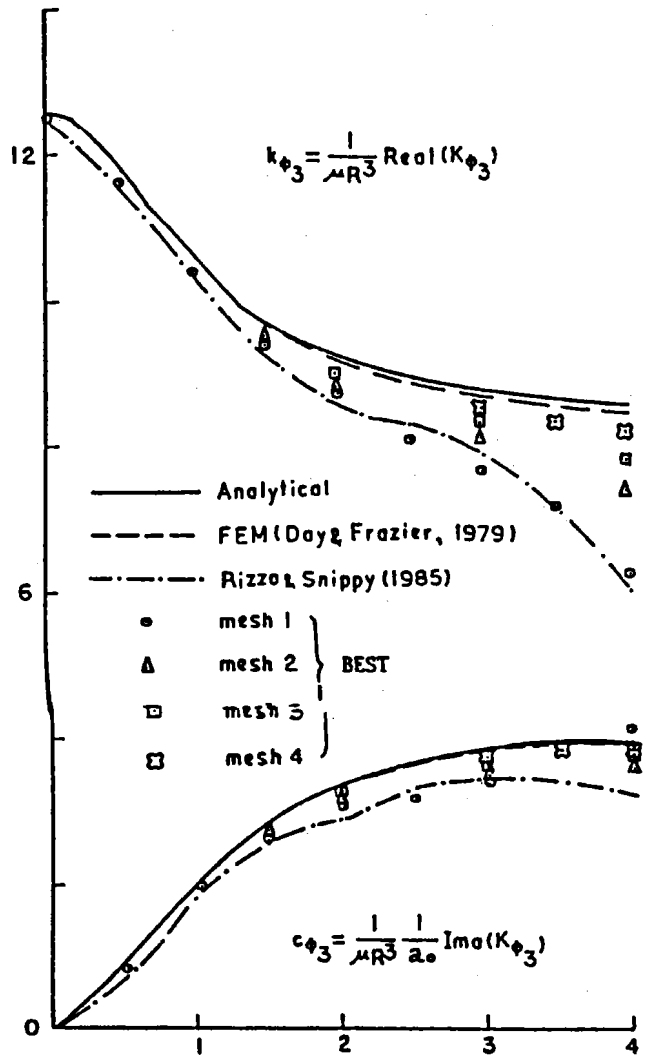


Fig. 7.4.3c Comparison of torsional stiffness coefficients for a hemispherical foundation

Application to the vibration isolation problem [41]:

In this example, results using **GPBEST** for a number of vibration isolation cases are compared with the actual field observation of Woods [85]. He carried out a series of field tests 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 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 ρ = 104 lb/ft^3, water content w = 7%, void ratio e = 0.61, pressure wave velocity v_p = 940 ft/sec, and the Rayleigh wave velocity v_R = 413 ft/sec rests on a layer of sandy silt (ML) with dry density ρ = 91 lb/ft^3, water content w = 23 %, void ratio e = 0.68, and pressure wave velocity v_p = 1750 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 ν, and mass density ρ. The material properties of the top layer are basically determined by the relations among the pressure, shear and the Rayleigh wave velocities. Accordingly, the values are taken as shear modulus G = 647200 lb/ft^2, Poisson's ratio ν = 0.35, and the mass density ρ = 3.25 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 lb/ft^2, Poisson's ratio ν = 0.35, and the mass density ρ = 2.84 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^2 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.

Periodic Dynamic Analysis

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 **GPBEST** in the areas of transient dynamics as well as those occurring in the areas of periodic and transient acoustics.

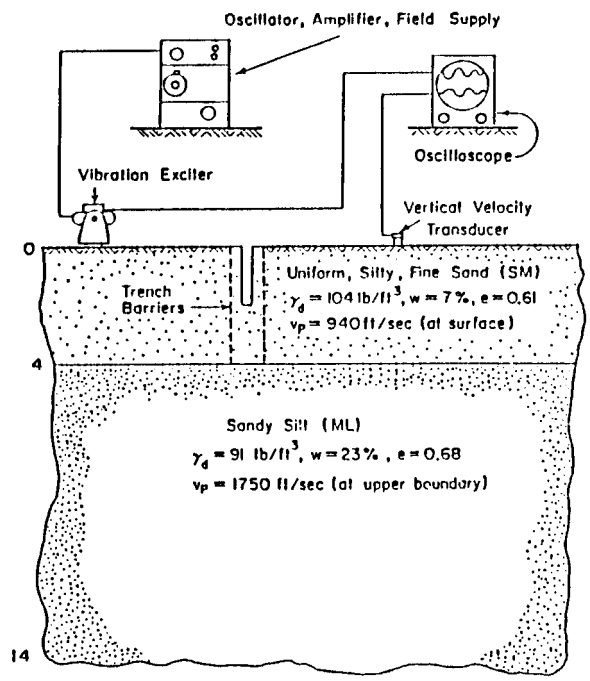


Fig. 7.4.4a Field site soil properties and schematic of instrumentation

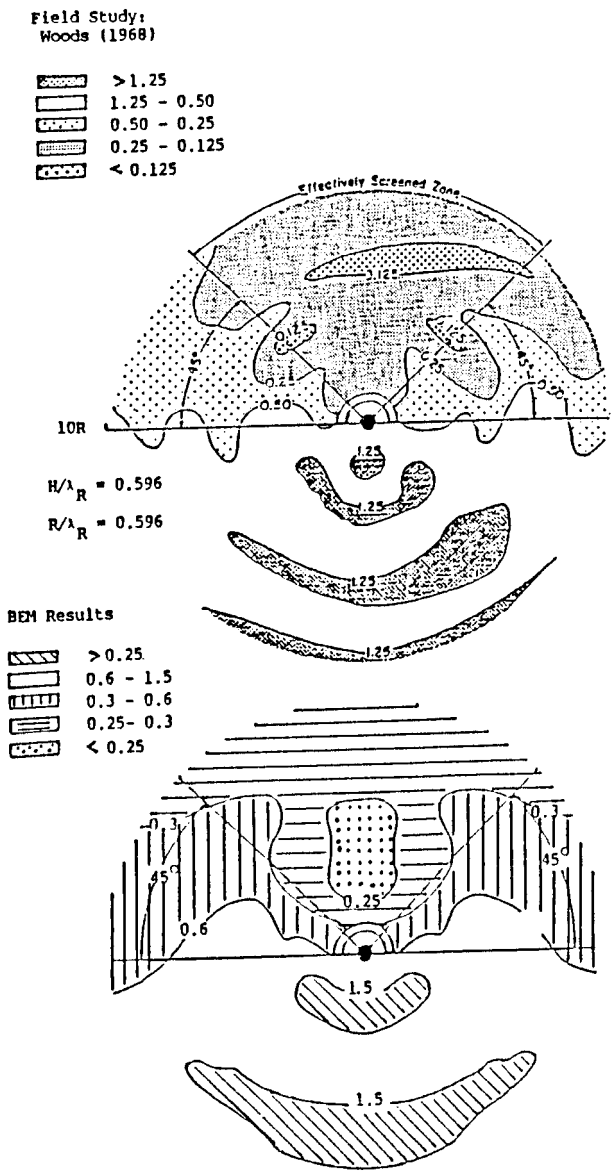


Fig. 7.4.4b Amplitude reduction factor contour diagram

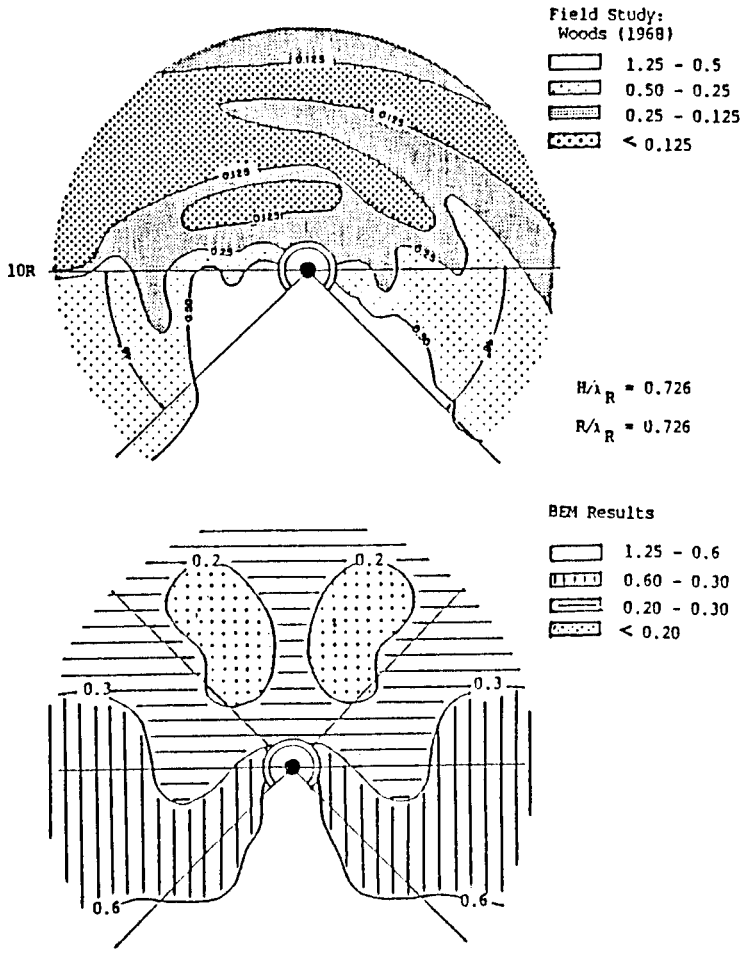


Fig. 7.4.4c Amplitude reduction factor contour diagram

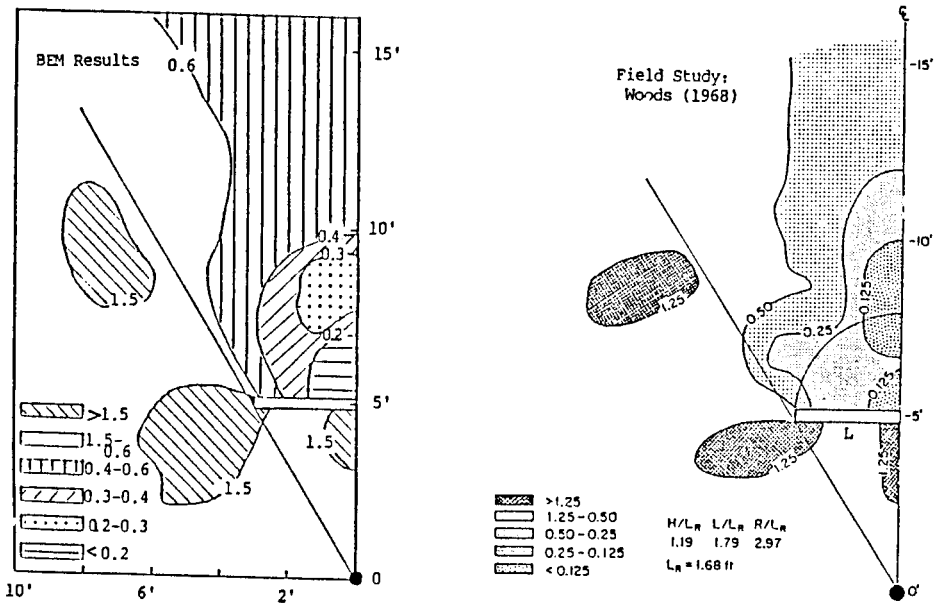


Fig. 7.4.4d Amplitude reduction factor contour diagram

7.5 POROELASTIC ANALYSIS

Consolidation under a strip load [44]:

A number of analytical, as well as, finite element solutions are available for the consolidation of a single poro-elastic stratum beneath a strip load. The geometry for this problem is completely defined in the x-y plane by the depth of the layer H, and the total breadth of the strip load 2a. The lower boundary remains impervious, while free drainage is permitted along the entire upper surface. For convenience, non-dimensional forms of the material parameters are utilized. In particular, let E (Young's modulus) = 1.0, ν (Poisson's ratio) = 0.0, κ (permeability) = 1.0, ν_u (undrained Poisson's ratio)= 0.5, and B (saturation) = 1.0. Notice, for this set of properties, that the diffusivity is unity. The presentation of results is simplified further by also assuming unit values for both the half-width a and the applied traction.

The boundary element mesh for the particular case of H/a = 5 is depicted in Figure 7.5.1a. It contains 12 elements, 25 boundary nodes, and 9 interior points. Symmetry is imposed along x=0. As in the finite element analysis cited here, the region is truncated at a horizontal dimension of L/a = 10. While theoretically the vertical elements along x=10 are unnecessary, closure of the region permits accurate determination of the strongly singular diagonal blocks of the F matrix via rigid body considerations.

First, results are presented for H/a = $\rightarrow \infty$. Figure 7.5.1b contains the pore pressure time history for a point, on the centerline, a unit depth below the surface. The well-known Mandel-Cryer effect, of increasing pore pressure during the early stages of the process, is evident in the graph. Actually, only the final portion of the rising pore pressure response was captured in the boundary element analysis, since a very large time step was selected. Even with this large step, however, the analytical solution is accurately reproduced. In Figure 7.5.1c, the pore pressure distribution with depth is shown for T = 0.1. Obviously, the boundary element solution are comparable to those obtained via finite elements.

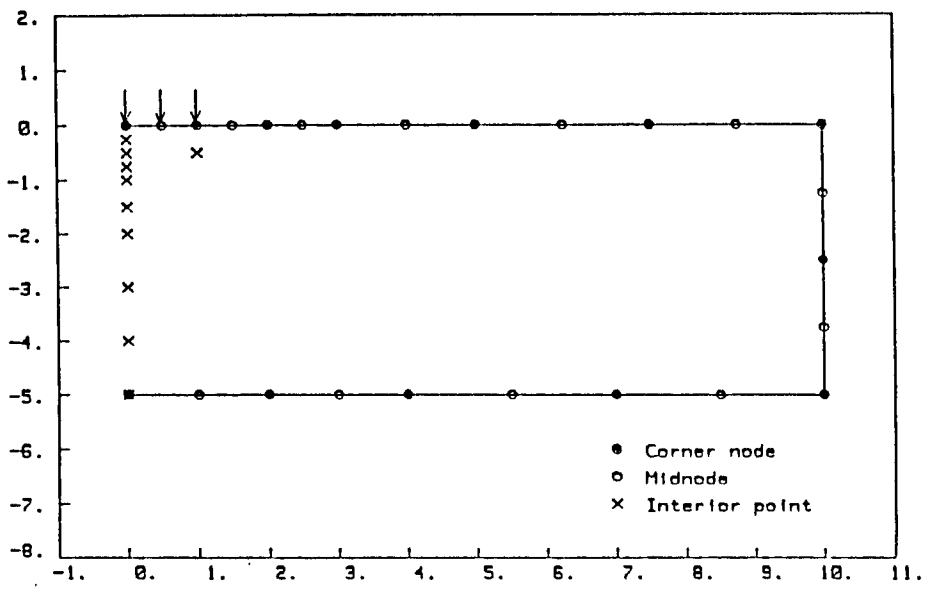


Fig. 7.5.1a Consolidation under a strip load
Boundary Element Model

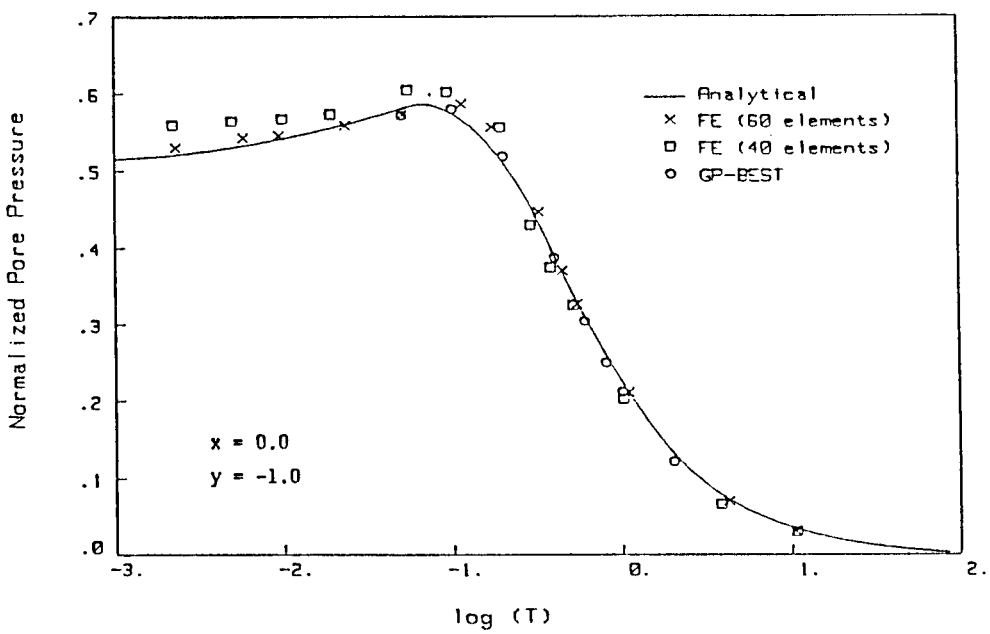


Fig. 7.5.1b Consolidation under a strip load
Results

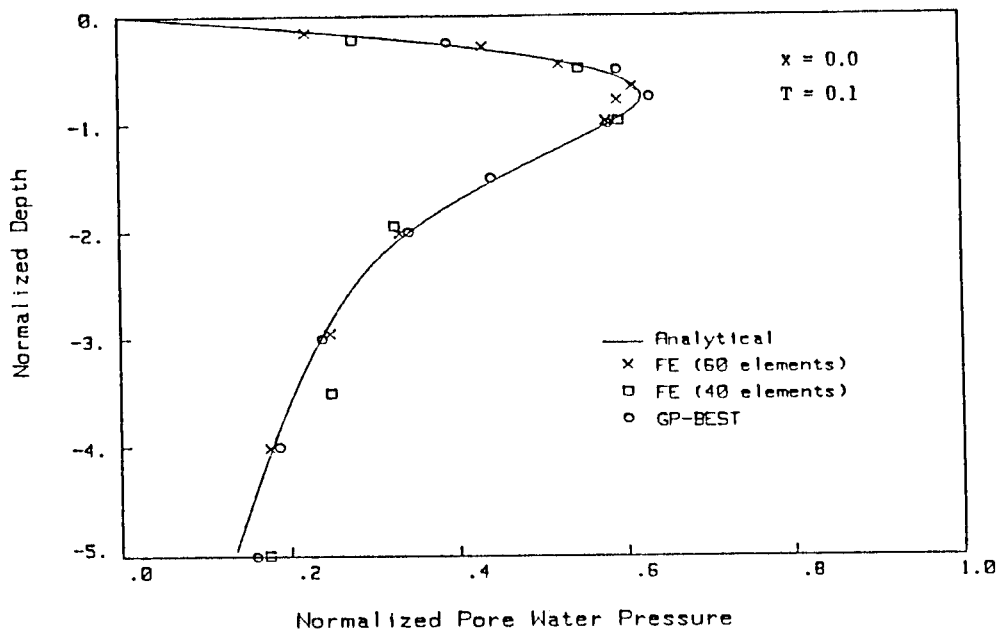


Fig. 7.5.1c Consolidation under a strip load
Results

Consolidation under a circular load:

A very similar problem involves the consolidation of a soil layer under a circular load. The parameter definitions are identical to those detailed in the previous section and will not be repeated here. The only exception is that a is now the radius of the circular load. Two H/a ratios, 10 and 1.0, are examined in the analysis.

The boundary element idealization for H/a = 10 is topologically equivalent to that shown in Figure 7.5.1a, although now the coordinates are r and z. on the other hand, for H/a = 1.0, a two region model is adopted in order to effectively reduce the aspect ratio of the domain. Figure 7.5.2a depicts this model. Region I contains 10 elements and 21 nodes, while Region II consists of 8 elements connecting 16 nodes. The BEM results for the degree of consolidation are compared with the analytical solution in Figure 7.5.2b. The correlation is good for H/a = 10, but, for H/a = 1.0, the agreement is almost exact. The additional refinement in the vicinity of the load, included in the two region model of Figure 7.5.2a, is the probable cause for the precise results obtained in the latter case.

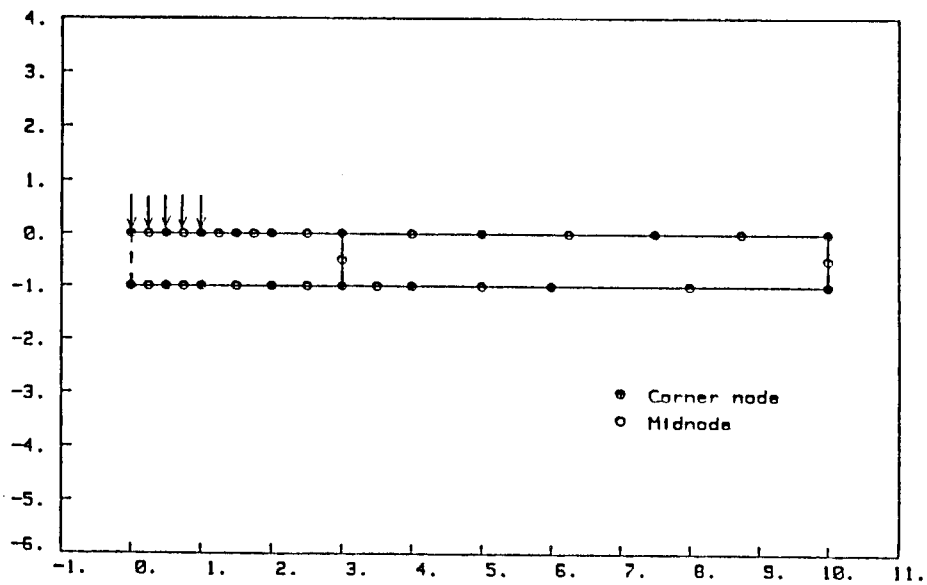


Fig. 7.5.2a Consolidation under a circular load
Boundary Element Model

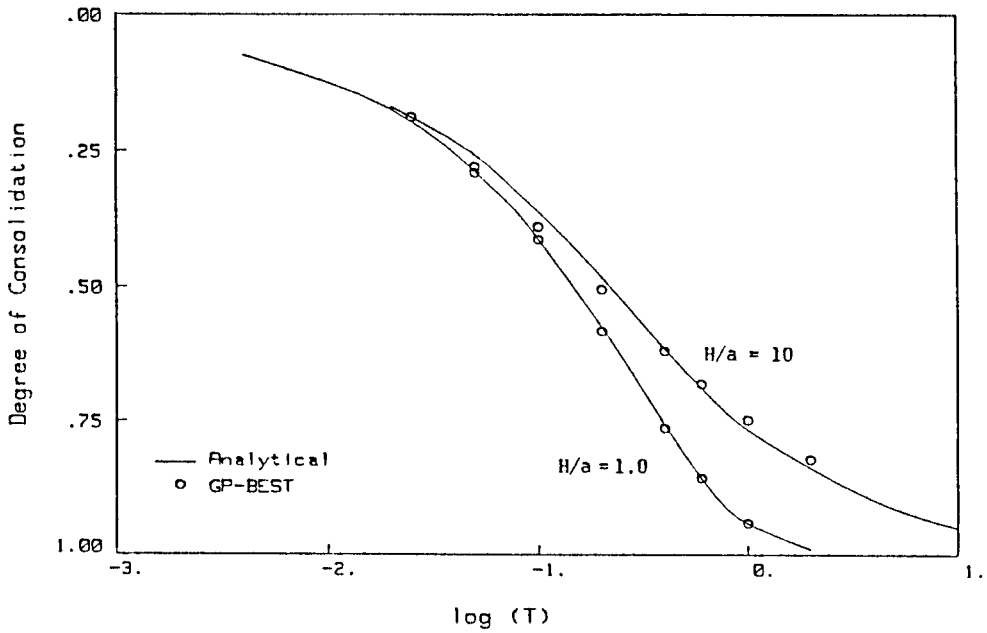


Fig. 7.5.2b Consolidation under a circular load
Results

7.6 THERMO-ELASTIC ANALYSIS

Turbine Blade [45]:

For a real practical application, the plane strain response of an internally cooled turbine blade is examined under startup thermal transients. The boundary element model of the blade is illustrated in Figure 7.6.1a. In this problem, the two GMR approach is chosen solely to enhance computational efficiency. This is accomplished by reducing the aspect ratio of individual GMR's and by creating a block banded system matrix. The leading (lefthand) GMR consists of 26 quadratic elements, while 24 elements are used to model the trailing (righthand) region.

The blade manufactured of stainless steel with the following thermomechanical properties:

$$E = 29.0 \times 10^6 \text{ psi} \qquad \rho c_\varepsilon = 368 \text{ in.} - \text{lb.}/\text{in}^{3o}\text{F}$$

$$\nu = 0.30 \qquad k = 1.65 \text{ in.} - \text{lb.}/\text{sec.}, \text{in}^{\circ}\text{F}$$

$$\alpha = 9.6 \times 10^{-6} \text{ in.}/\text{in.}^{\circ}\text{F}$$

During operation of hot gas flows outside the blade, while a relatively cool gas passes through the internal holes. The gas temperature transients are plotted in Figure 7.6.1b for a typical startup. Convection film coefficients are specified as follows:

Outer surface at the leading edge h = 50 in.-lb./sec.in.^{2o}F

Remainder of outer surface h = 20 in.-lb./sec.in.^{2o}F

Inner cooling hole surfaces h = 10 in.-lb./sec.in.^{2o}F

A time step of 0.2 sec. is employed for the boundary element analysis.

The response at two points, A, on the leading edge and, B, at midspan are displayed in Figures 7.6.1c and 7.6.1d. Notice that temperatures and stresses are consistently higher on the leading edge, reaching peak values of approximately 1500°F and -60 ksi, respectively. Also, as is evident from Figure 7.6.1d, significant stress reversals occur during this startup. As a next step, these numerical results could be used as input for a fatigue analysis to assess the durability of the design. In that regard, it should be emphasized that the stresses presented for points A and B are surface stresses, calculated by satisfying the constitutive laws, strain-displacement and equilibrium directly at the boundary point. This can be expected to produce much more accurate results than the standard practice utilized in finite element approaches of extrapolating interior Gauss point stress values to the boundary.

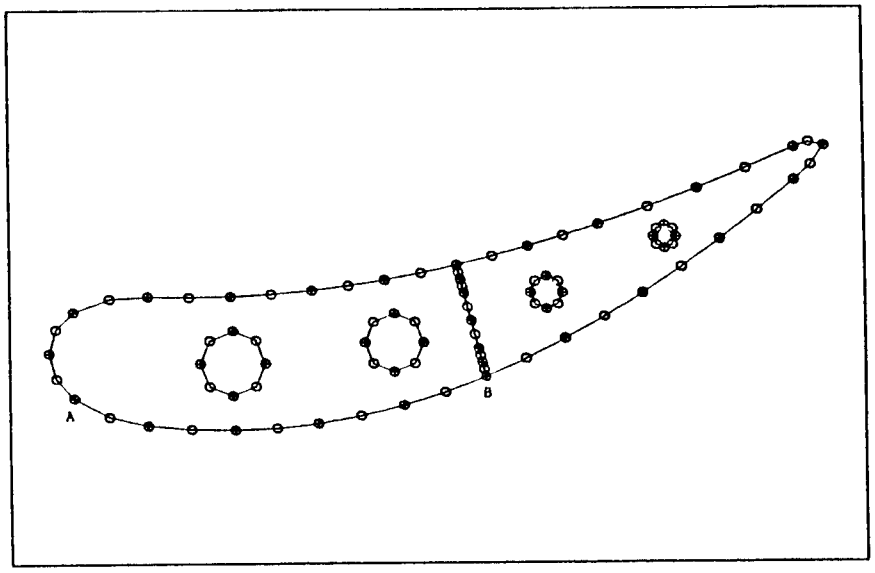


Fig. 7.6.1a Turbine Blade
Boundary Element Model

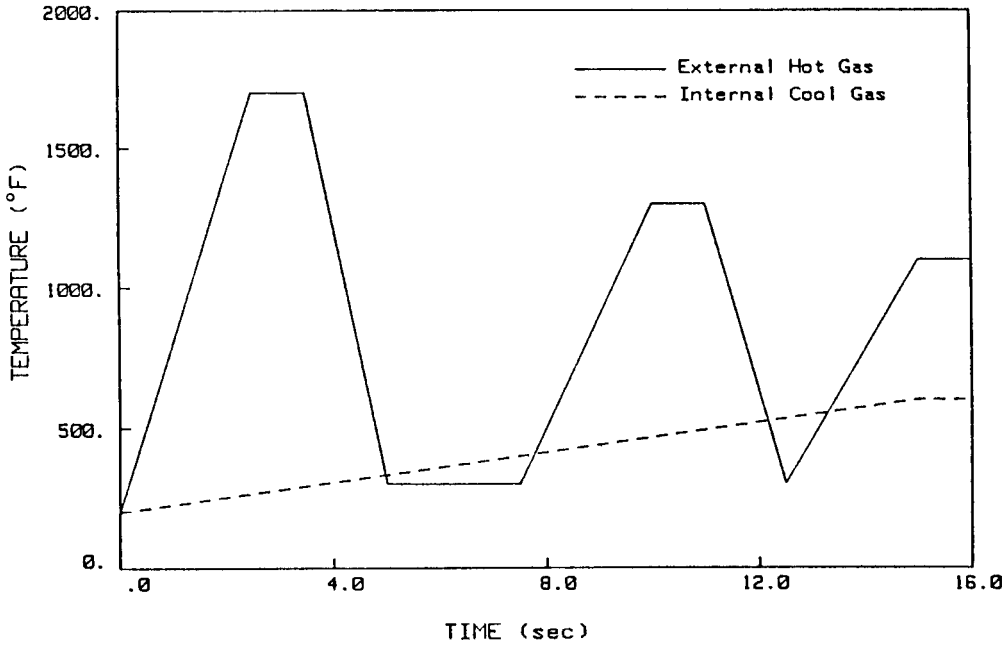


Fig. 7.6.1b Turbine Blade
Startup Transient

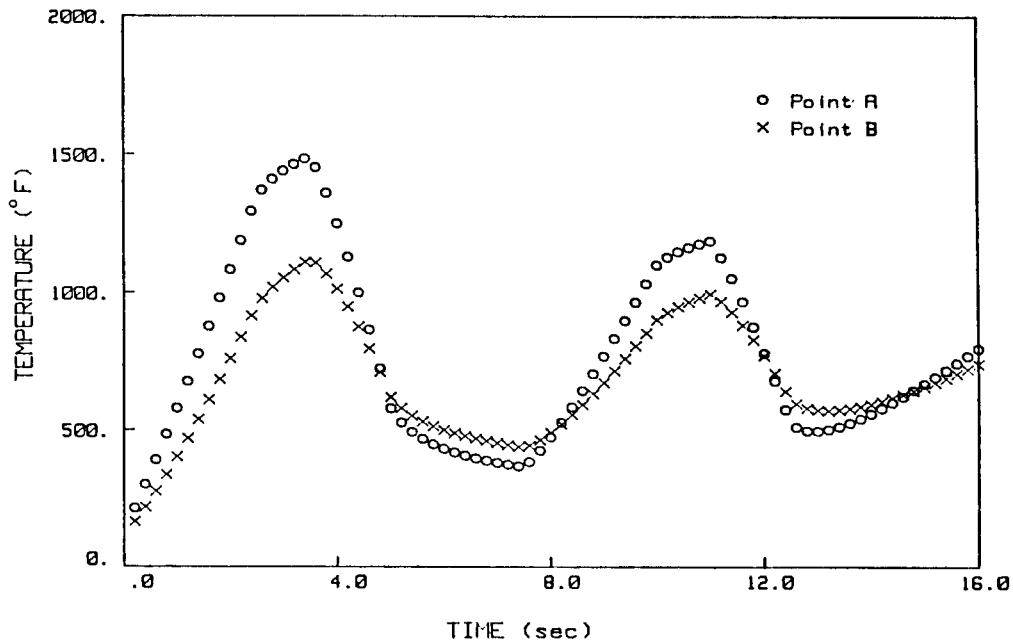


Fig. 7.6.1c Turbine Blade
GPBEST Results

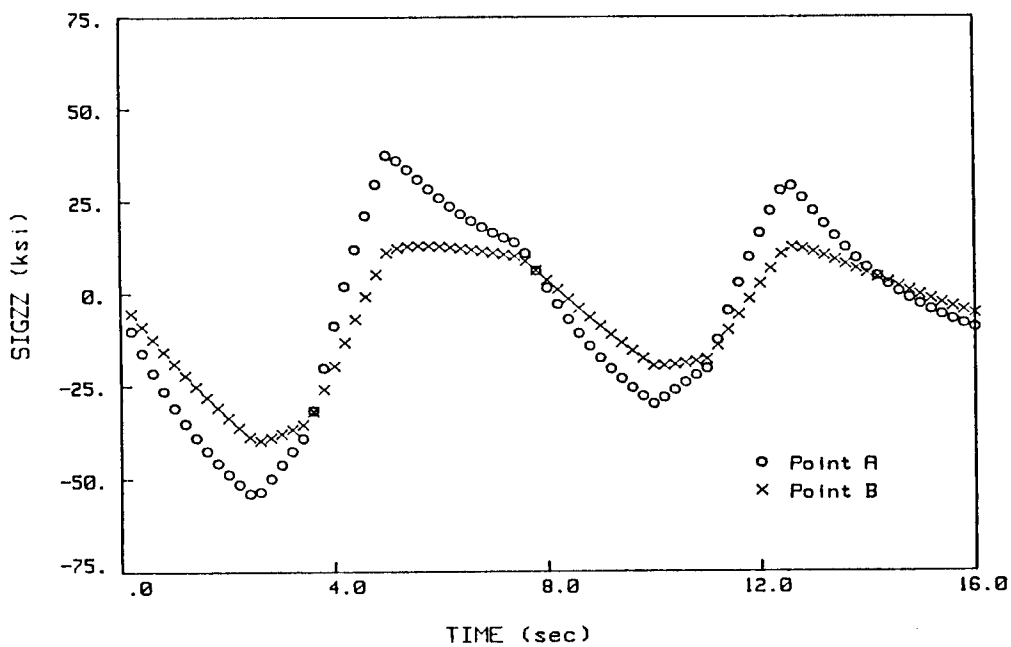


Fig. 7.6.1d Turbine Blade
GPBEST Results

Transient Response of a Tube and Disk Fin Heat Exchanger [46]:

As a good practical example, thermal stresses in a stainless steel tube and disk fin heat exchanger are determined for a typical startup transient. The tube, with an outer diameter of 0.375 in. and a wall thickness of 0.050 in., is brazed around its periphery to a 0.020 in. gauge disk fin. A fillet radius of 0.015 in. is assumed between the tube and fin. Figure 7.6.2a details the cross-sectional geometry.

The heat exchanger is maintained, in a stress-free state, at a uniform temperature of 400°F by a liquid flowing continuously inside the tube. Then, suddenly at time zero the outer surfaces of the tube and fin are exposed to a hot gas, initially at 1400°F. The temperature profiles for both the hot gas and cool liquid are shown in Figure 7.6.2b. The corresponding convection coefficients for the outer and inner surfaces are 10 and 50 in.lb./sec.in.^{2}°F, respectively.

In addition, the following material properties for the stainless steel apply:

$$E = 29.0 \times 10^6 \text{ psi} \qquad \rho c_\varepsilon = 368 \text{ in.} - \text{lb.}/\text{in}^3\ ^\circ\text{F}$$

$$\nu = 0.30 \qquad k = 1.65 \text{ in.} - \text{lb.}/\text{sec.}, \text{in}^o\text{F}$$

$$\alpha = 9.6 \times 10^{-6} \text{ in.}/\text{in.}^o\text{F}$$

For the **GPBEST** analysis, a 15° slice of one half of a fin is isolated as depicted in Figure 7.6.2c. In order to reduce computation time, a two GMR approach was taken. One GMR represents only the fin, while the other contains the tube and the entire braze joint. The model consists of a total of 40 quadratic surface elements connecting 124 nodes. A time step of 0.05 sec. is selected based upon the material properties and element sizes.

Results are presented for points A (inner wall of tube), B (fillet radius) and C (outer edge of fin) as indicated in Figure 7.6.2a. The temperatures at these three points, as calculated via the **GPBEST** analysis, are displayed in Figure 7.6.2d. As expected, the thin fin responds much more quickly to the hot gas than does the tube. This produces compressive stresses in the fin and tensile stresses in the tube as is evident from Figure 7.6.2e, which portrays only the circumferential component. For the point near the base of the fillet radius, this circumferential component is not as significant, however, a large radial stress is present as shown in Figure 7.6.2f. Interestingly, the startup transient examined here is not overly severe. In fact, peak transient stresses are less than twice the corresponding steady-state values.

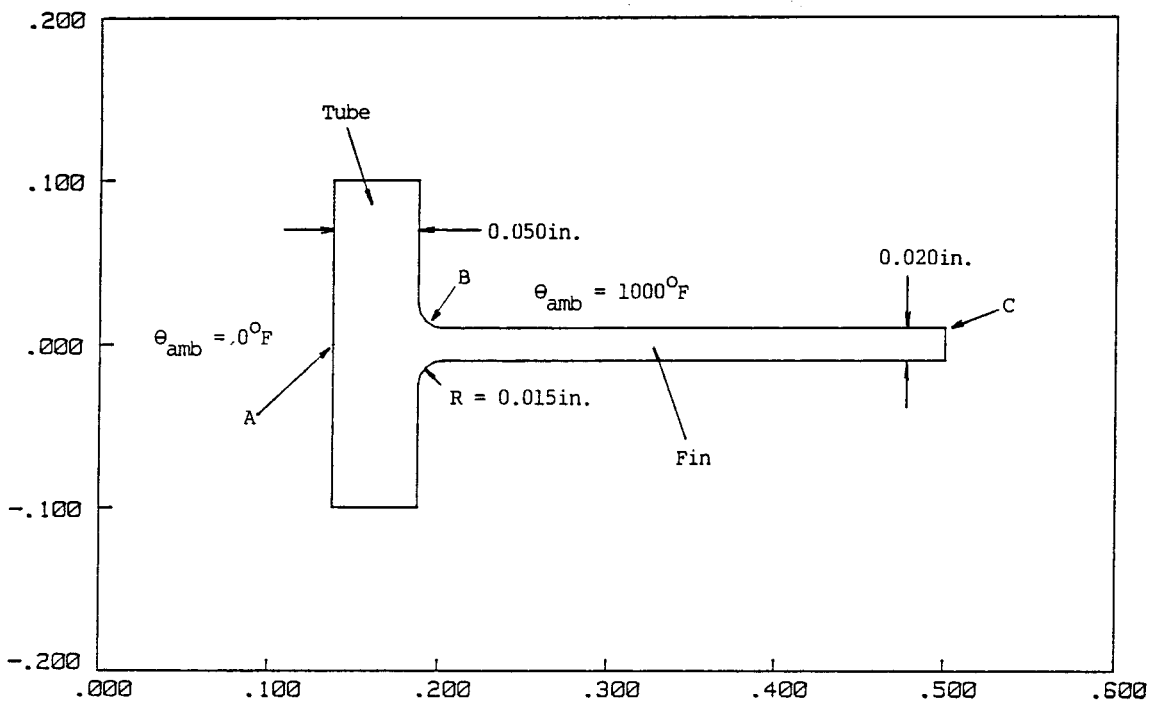


Fig. 7.6.2a Tube and Fin Heat Exchanger
Problem Definition

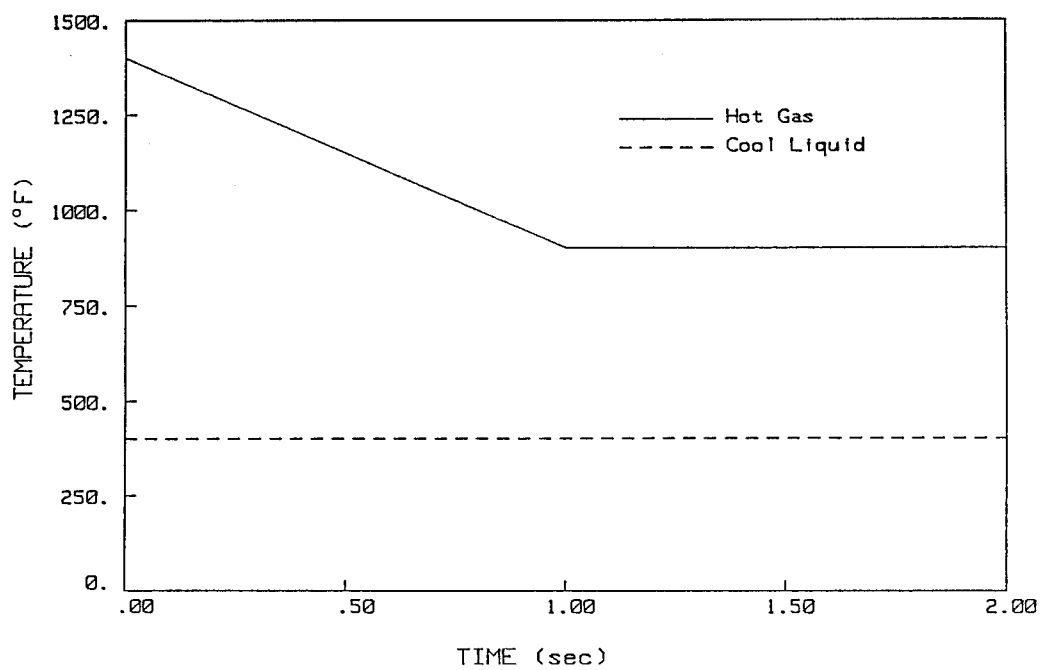


Fig. 7.6.2b Tube and Fin Heat Exchanger
Ambient Temperature

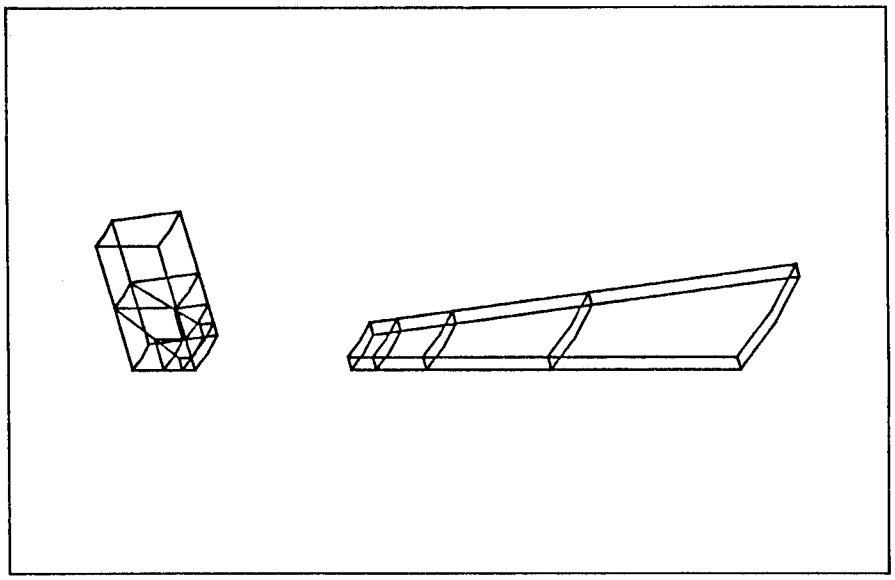


Fig. 7.6.2c Tube and Fin Heat Exchanger - 2 GMR
Boundary Element Model

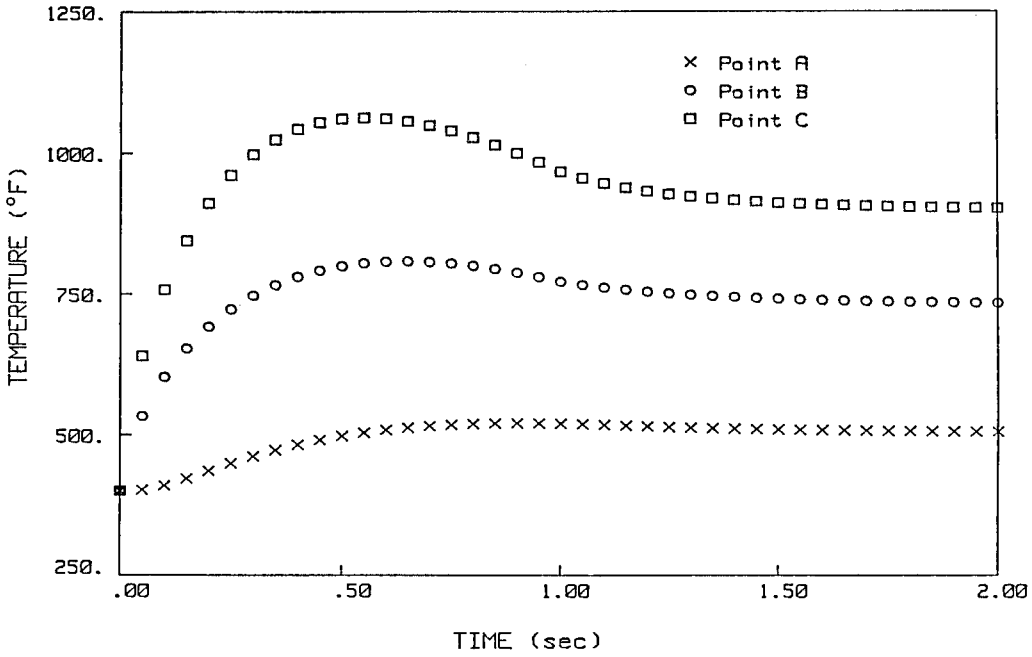


Fig. 7.6.2d Tube and Fin Heat Exchanger
GPBEST Results

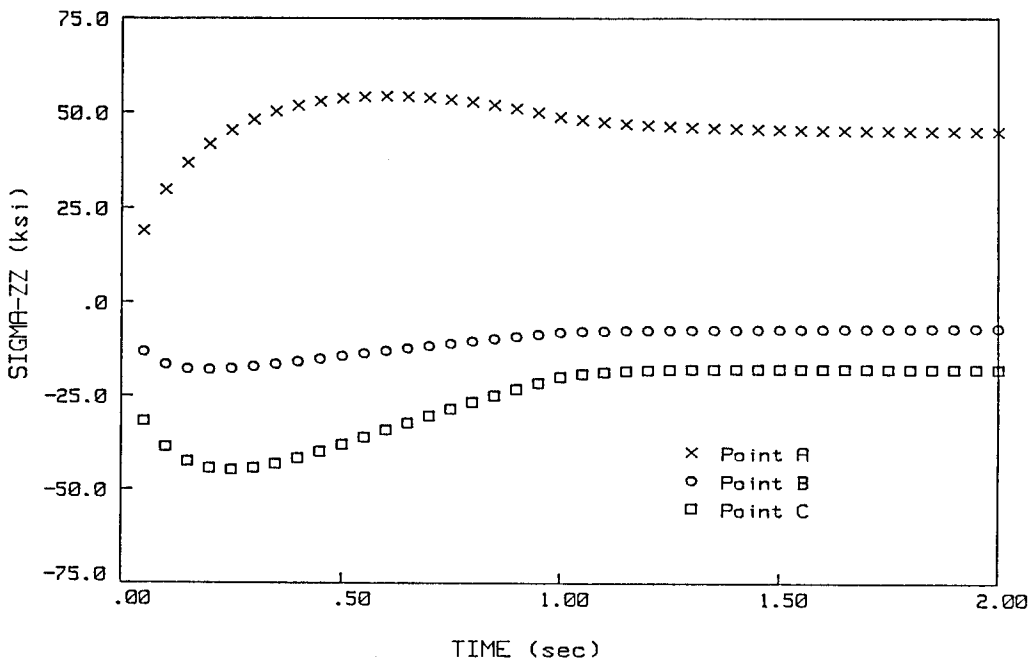


Fig. 7.6.2e Tube and Fin Heat Exchanger
GPBEST Results

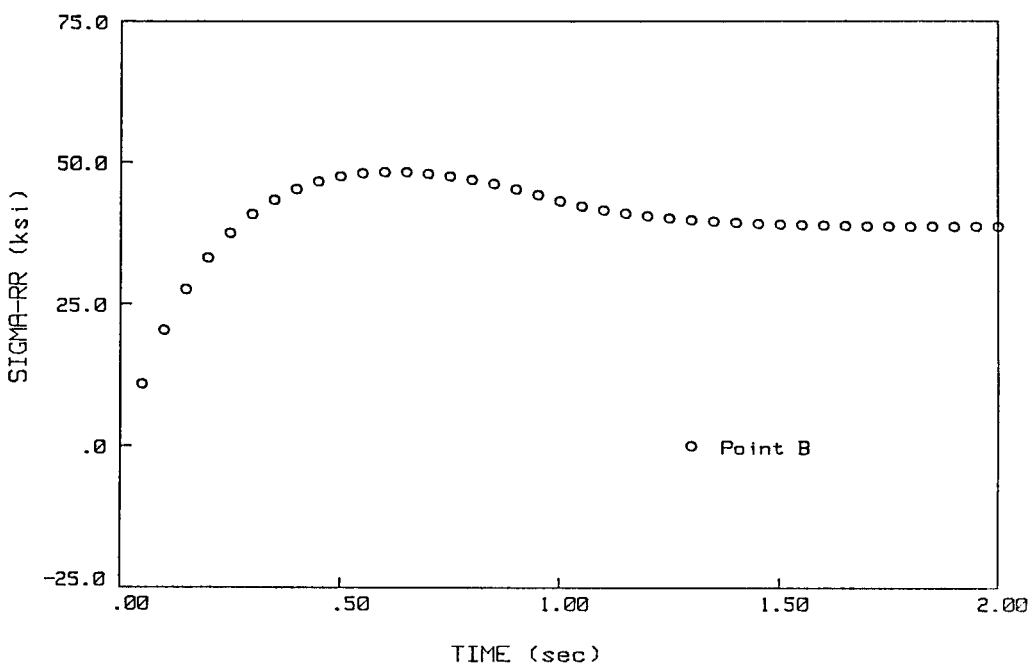


Fig. 7.6.2f Tube and Fin Heat Exchanger
GPBEST Results

7.7 ELECTROMAGNETIC ANALYSIS

Electrostatics of a VLSI Circuit:

Consider an electrostatic analysis of the simplified two-dimensional representation of a VLSI circuit depicted in Figure 7.7.1a. As shown in that figure, four conductors are separated by a pair of insulators with permitivities ϵ of 7.0 and 3.9. Notice that symmetry is assumed along both vertical edges. Analysis is required to determine the capacitance of the circuit.

The boundary element model, detailed in Figure 7.7.1b, consists of two generic modeling regions (GMR's). The lower dielectric is defined by 18 three-noded line elements, while 15 elements are used to model the upper dielectric. Capacitance C_{ij} is determined by setting the electric potential, V of the ith conductor to unity, maintaining the other three conductors at zero potential, and then computing the surface charge on the jth conductor. It should be noted that the calculation of this surface charge is directly available from the boundary element solution, since boundary flux $(\epsilon \frac{\delta V}{\delta n})$ is a primary variable.

Table 7.7.1a provides the capacitance determined via **GPBEST** , along with results obtained in a finite element analysis reported by Szabo [93]. The finite element results were obtained by utilizing a 44 element p-version analysis. All values of C_{ij} obtained by the two methods agree to within 1.0 %. For the two-dimensional case considered here, the geometric modeling of the VLSI circuit is somewhat easier with a BEM approach. However, actual VLSI design problems are typically three-dimensional in nature [94]. In such instances, modeling effort, and in most cases computing time, will be significantly lower for BEM versus any of the domain-based methods.

GPBEST Results

i	C_{i1}	C_{i2}	C_{i3}	C_{i4}	$\sum_{j=1}^{4} C_{ij}$
1	24.85	-12.13	-12.32	-0.40	0.00
2	-12.13	18.75	-2.27	-4.35	0.00
3	-12.31	-2.27	25.23	-10.65	0.00
4	-0.40	-4.35	-10.64	15.40	0.01
$\sum_{i=1}^{4} C_{ij}$	0.01	0.00	0.00	0.00	

Finite Element Results (Szabo, 1988, Table IIb)

i	C_{i1}	C_{i2}	C_{i3}	C_{i4}	$\sum_{j=1}^{4} C_{ij}$
1	24.76	-12.10	-12.24	-0.41	0.01
2	-12.11	18.68	-2.24	-4.35	0.02
3	-12.24	-2.24	25.08	-10.60	0.00
4	-0.41	-4.34	-10.60	15.36	0.01
$\sum_{i=1}^{4} C_{ij}$	0.00	0.00	0.00	0.00	

Table 7.7.1a Capacitance of a VLSI Circuit

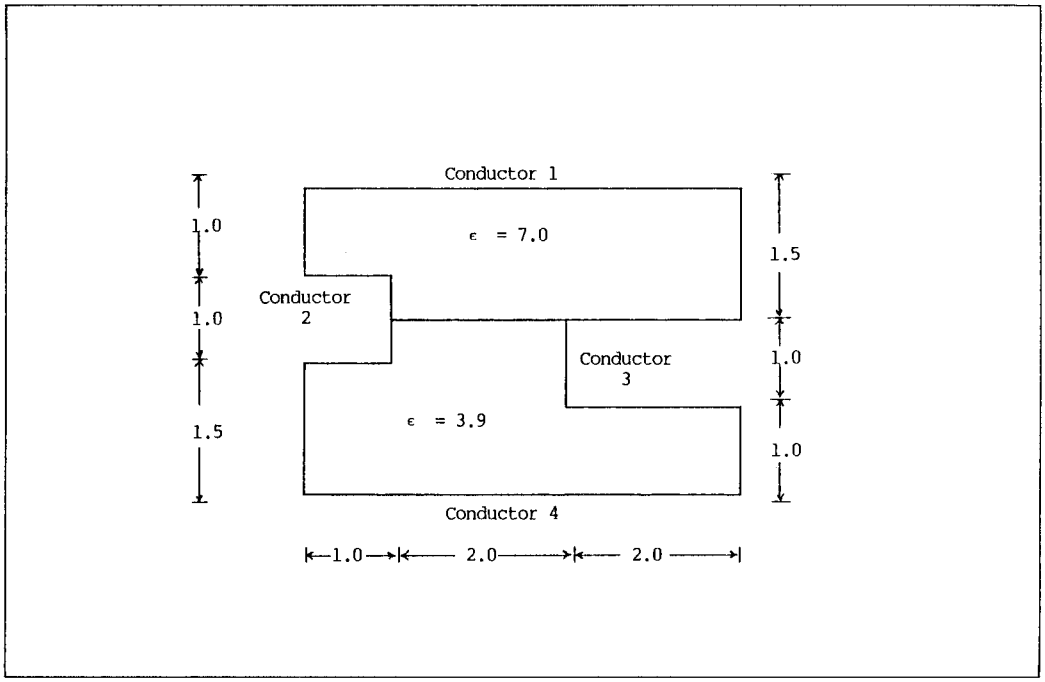


Fig. 7.7.1a Electrostatics of a VLSI Circuit
Problem Definition

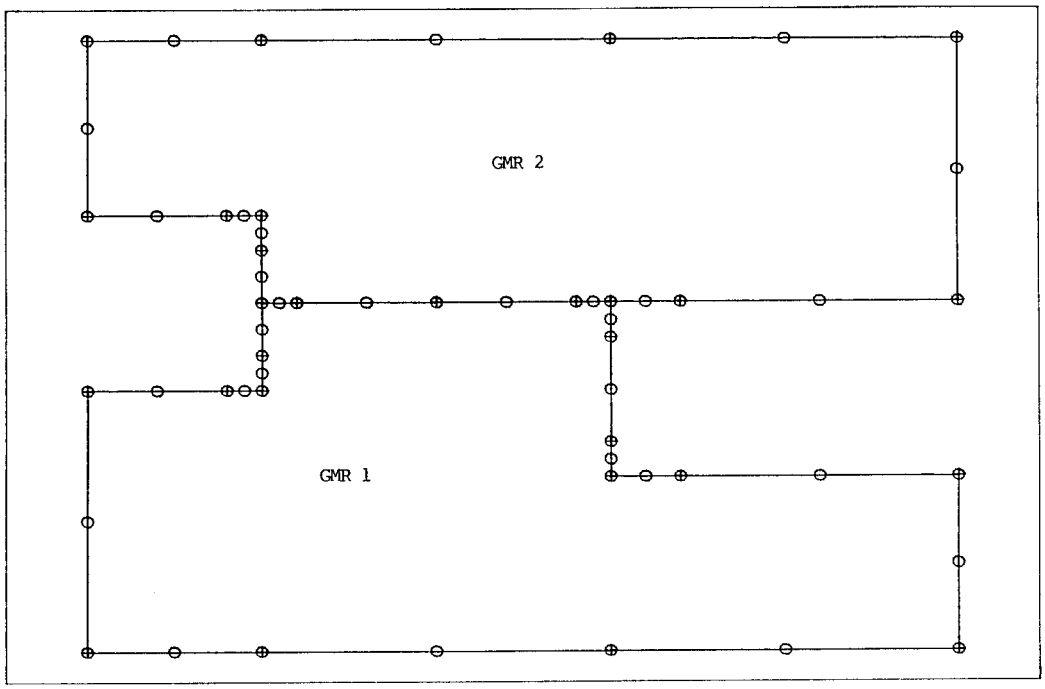


Fig. 7.7.1b Electrostatics of a VLSI Circuit
Boundary Element Model

7.8 FRACTURE ANALYSIS

Elliptic Crack in a Rectangular Block :

A GPBEST analysis of an elliptic crack in a rectangular block is carried out. The problem geometry as well as the boundary element model of one eighth of the body are shown in Figure 7.8.1a. The boundary element model consists of 64 quadratic elements and 188 source points. The first row of elements on both sides of the crack front are modeled by quarter point elements with traction singular modifications for those elements ahead of the crack.

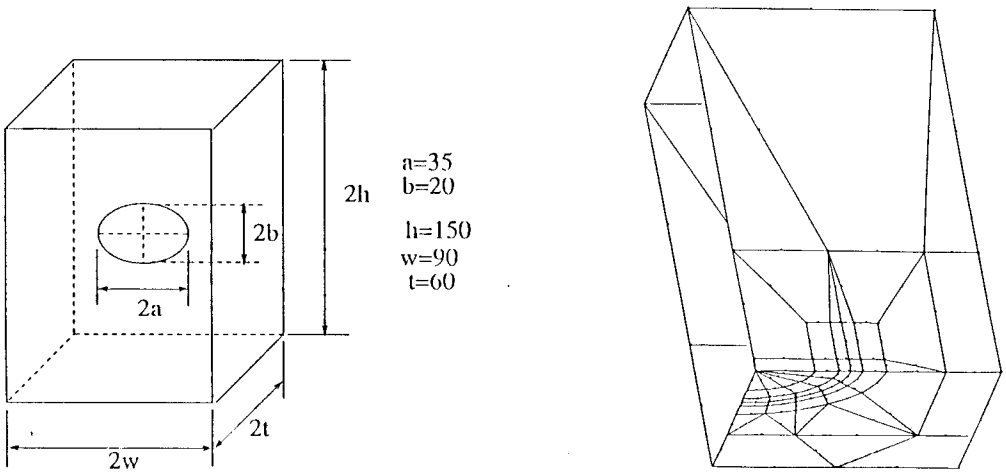


Figure 7.8.1a: Embedded Elliptic Crack

The stress intensity factors (SIFs) computed along the crack front are shown in Figure 7.8.1b. The SIFs are normalized with respect to $\sigma_o\sqrt{\pi b}$, where σ_o is the applied tensile load and b is the semi-minor axis of the crack.

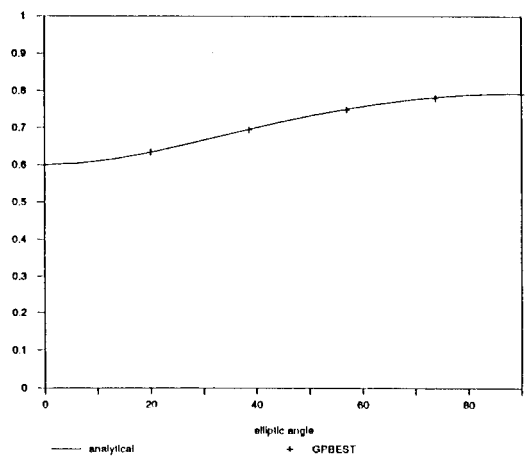


Figure 7.8.1b: Stress Intensity Factors

Inclined Crack in a Square Plate :

An inclined crack of length 1.1547 in a plate of 20*20 is studied by using GPBEST. The plate is modeled as a two region problem by dividing the plate along the crack surface as shown in Figure 7.8.2.

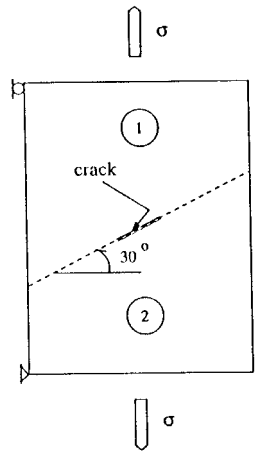


Figure 7.8.2: Inclined Crack

The mode I and mode II stress intensity factors normalized with respect to the analytical solution for the same crack in an infinite plate are 1.009 and 1.002.

Buried Circular Crack [116,117]:

A circular crack of radius 1 embedded in a cylinder is studied by using GPBEST. The geometry and the boundary element model of one quarter of the half-cylinder are shown in Figure 7.8.3. The mode I stress intensity factor normalized with respect to the analytical solution for the same crack in an infinite region is 1.012.

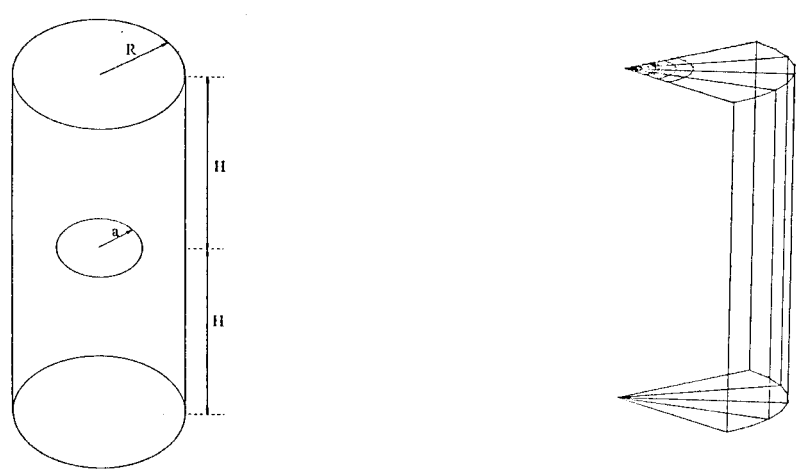


Figure 7.8.3: Buried Circular Crack

Center and Edge Cracks in Finite Plates [118,119]:

Two problems are analyzed by using GPBEST; the first is a center crack problem and the second is an edge crack problem. The problem geometries are shown in Figure 7.8.4a. The plates are modeled by dividing the geometry into two regions along the material demarcation.

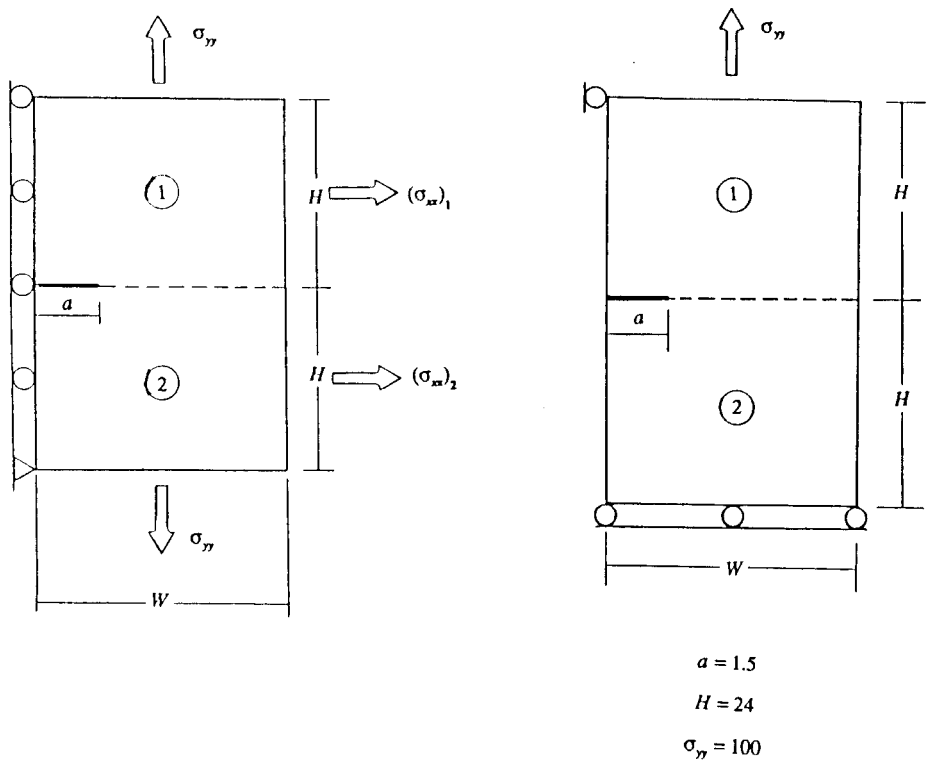


Figure 7.8.4a: Interface Crack (a) Center Crack (b) Edge Crack

The complex stress intensity factors (SIFs) are computed for different geometries of the plates. The real and imaginary parts of SIFs for various material combinations are shown in Figure 7.8.4b for the center crack and in Figure 7.8.4c for the edge crack. The parameter γ is given in terms of material properties as

$$\gamma = \frac{\alpha_1 \mu_2 + \mu_1}{\alpha_2 \mu_1 + \mu_2}$$

$$\alpha_i = 3 - 4\nu_i \text{ for plane strain}$$

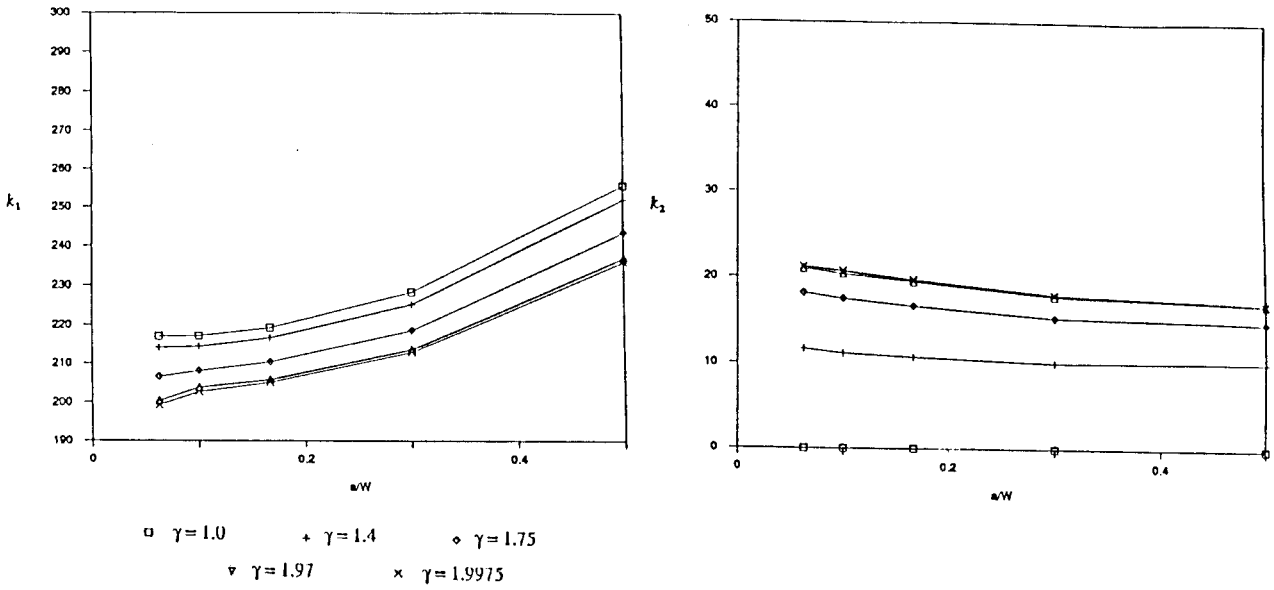


Figure 7.8.4b: Stress Intensity Factors (a) Real (b) Imaginary

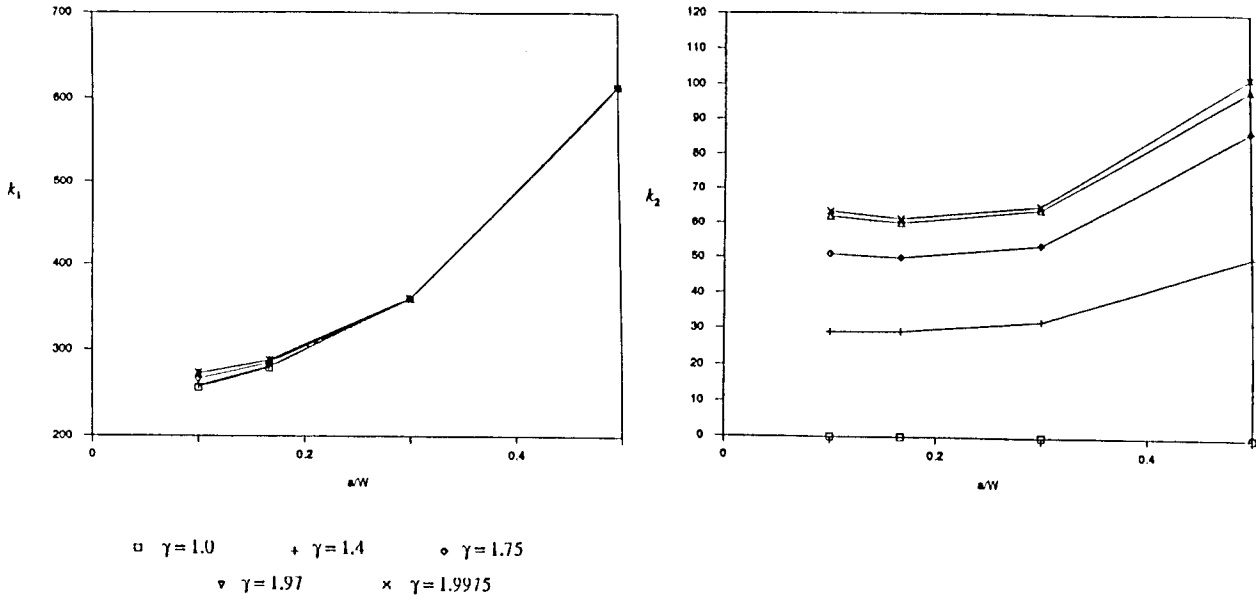


Figure 7.8.4c: Stress Intensity Factors (a) Real (b) Imaginary

Arbitrary Crack in a Rotating Disk :

An arbitrarilly located crack in a rotating disk is studied by using GPBEST. The geometry of the problem is defined in Figure 7.8.5a. The boundary element model of one-half of the disk is also shown in Figure 7.8.5a.

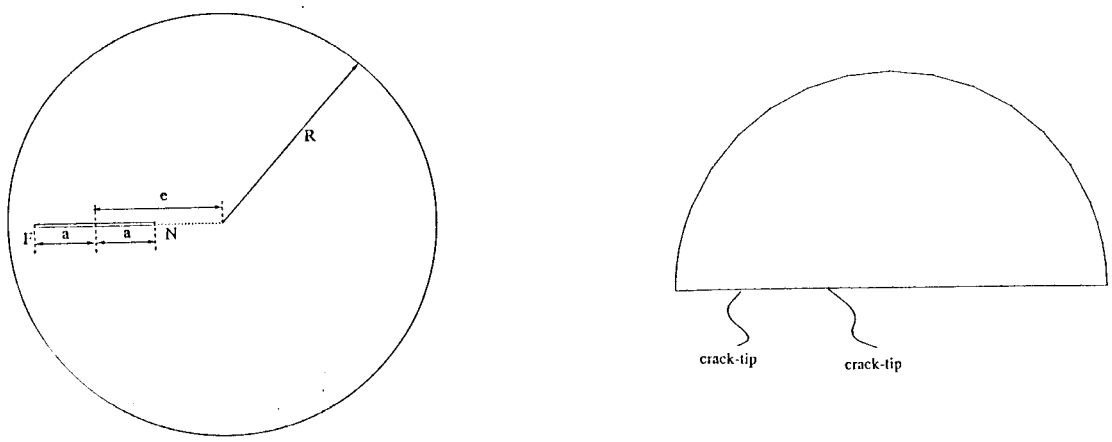
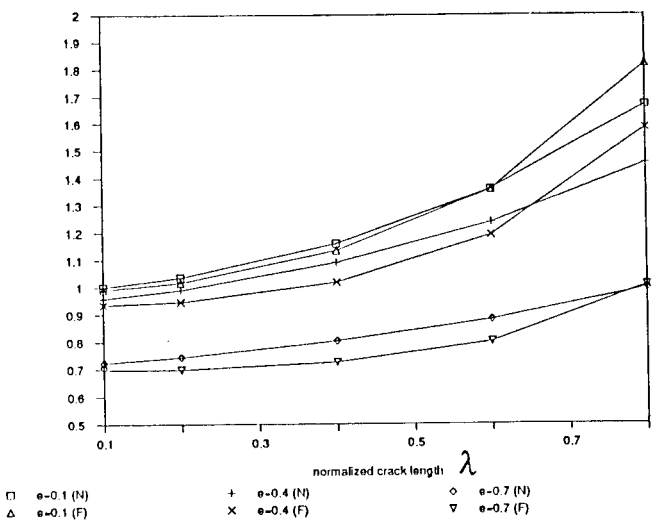


Figure 7.8.5a: Crack in a Rotating Disk

The stress intensity factors (SIFs) computed are normalized by $\sigma_o\sqrt{\pi a}$, where σ_o is the stress in an uncracked plate rotating at an angular veclocity of ω. The stress intensity factors at ends N and F are shown in Figures 7.8.5b.



$$\varepsilon = e/R$$

$$\lambda = \frac{a}{(R-e)}$$

Figure 7.8.5b: Mode I SIFs at Crack-tips N and F

Circular Crack in a Rotating Cylinder [120,121,122]:

A circular crack embedded in a rotating cylinder is analyzed. Due to symmetry only one quarter of a half-cylinder is modeled. Different problems are run by changing the radius R of the cylinder. A typical mesh for $a/R = 0.3$, where a is the crack radius, is shown in Figure 7.8.6a. The model consists of 66 quadratic elements and 184 source points.

Figure 7.8.6a: Circular Crack Model

The problem is studied under two conditions; in the first case the cylinder is restrained at both ends and in the second case the cylinder is free at both ends. The stress intensity factors (SIFs) for both cases are shown in Figure 7.8.6b. The SIFs are normalized with respect to $2\sigma_o\sqrt{\frac{a}{\pi}}$, where σ_o is the out-of-plane stress at the center of the uncracked cylinder rotating at an angular velocity w.

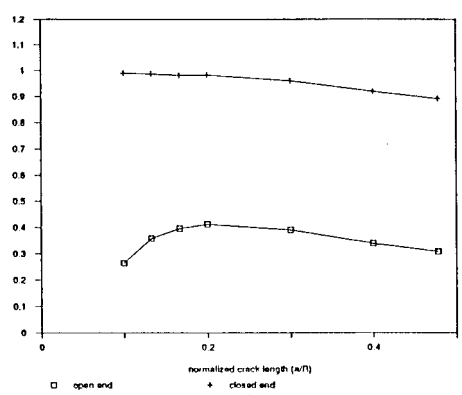


Figure 7.8.6b: Stress Intensity Factors for Circular Cracks

7.9 ELASTO-PLASTIC ANALYSIS

Notched Tensile Plate [125]:

The boundary element method results in accurate stress results near the boundary. Fortunately, most plastic zones will form and grow near the boundary where the stresses are most accurate. A mesh convergence study has been carried out for the problem shown in Figure 7.9.1a. The plane strain notched tensile specimen was modeled with three different BEM meshes (Mesh2, Mesh4, Mesh8), in order to study the effect of the chosen mesh on the analysis. The material model is a perfectly plastic Von Mises model with no hardening effect and an initial yielding strength of σ_o.

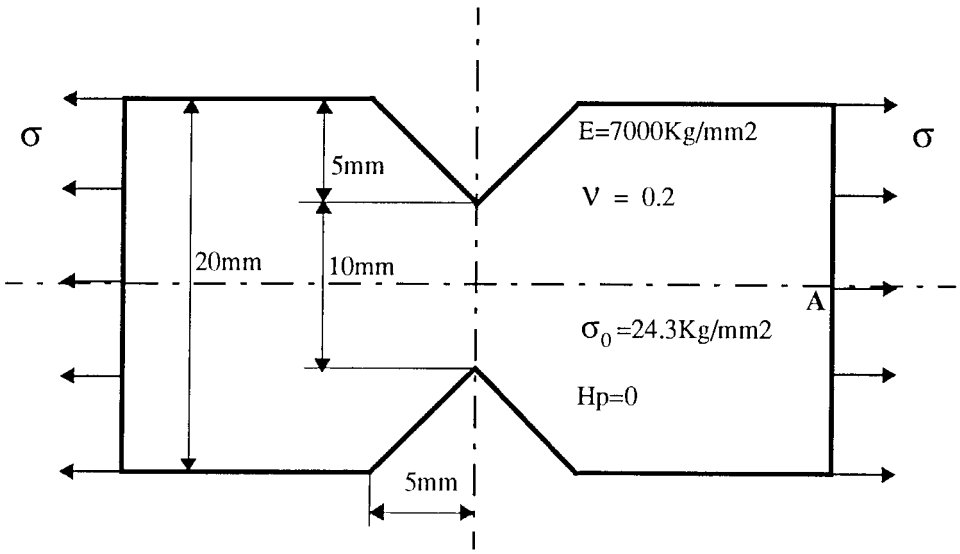


Figure 7.9.1a: Notched Tensile Plate

Upper and lower bound solutions are available for this problem. Also, this problem has been solved using finite element analysis. Although the finite element mesh used was quite fine (169 triangular elements), the limit load obtained by FEM was higher than the upper bound solution.

On the other hand, the GPBEST analysis for all three meshes are in the proper bounds (Figure 7.9.1b).

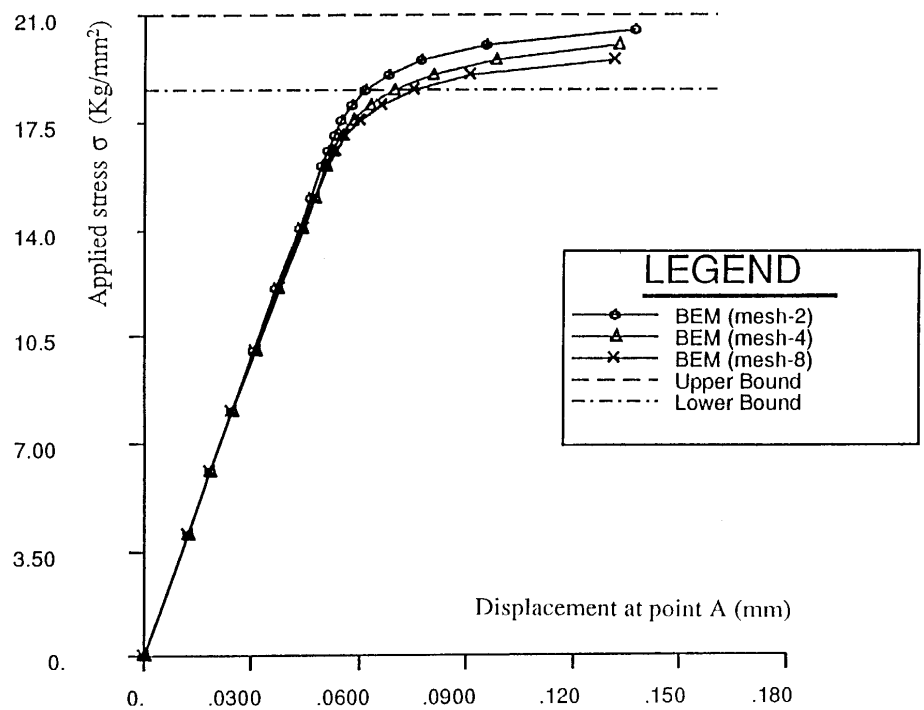


Figure 7.9.1b: Displacement Load Response Curves

Elasto-Plastic Contact Analysis of Disc Brake Assembly [125]:

Figure 7.9.2a shows a typical automobile disc brake which consists of four parts: brake rotor, brake pad, piston and caliper. Hydraulic pressure is applied inside the caliper to activate the brake which forces the two brake pads against the rotor.

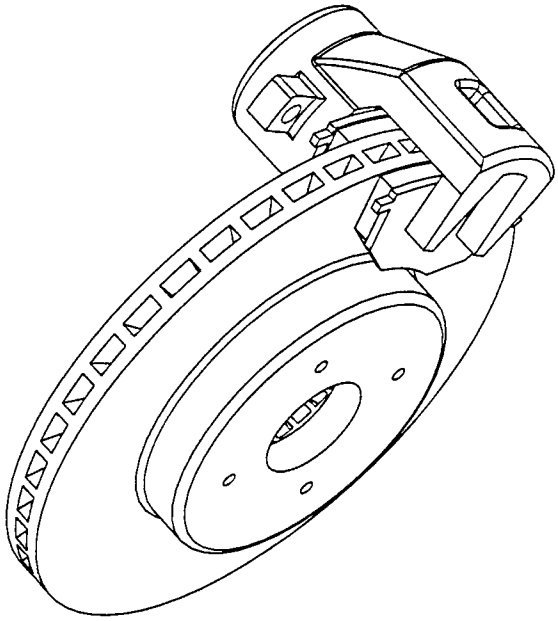


Figure 7.9.2a: Disc Brake Assembly

A three-dimensional GPBEST model was built using PRO/ENGINEER. Contact conditions are specified at the piston/pad and pad/rotor interfaces. A combined elasto-plastic nonlinear contact analysis was run.

Figure 7.9.2b shows the deformation of the model under pressure. Part of the brake pad lifts away from the rotor which is captured by the analysis.

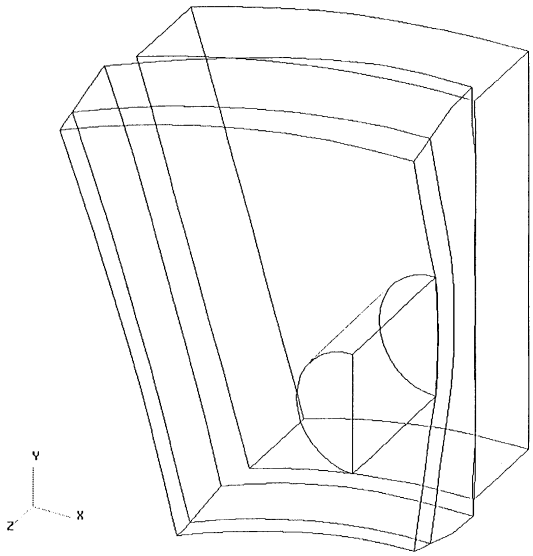


Figure 7.9.2b: Displacement Due to P=11 ksi Load

Extrusion of a Metal Strip [125]:

A plane strain elasto-plastic analysis was carried out on the problem in Figure 7.9.3a. A crude mesh is illustrated in Figure 7.9.3b where the extruded material is modeled with interior cells, but the steel die (C-D-E-F) is modeled only on the boundary.

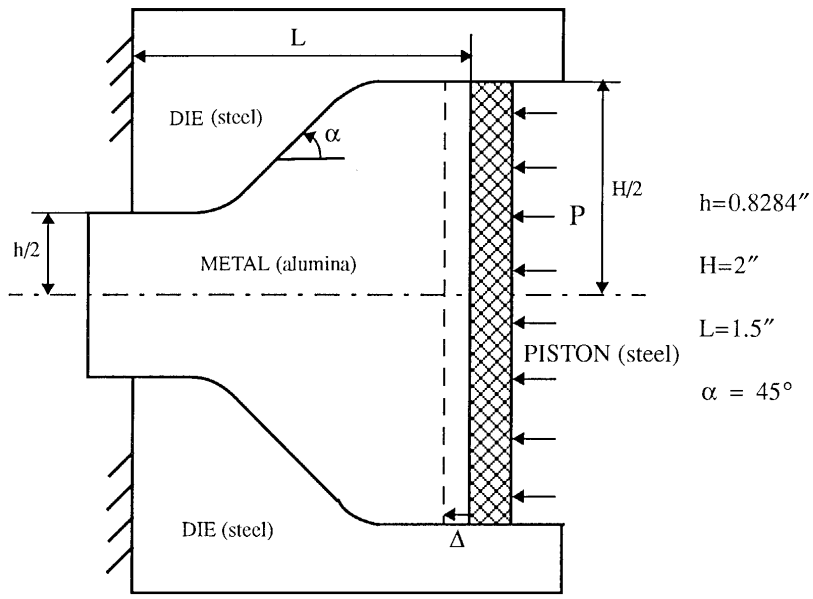
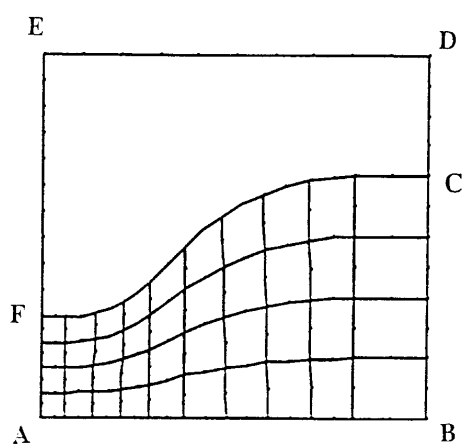


Figure 7.9.3a: Extrusion Through a Steel Die



	Die	Metal
Boundary Elements	22	28
Interior Cells	0	40
Boundary Nodes	45	57
Total Nodes	45	149

Figure 7.9.3b: Mesh for Extrusion Problem

The resulting pressure vs. displacement curve along B-C is shown in Figure 7.9.3c. Notice the good agreement between the GPBEST analysis and theoretical considerations.

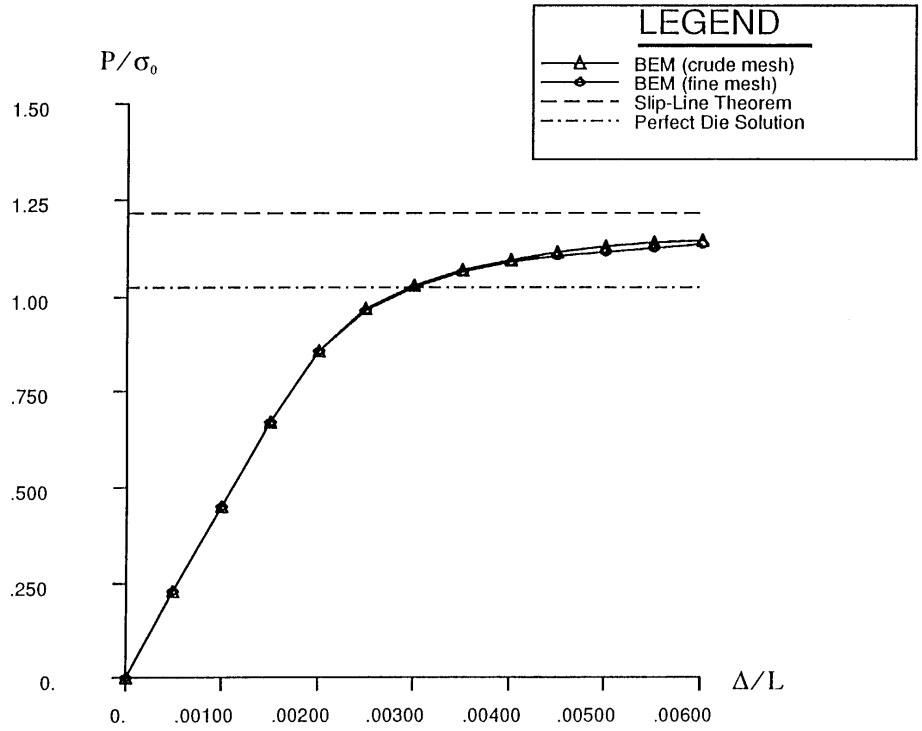


Figure 7.9.3c: Results for Extrusion Problem

8.0 GPBEST DISTRIBUTED COMPUTING ENVIRONMENT (GPBEST-DCE)

General Purpose Boundary Element Solution Technology - Distributed Computing Environment (**GPBEST-DCE**) offers significant advantages by allowing a heterogeneous collection of workstations, supercomputers, and multi-processor machines to function as a single high-performance parallel machine. **GPBEST-DCE** is portable and can run on a wide variety of modern platforms which have been used successfully for solving large-scale problems in the science and engineering community.

Designed to link computing resources and provide users with a parallel platform for running **GPBEST** , the physical location or number of different computers used is of no consequence to **GPBEST-DCE** . When **GPBEST-DCE** is properly installed, it is capable of combining heterogeneous networked computing platforms to achieve much higher levels of performance.

The overall **GPBEST-DCE** system has three primary objectives :

. Utilize idle CPU hours for productive work

. Provide drastically reduced wallclock time for design project turn-around

. Provide a system-wide load balanced computational resource

Using the state-of-the-art boundary element technology with **GPBEST** and integrating the flexibility of a distributed computing environment, even larger more realistic problems can now be solved with greatly reduced wallclock time.

The primary objective of this User's Manual for this reduced version is to provide an overview of some **GPBEST-DCE** capabilities, along with detailed descriptions of the input data requirements. Sufficient material is included to facilitate the preparation of **GPBEST-DCE** input data sets.

A list of **GPBEST-DCE** general features is shown in Table 8.1. The **GPBEST** analyses that can be executed under a distributed computing environment is given in Table 8.2. In Section 8.1, **GPBEST-DCE** installation instructions and password file information is presented. Section 8.2 lists the contents of the **GPBEST-DCE** distribution, and Section 8.3 explains the system requirements for **GPBEST-DCE** operation. The main guide to using **GPBEST-DCE** is located in Section 8.4, followed by the basic command line execution Section 8.5. General limits and startup problems is presented in Section 8.6 and 8.7. A sample **GPBEST-DCE** log file is annotated in Section 8.8. (Note that the input cards for **DCE** are described in full in Section 5.8 of this manual.)

NOTE :

A GPBEST-DCE LICENSE IS REQUIRED FOR **GPBEST** TO OPERATE
IN A DISTRIBUTED COMPUTING ENVIRONMENT

TABLE 8.1
GENERAL FEATURES OF GPBEST-DCE

- Automatic **GPBEST-DCE** installation checking
- Automatic error checks of input data
- Automatic elimination of down hosts
- Automatic user and load sensing of available hosts
- Automatic unique data and application file handling
- Automatic **GPBEST-DCE** configuration startup and shutdown
- Automatic GMR and host work load calculations
- Automatic dynamic **GPBEST-DCE** load balancing
- Automatic fault tolerance algorithm
- Automatic serial and **GPBEST-DCE** mode time keeping
- Automatic speedup and time savings calculations
- Automatically releases hosts from the **GPBEST-DCE** configuration when no longer needed
- **GPBEST-DCE** courtesy weighting factor capability
- **GPBEST-DCE** variable priority factor capability
- **GPBEST-DCE** specific host memory allocation capability
- Automatic **GPBEST-DCE** datacheck and log file
- Documented **GPBEST-DCE** GMR log files
- Documented **GPBEST-DCE** GMR result file
- Documented **GPBEST-DCE** execution log and result file

TABLE 8.2
ENGINEERING ANALYSIS BY GPBEST-DCE

A. Elastic Analysis

- Two, three-dimensional, and axisymmetric problems for isotropic media
- Anisotropic media for two and three dimensional problems
- Centrifugal and self-weight body forces
- Spring-loaded or sliding interfaces
- Gaps, contacts and interference fits
- Problems involving symmetric modes of loading for axisymmetric bodies under non-axisymmetric loading
- Mixed regions involving plane stress and axisymmetric bodies
- Sampling planes
- Holes
- Point forces

B. Thermal Stress Analysis

- Two, three-dimensional and axisymmetric problems
- Steady state linear thermal stress analysis
- Transient linear thermal stress analysis
- Concurrent heat transfer and stress analysis for isotropic and cross-anisotropic bodies
- Cross-anisotropic materials for three-dimensional thermal stress analysis
- Gaps, contacts and interference fits
- Sampling planes
- Point forces and heat sources

C. Heat Transfer Analysis

- Two, three-dimensional and axisymmetric problems involving isotropic as well as anisotropic media
- Steady state heat transfer
- Transient heat transfer
- Thermal resistance across subregions

- Higher order time stepping for two-dimensional, three-dimensional and axisymmetric regions
- Other potential flow or diffusive systems, such as corrosive electolytic processes, electrostatics, and ground water flow
- Sampling planes
- Point sources of heat

D. Fracture Mechanics Analysis

- Elastic analysis with body forces
- Thermal fracture analysis

8.1 INSTALLATION INSTRUCTIONS

Please read all the instructions carefully. It is highly recommended that you request your system administrator to do this installation.

Items 1 through 6 explain how the executables, examples and interface directories are unloaded from tape and stored in an area that is accessible to all users. The **GPBEST-DCE** installation tape contains approximately 8 megabytes of information. Therefore, a file system which has at least 10 megabytes available should be used. Typically, on unix systems, the /usr/local directory is used for storing all non-system programs. In this installation guide, /usr/local is used as the **GPBEST-DCE** installation directory. Substitute /usr/local with the name of your installation directory if you are using another directory.

NOTE: If /usr/local directory is changed to any other directory, a symbolic link must be made in the /usr/local directory pointing to the location of **GPBEST-DCE** installation directory.

Before starting, you will need to know the device name of your tape drive. Generally, on systems with one tape drive, the following device names are used:

SUN systems: /dev/rst0
IBM RS6000 systems: /dev/rmt0
Silicon Graphics systems: /dev/tape
Hewlett Packard 9000/700 series systems: /dev/rmt/0m

I. Login as superuser.

II. Create directory and read **GPBEST** tape.

1. Create a Directory where you want the **GPBEST** files to reside and set permissions so that the files are accessible by all users. The choice of where the files are stored can be made by the user. For example, create a sub-directory called gpbest under the directory /usr/local of your system.

   “ `cd /usr/local` ”

   “ `mkdir gpbest` ”

   “ `chmod ugo+rx gpbest` ”

2. Change to /usr/local/gpbest directory.

   “ `cd gpbest` ”

3. Insert tape into drive.

4. Copy contents of tape to the directory. Substitute /dev/tape for the appropriate tape device.

   “ `tar xvpf /dev/tape` ”

5. Add the following lines to all users .cshrc file so they can execute **GPBEST-DCE** .

```
if ( -f /usr/local/gpbest/lib/dce-cshrc ) then
source /usr/local/gpbest/lib/dce-cshrc
else
echo GPBEST-DCE environment not properly setup
echo Correct path or file permissions on dce-cshrc file
echo Refer to Installation Chapter 8.2 GPBEST Users Manual
endif
```

6. The above listed installation procedure can be used for each individual machine that the user wishes to use **GPBEST-DCE** or can be used on a local server which all **GPBEST-DCE** machines have direct access. The complete pathname to the **GPBEST-DCE** installation must be designated for each individual machine. Note: **GPBEST-DCE** installation is designed to be compatible (using the same directory structure) with any previously installed **GPBEST** distribution.

III. Create a second directory for password file.

1. Create a sub-directory for password file under the root directory called /gpbest and set permissions so that it is accessible to all users. (You can also create a symbolic link to another directory; but since the contents of this directory are less than 1K, it is not recommended that you do so.) The directory where the password file is stored must be under /gpbest.

GPBEST-DCE installation is now complete. If you encounter any problem, please send us a fax at (716) 639-1919 with a brief description of your problem.

PLEASE NOTE: Lines within quotes are unix commands.

8.2 CONTENTS OF GPBEST-DCE SYSTEM

This distribution contains four sub-directories. The variable DCE_ARCH is the executing host's architecture which is one of the following:

HPPA9 - Hewlett Packard - HP-9000 PA-RISC 9.xx
HPPA10 - Hewlett Packard - HP-9000 PA-RISC 10.xx
RS6K - IBM/RS6000 - AIX
SGI5 - Silicon Graphics - IRIX 5.x
SGI64 - Silicon Graphics - IRIX 6.4
SUN4SOL2 - Sun 4, SPARCstation - Solaris 5.x

Note that the files in the 'examples/dce' sub-directory are 'compressed' to save space.

To uncompress the files, use the following commands:
`uncompress filename.Z` for a single file
or
`uncompress *.Z` for all files in the directory

1. bin/DCE_ARCH — Contains executables named **best-dce** , **dce-slave** and **dce-gs.**

best-dce — Main **GPBEST-DCE** executable for distributed computing environment analyses.

best-slave — Slave **GPBEST-DCE** executable for distributed computing environment analyses.

dce-gs — **GPBEST-DCE** group functions and communication executable for distributed computing environment analyses.

2. lib — Contains networking daemon and setup files.

dce-arch — Executable unix shell script sets the DCE_ARCH environment variable to the DCE architecture abbreviation of the executing host for distributed computing environment analyses.

dce-d — Executable shell script which determines the executing hosts architecture for the networking daemon and communication executable.

dce-cshrc — This text file must be customized if **GPBEST-DCE**

installation does not reside in /usr/local/gpbest.

3. lib/DCE_ARCH — Contains networking daemon and setup files.

dce-d — **GPBEST-DCE** networking daemon and communication executable for specific DCE_ARCH architecture.

dce-gs — **GPBEST-DCE** group functions and communication executable for specific DCE_ARCH architecture.

4. examples — Contains subdirectories which contain examples of **GPBEST-DCE** data (*.dat) and result (*.res) files.

There is one directory under examples:

dce.summary: Contains a tabular representation of the features covered in all **GPBEST-DCE** examples.

examples/dce: Contains examples for **GPBEST-DCE** execution.

8.3 GPBEST-DCE SYSTEM REQUIREMENTS

GPBEST-DCE .rhosts File

All **GPBEST-DCE** available hosts must be included in a .rhosts file that resides in the users $HOME directory on each available machine. This file will be used when the standard unix commands ‘‘rsh’’ and/or ‘‘remsh’’ are used to remotely start the **dce-d** daemon. The contents of this file are normally the complete internet host’s name followed by the user name. An example .rhosts file:

```
hal.gpbest.com mlgreen
ice.bestview.com mlgreen
bills.bestview.com mlgreen
bugs.bestview.com mlgreen
isaac.gpbest.com mlgreen
```

You can check if this is set correctly by using the following command to and from all available hosts you wish to include in your host pool.

```
rsh remote_host_complete_internet_name ls
        or
remsh remote_host_complete_internet_name ls
```

This command should output a listing of your $HOME directory on the remote machine. If an error message appears, check the .rhosts file on the remote machine.

GPBEST-DCE User Names

In order for **GPBEST-DCE** to start the remote **dce-d** daemon, the user name on the remote machine must be the same as the local machine. This will be the default setting of **GPBEST-DCE** and may be changed to support different user login names in future versions.

I/O at Login Time

When starting remote **dce-d** daemons, the user must eliminate any I/O in your .cshrc file. This I/O will interfere with the startup of the DCE system. In order to print, put commands in an ‘‘if’’ statement to print out only when an interactive login session is activated.

An example:

```
if ( { tty -s && $?prompt } ) then
    echo user name is $LOGIN
    who
endif
```

GPBEST-DCE dce-cshrc File

GPBEST-DCE must set some specific environment variables in order to determine the executing host architecture and locate the specific executable files. From the Installation Chapter, the following lines must be added to the .cshrc files located in the $HOME (login home directory) directory on each available host the user wishes to include in the host pool.

```
if ( -f /usr/local/gpbest/lib/dce-cshrc ) then
    source /usr/local/gpbest/lib/dce-cshrc
else
    echo GPBEST-DCE environment not properly setup
    echo Correct path or file permissions on dce-cshrc file
    echo Refer to Installation Chapter 8.2 GPBEST Users Manual
endif
```

Note: if the user has not installed **GPBEST-DCE** in the /usr/local/gpbest directory, this path must be changed to the full path location of the gpbest directory. Correct the following "**full_path_location** ".

```
if ( -f full_path_location/gpbest/lib/dce-cshrc ) then
    source full_path_location/gpbest/lib/dce-cshrc
else
    echo GPBEST-DCE environment not properly setup
    echo Correct path or file permissions on dce-cshrc file
    echo Refer to Installation Chapter 8.2 GPBEST Users Manual
endif
```

The dce-cshrc file also must be corrected with the "**full_path_location**".

```
if (! $?DCE_ROOT) then
    if ( -d full_path_location/gpbest ) then
        setenv DCE_ROOT full_path_location/gpbest
    else
        echo DCE_ROOT not defined
        echo To use DCE, define DCE_ROOT and rerun your .cshrc
    endif
endif
```

```
if ($?DCE_ROOT) then
    setenv DCE_ARCH ‘$DCE_ROOT/lib/dce-arch‘
    set path=( $path $DCE_ROOT/bin/$DCE_ARCH $DCE_ROOT/lib )
endif
```

8.4 GUIDE TO USING GPBEST-DCE

The Distributed Computing Environment System

Distributed Computing Environment (DCE) is an integrated set of software tools and libraries that form a flexible concurrent computing framework on networked computers of the same or different architectures. The objective of the DCE system is to dynamically connect computers to form a concurrent or parallel computational resource. The user configures a host pool that can consist of single- and/or multi-processor machines which may be used to exploit the capabilities of each specific machine by positioning more computationally demanding tasks on the most appropriate computers. The DCE system supports heterogeneity in terms of machines and networks.

The DCE system is a daemon, called **dce-d** , that resides on all computers making up the host pool. **dce-d** is designed so that any user with a valid login can install this daemon on a machine and configure a parallel computational resource. This daemon will never actually be executed directly by the user, but it will be called from within the **best-dce** executable. Multiple users can form overlapping parallel computational configurations, and each user can execute **best-dce** simultaneously. BUT, **GPBEST-DCE** LIMITS THE USE OF ONE PARALLEL COMPUTATIONAL CONFIGURATION PER USER AT ONE TIME ON A SPECIFIC MACHINE.

General Performance Considerations

In general, application performance is dependent on the form of parallelism used. There are three main forms which can be used: functional, data, and a mixed data-functional parallelism.

We define functional parallelism to be different machines within the DCE configuration performing different tasks. For example, a vector supercomputer may solve a part of a problem suited for vectorization, a multiprocessor may solve a part of the problem that is suited to parallelization, and a graphics workstation may be used to visualize the generated data in real time. Therefore each machine performs different functions.

In a data parallelism application, the data is partitioned and distributed to all the machines in the DCE configuration. Operations are performed on each set of data and information is passed until the problem is solved. Data parallelism is very popular on distributed-memory multiprocessors because one general parallel program can be executed on all of the processors. Data parallelism can also be very scalable with near perfect speedup on embarrassingly parallel problems (problems which require no communication or synchronization).

Both of these forms of parallelism can be mixed in a hybrid that exploits the strengths of each machine as well as partitioning data between machines.

The original **GPBEST** code implemented the data parallelism form by decomposing Geometric Modeling Regions (GMRs) into an interface residual system. The natural progression

of this work would be to decompose these independent GMRs on a parallel platform. In order to add as much flexibility as possible, **GPBEST** has been integrated within a heterogeneous DCE configuration. The concepts of data parallelism and heterogeneous DCE configurations are important when considering how to increase the performance of **GPBEST-DCE** on specific applications.

First, consider the effects of having a different computational power on each machine in the configuration. This can be due to having different brands of workstations where there may be orders of magnitude difference in power or having a heterogeneous collection of machines in the DCE configuration which differ in their computational rates. Even a homogeneous configuration of machines will not have exactly the same computational rates due to multitasking of other users on all or a subset of the machines. If the user divides his problem into identical GMRs, one for each machine, then the above considerations will adversely effect the applications performance. The application will run as slow as the task on the slowest machine. Therefore, the user must keep in mind the diversity of the DCE configuration when dividing his problem into GMRs. In general, if all machines enrolled in DCE have essentially the same computational rate, all the GMRs should essentially have the same number of degree of freedom (DOF). As the diversity increases with respect to computational rates, the GMRs should reflect varying DOF in an attempt to match higher computational rate machines with larger DOF GMRs.

DCE Installation Checking

GPBEST-DCE performs extensive installation checking upon startup to ensure proper execution. Refer to Chapter 8.3 **GPBEST-DCE** System Requirements and follow all instructions carefully or **GPBEST-DCE** will not execute.

First the environment variables that **GPBEST-DCE** needs to configure and execute are DCE_ARCH, DCE_ROOT, and DCE_VERSION. The DCE_ARCH environment variable is required to determine which architecture-dependent executable should be used for each available host machine. The DCE_ROOT environment variable is defined as the top level directory where the /bin and /lib **GPBEST-DCE** executables are located. Note: DCE_ROOT can be defined wherever the user desires as long as each available host machine also has the same DCE_ROOT pathname defined. The executable **GPBEST-DCE** files must also reside in the pre-defined directory structure that is distributed from the Boundary Element Software Technology Corporation. The DCE_VERSION environment variable must be defined as the current **GPBEST-DCE** version installed to eliminate mixing of executables from different version numbers.

The user must also set the environment variable PATH to reflect the addition of the **GPBEST-DCE** executable directory locations. This will automatically be set when the user sources the dce-cshrc file located in the **GPBEST-DCE** distribution directory gpbest/lib/ . Note: this file will need modifications if the user does not use the default **GPBEST-DCE** installation provided or the user does not use the csh as a login shell.

There are many options available to the user on how and where to execute **GPBEST-DCE** , but one requirement which is essential to **GPBEST-DCE** execution is the ability for the user to have read and write permissions in all directories specified for execution of **GPBEST-DCE** . This includes the directory in which the command line execution of **GPBEST-DCE** occurs. **GPBEST-DCE** communicates with the operating system from this directory and often reads and writes temporary working files in this directory. Subsequently **GPBEST-DCE** will check the current working directory for read and write permissions before proceeding with the current **GPBEST-DCE** execution. This applies to all SCRAtch, MAIN, and RESULT directories that are supplied to **GPBEST-DCE** .

Process Priority Level

In a multiprocessing system, processes must be scheduled. UNIX uses a priority-based algorithm intended to provide good throughput for processes that need extensive use of the CPU while being fair to those processes that do I/O primarily. Priorities are computed dynamically so that no information about the future needs of the process must be predicted by the user of the process or the operating system.

There are many different algorithms for non-real-time scheduling, but their general behavior is similiar. For our purpose, the algorithm we consider here assigns to each process both a static and a dynamic priority (the priority is the sum of the dynamic and static priorities). In an unusual terminology, the lower the value of this combined priority, the more likely the process is to get access to the CPU from the scheduler. A user has no real control of the computation of the dynamic priority other than in the structure and CPU needs of his or her program, so the most obvious strategy in controlling the CPU access of a process is to raise the base priority. We have provided the capability to do this by using the PRIOrity card.

The PRIOrity value can be any integer value from 0 to 20. A lower PRIOrity value is better than a higher one for increasing CPU access. This capability has been implemented to give the ordinary user the ability to lower the CPU access of his or her process by increasing the PRIOrity value. This may seem counter intuitive but by raising the PRIOrity value other users that are conducting interactive login sessions will be less effected. Background processes can this reason only positive PRIOrity values are allowed in **GPBEST-DCE** .

Down Host Elimination

GPBEST-DCE performs a internet host name and aliveness check on all machines in the user-supplied host pool. The complete internet host name input by the user will be used by the standard unix command ‘‘rup’’ to determine if the host name specified is correct and if the host is alive. The following **GPBEST-DCE** output confirms the host name is correct and that the host is alive and functioning:

```
CHECK ON HOST: hal.gpbest.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING
```

```
CHECK ON HOST: bills.bestview.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING

CHECK ON HOST: isaac.gpbest.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING
```

If the host name is incorrect or available host specified is down, **GPBEST-DCE** will automatically eliminate the down host and output the following message:

```
** WARNING 2100
   CHECK ON HOST: hal.gpbest.com
   WAS NOT SUCCESSFUL. HOST IS NOT FUNCTIONING
   REMOVING HOST FROM AVAILABLE LIST
```

Load Sensing

Normally, computational analysis departments do not purchase identical workstations with the exact same random access memory (RAM), virtual memory configuration, million instructions per second (MIPS) rating and operating system. If the machine model and central processing unit (CPU) are identical, changing the RAM, CPU, virtual memory configuration, etc. can significantly modify the MFLOp rating of a specific machine. Even if a network of identical workstations with the same system parameters exists, the computational rates of individual machines can dynamically change due to multi-tasking and current machine loading. This can lead to a system-wide load imbalance where several users load some machines heavily and other machines are lightly loaded or not loaded at all. In order to perform a system-wide load balance of the locally networked machines, **GPBEST-DCE** has integrated some basic system calls to sense the load on each machine included in the host pool provided by the user. The following table represents a sample output from **GPBEST-DCE** :

```
CURRENT MACHINE USER AND LOADING STATISTICS:
HOSTNAME  1 MINUTE LOAD   5 MINUTE LOAD   15 MINUTE LOAD   # USERS
-----------------------------------------------------------------
hal              .16             .11              .07          1
bills            .00             .00              .00          0
isaac            .00             .01              .02          1


CURRENT MACHINE LOAD RATING:
HOSTNAME  MFLOP RATING  LOAD RATING
-----------------------------------------------------------------
hal               42       39.252
isaac             28       27.451
bills             18       18.000
```

By using the standard unix command ‘‘rup’’, **GPBEST-DCE** determines the one, five, and fifteen minute CPU loading on each of the machines provided by the user. The number of users currently logged in on each machine is also determined by the standard unix command ‘‘rusers’’.

The user must also input a "**courtesy factor** " which weights the number of logged in users more or less compared to the current machine loading.

Computer	Manufacture Mflop/s	GPBEST-DCE Mflop/s
IBM RS/6000-340 (42 MHz)	15	15
IBM RS/6000-590 (66 MHz)	130	120
IBM RS/6000-390 (67 MHz)	53	42
HP 9000/735 (99 MHz)	42	28
SUN Sparc10/51 (50 MHz)	10	10
SGI-INDIGO 2 (100 MHz)	15	15
DEC ALPHA 3000-300 (133 MHz)	25	25

Table 8.3 Popular MFLOp Ratings

Using this information, along with the machine-dependent unloaded MFLOp rating supplied by the user (refer Table 8.3), **GPBEST-DCE** can determine a "**load rating factor"** from the following formula:

$$\text{Load Rating Factor} = \frac{\text{MFLOP Rating}}{(0.1 * \text{FACTOR} * \#\ \text{USERS}) + (1 + \text{CURRENT 5 MIN LOAD})}$$

The **load rating factor** represents the unloaded MFLOp rating divided by a weighted combination of logged-in users and current machine loading. Also note that the **load rating factor** includes the addition of 1 to the current load which represents the newly spawned **GPBEST-DCE** process. Essentially, the **load rating factor** shows the amount of MFLOp power that the specified machine can allocate to a **GPBEST-DCE** process.

Considering the heterogeneous mixture of workstations and supercomputers that most computational engineering departments use, the above load-sensing scheme can determine which machines are best suited to execute **GPBEST-DCE** processes. If all local network machines that are licensed to execute **GPBEST-DCE** are included in the host pool designated by the user, **GPBEST-DCE** will automatically determine the appropriate hosts to solve multiple GMR **GPBEST** problems. For multiple **GPBEST-DCE** users including all locally networked machines in their host pools, **GPBEST-DCE** will perform system-wide load balancing when **GPBEST** jobs are submitted.

Multiple Processes on GPBEST-DCE Hosts

In today's rapidly expanding computer frontier, Multi-processor or high performance RISC based computers are common place. **GPBEST-DCE** can better utilize these resources by executing multiple processes on these specific hosts. The ordinary user can now place the burden of which machines to use and how many processes to execute in the hands of **GPBEST-DCE** .

An example of how to implement the load balancing capabilities of **GPBEST-DCE** will help the user understand the basic concepts here.

Example 1: The computational mechanics lab of interest has two Multi-processor machines and three high performance RISC machines. The following table represents the individual machine specifications:

MACHINE NAME	NUMBER OF PROCESSOR	MFLOP RATING	RAM MEMORY	PRIMARY FUNCTION
moriarity	6	85	3 GIG	Time Share Machine
gomez	4	57	1 GIG	Time Share Machine
byte	1	150	1 GIG	User Workstation
hal	1	42	256 M	User Workstation
isaac	1	65	512 M	User Workstation

The following file is a sample **GPBEST-DCE** input file for this lab:

```
**DCE
HOST moriarity
SCRAtch /tmp/mlgreen/MORIARITY
MFLOp 85
MEMOry-dce 200
MULTiple 4

HOST gomez
SCRAtch /tmp/mlgreen/GOMEZ
MFLOp 57
MEMOry-dce 256
MULTiple 2

HOST byte
SCRAtch /tmp/mlgreen/BYTE
MFLOp 150
MEMOry-dce 128
MULTiple 2
PRIOrity 15

HOST hal
SCRAtch /tmp/mlgreen/HAL
MFLOp 42
MEMOry-dce 64
MULTiple 1
PRIOrity 20

HOST isaac
SCRAtch /tmp/mlgreen/ISAAC
MFLOp 65
MEMOry-dce 64
MULTiple 1
PRIOrity 18

MAIND /moriarity/mlgreen/MAINRUN1

RESULTD /moriarity/mlgreen/RESRUN1
```

```
MAXTask 5
```

The user can save this file as his or her standard **GPBEST-DCE** configuration file and append it to a valid **GPBEST** analysis data file to run **GPBEST-DCE** . As we can see from the above **GPBEST-DCE** data file, the user has given the essential parameters necessary for **GPBEST-DCE** to compute a LOAD RATING table for this specific **GPBEST-DCE** configuration. A typical analysis would look like this:

```
CURRENT MACHINE USER AND LOADING STATISTICS:
HOSTNAME  1 MINUTE LOAD   5 MINUTE LOAD   15 MINUTE LOAD   # USERS
-----------------------------------------------------------------
moriarity     5.16            3.11             2.07          3
gomez         2.16            1.11             1.07          2
byte          1.56            1.21             1.02          1
hal            .00             .00              .00          0
isaac          .00             .11              .03          1


CURRENT MACHINE LOAD RATING:
HOSTNAME  MFLOP RATING  LOAD RATING
-----------------------------------------------------------------
byte             150       99.338
isaac             65       63.107
byte             150       59.761
moriarity         85       56.013
hal               42       42.000
gomez             57       37.134
moriarity         85       33.767
moriarity         85       24.165
gomez             57       22.485
moriarity         85       18.816
```

Since the **GPBEST-DCE** data includes the MAXTask card the program will only execute processes on the five highest load rating machines. **GPBEST-DCE** computes the load ratings as a function of the current lab system loadings again in choosing the appropriate host to decompose the residual system.

Load Imbalance

In many parallel applications, it is impossible to predict the size of the subtasks assigned to various machines. Hence, the problem cannot be subdivided statically among the machines while maintaining a uniform work load. If different machines have different work loads, some machines may be idle during part of the time that others are working on the problem.

Some or all machines must synchronize at certain points during the parallel program execution. If all machines are not ready for synchronization at the same time, then ones that are ready sooner will be idle until the rest are ready. Whatever the cause of idling, the total idle time of all the machines contributes to the parallel overhead.

When computational loads are varying, a dynamic load balancing scheme is required. The most popular method is called the Pool of Tasks paradigm. **GPBEST-DCE** has implemented this paradigm using a pool of GMRs which are queued in order of priority to respective idle hosts. The pool of GMRs is sorted by DOF size, and larger GMRs are assigned higher priority numbers. With this method, all of the slave hosts are kept busy as long as there are GMRs left in the pool. By efficiently distributing the GMRs in this fashion, idling time is minimized when all hosts synchronize for the residual system solution and back substitution.

GPBEST-DCE Queuing Host Capability

In the event that the user wants to use a low-power machine to queue a **GPBEST-DCE** job to other high-power workstations or supercomputers, the user must execute the **best-dce** command line on the machine which is choosen to be the queuing host. This host must also be included in the user-input ‘‘HOST’’ card, even though this host will only time and queue the **GPBEST-DCE** job. By using a queuing host, the user can include his/her desktop workstation in the **GPBEST-DCE** execution without actually decomposing GMRs but it may serve the MAIN directory disk space and move GMR scratch files.

Examples of **GPBEST-DCE** queuing host capabilities.

1) The user has a low-power desktop workstation ‘‘byte.gpbest.com’’ that serves the ‘‘/byte’’ disk space. The queuing host is designated to be ‘‘byte.gpbest.com’’ by adjusting the unloaded MFLOp rating to 0, thereby insuring a low-load rating factor. The ‘‘LIMIt’’ card will be used to determine the best three hosts to execute **GPBEST-DCE** .

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE

  HOST sparc.bestview.com
  SCRAtch /sparc/mlgreen/SCRATCH2
  MFLOp 35

  HOST hal.gpbest.com
  SCRAtch /run/mlgreen/HAL
  MFLOp 42

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29

  HOST byte.gpbest.com
  SCRAtch /byte/mlgreen/BYTE
  MFLOp 0

  RESULT /run/mlgreen/RESULT

  MAIN /byte/mlgreen/MAIN
```

```
LIMIt 3
```

GPBEST-DCE command line execution

```
cd /run/mlgreen/MAIN
  < gpbest.dat best-dce   >& /byte/mlgreen/RESULT/gpbest.out &
```

2) A user has three high-powered machines with very limited scratch disk space, but also has a large disk served by a desktop machine (byte.gpbest.com) and wishes to solve a large **GPBEST** problem without using all of the scratch disk space of the high-powered machines. The low-powered desktop machine will queue the **GPBEST-DCE** job to the high-powered machines and serve all of the disk space.

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /byte/mlgreen/HAL
  MFLOp 42

  HOST isaac.gpbest.com
  SCRAtch /byte/mlgreen/ISAAC
  MFLOp 29

  HOST ice.bestview.com
  SCRAtch /byte/mlgreen/ICE
  MFLOp 21

  HOST byte.gpbest.com
  SCRAtch /byte/mlgreen/BYTE
  MFLOp 0

  RESULT /byte/mlgreen/RESULT

  MAIN /byte/mlgreen/MAIN

  LIMIt 3
```

GPBEST-DCE command line execution

```
cd /byte/mlgreen/MAIN
  < gpbest.dat best-dce   >& /byte/mlgreen/RESULT/gpbest.out &
```

Timing and Speedup Calculations

The serial run time of a program is the time elapsed between the beginning and the end of its execution on a sequential computer. To calculate the serial run time during a parallel execution, the individual GMR decomposition wallclock times are summed to yield a combined total. The total time calculated with this method represents the serial run time for a sequential implementation of a problem. The parallel run time is the time that elapses from the moment that a parallel

computation starts to the moment the the last processor finishes execution. The parallel run time will also include any added setup and computation required for the parallel algorithm.

When evaluating a parallel system, we are often interested in knowing how much performance gain is achieved by parallelizing a given problem over a sequential implementation. Speedup is a measure that captures the relative benefit of solving a problem in parallel. It is defined as, ‘‘the ratio of the time taken to solve a problem in a serial mode to the time required to solve the same problem in a parallel or DCE mode.’’ For a given problem, more than one serial run time can be calculated, but it is impractical to run serial and DCE mode timings to calculate the speedup. When calculating a serial run time, it is natural to use the sequential algorithm that solves the problem in the least amount of time. Given a parallel algorithm, it is fair to judge its performance with respect to the fastest sequential algorithm for solving the same problem. Since the fastest sequential algorithm is not known, we will use the calculated serial run time as our practical choice. We can then calculate the speedup and compare the performance of the parallel system to the serial mode for solving the same problem. Theoretically, speedup can never exceed the number of machines solving the problem in DCE mode.

During program execution, all GMR decomposition wallclock times are recorded. Serial and DCE mode execution times are calculated, along with speedup, as each GMR decomposition is completed. The estimated time savings of the parallel algorithm compared to the sequential algorithm is also calculated as GMRs are completed. The following summary table is written to the DCE execution log file designated by the user on the command line execution of **GPBEST-DCE** .

REGION	SERIAL MODE TIME	DCE MODE TIME	SPEEDUP	TIME SAVINGS
DCELOG.DATCHECK	.8	1.0	.810	-.2
DCELOG.A103	124.3	124.5	.998	-.2
DCELOG.A105	268.7	145.5	1.847	123.2
DCELOG.A102	451.4	183.8	2.457	267.7
DCELOG.A104	513.3	211.5	2.427	301.8
DCELOG.A107	552.4	225.5	2.450	326.9
DCELOG.A106	567.4	228.5	2.483	338.9
DCELOG.A101	668.9	237.8	2.813	431.1

NOTE: TIME UNIT MINUTES

GPBEST-DCE Resource and Directory Structure

GPBEST-DCE directory and file structure is more complex than serial mode **GPBEST** execution. In the previous **GPBEST** serial code, all scratch and result files were contained in one directory (current working directory where the command line is executed). A fast scratch directory is normally choosen to increase the performance of the **GPBEST** serial and reduce the network loading. This system works very well when executing on a single host.

GPBEST-DCE

The main objective of **GPBEST-DCE** is to minimize network loading and utilize fast scratch disk space to increase performance. A pictogram representation of a general **GPBEST-DCE** directory structure is presented in Figure 8.1. The Current Working Directory (CWD) upon command line execution of **best-dce** is the default directory for the MAIN and RESULT directories.

The MAIN working directory is used for storage of GMR scratch files that contain the decomposed solution matrix and will generally require a majority of the disk storage for the **GPBEST-DCE** execution. The MAIN working directory should be located on the server or disk served by a machine that is not included in the available hosts input by the user. This will eliminate the load due to serving disk space from the available hosts in the DCE system. The MAIN directory must also have a network connection to all available host scratch directories specified by the user. In other words, /tmp and other local directories which are only accessible on the local machine are not valid host scratch directories. The MAIN directory must have a network connection with the RESULT directory if the user chooses to specify one. The decomposed GMR scratch files are copied to the MAIN directory from all host scratch working directories as they are completed. The MAIN directory must support two-way system I/O with the host scratch directories (problem data file must be copied to all host scratch directories and GMR file must be copied from all host scratch directories). The host scratch directories are cleaned after each GMR decomposition to reduce the amount of disk space required in solving multiple GMR problems. In general, the host scratch working directories should be fast disk access with enough disk space to decompose either the largest GMR or solve the residual system, whichever requires more space. The host scratch working directories do not need to be mounted on all available hosts since there are no transfers from one host to another.

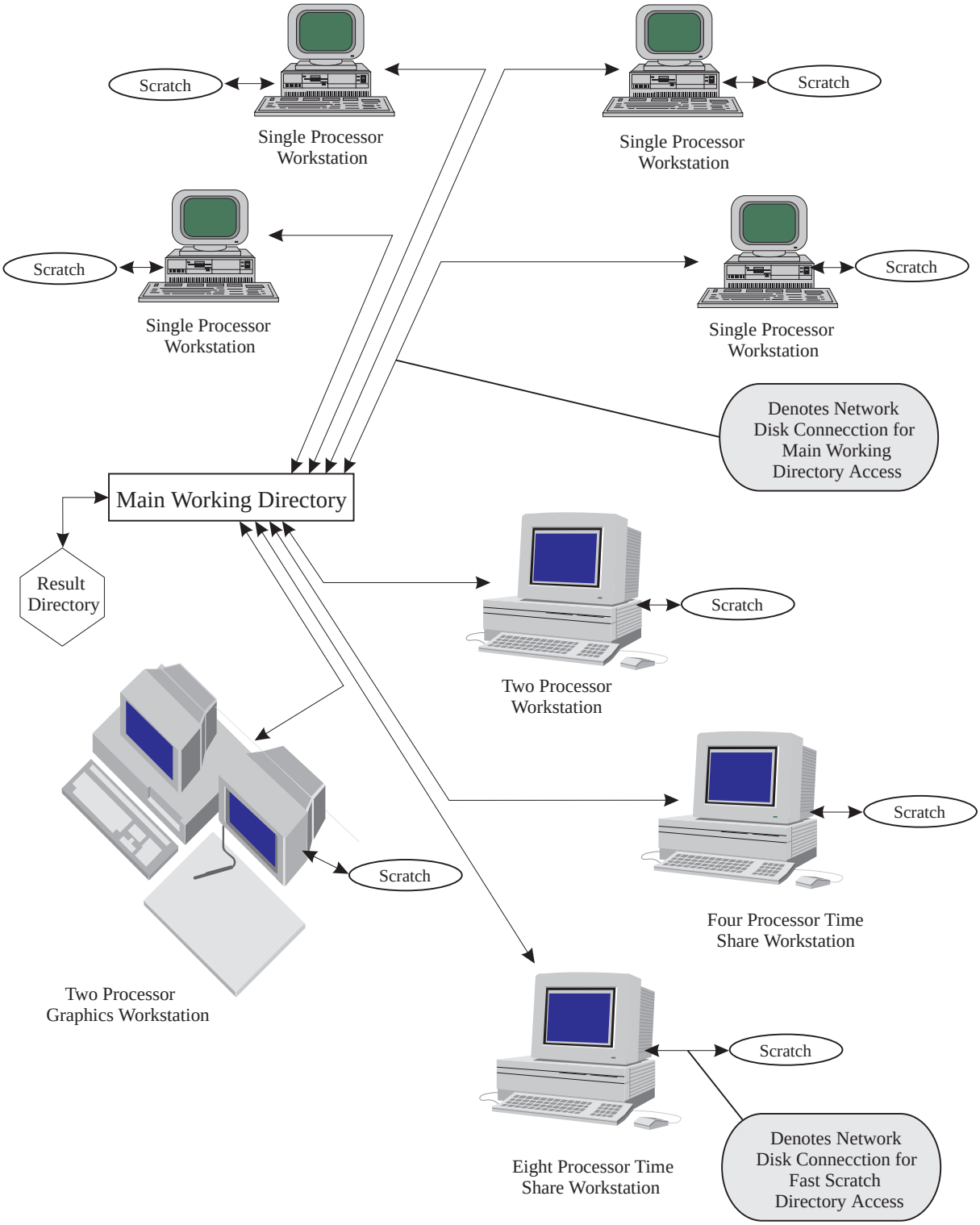


Figure 8.1: GPBEST-DCE Resource and Directory Structure Overview

The RESULT directory contains the final **GPBEST-DCE** GMR and result log files. If the user chooses to use the RESULT input card, the RESULT directory must have a network connection to the MAIN working directory. The default directory is the CWD upon command line execution.

Examples of **GPBEST-DCE** MAIN and RESULT directory use:

1) The user specifies a RESULT directory and uses the CWD default value for MAIN working directory, but would also like the command line log and result file are to be located in the RESULT directory.

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /run/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 256

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29
  MEMOry-dce 164

  RESULT /run/mlgreen/RESULT.test1
```

GPBEST-DCE command line execution

```
 < test1.dat best-dce   >& /run/mlgreen/RESULT.test1/test1.out &
```

2) The user uses the default CWD value for the RESULT directory and specifies the MAIN working directory. The command line log and result file are to be located in the RESULT directory.

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /run/mlgreen/HAL
  MFLOp 42

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29

  MAIN /run/mlgreen/SERVER
```

GPBEST-DCE command line execution

```
cd /run/mlgreen/SERVER
 < test2.dat best-dce   >& test2.out &
```

The host SCRAtch directory contains the current GMR decomposition scratch files. The SCRAtch directory receives all **GPBEST-DCE** slave direct output and stores the current problems data file. **GPBEST-DCE** uses a unique file handle which appends the process identification number to all **GPBEST-DCE** files. However, the **GPBEST-DCE** slaves use standard **GPBEST** naming protocol. When the user specifies a SCRAtch directory for each available host, there are several criteria that must be met:

1) The SCRAtch directory for any host must not be the MAIN directory
2) The SCRAtch directory for all hosts must be unique
3) The SCRAtch directory for all hosts must only be used for the current problem (also true for MAIN directory)

Each user can configure ONLY one DCE system at a time but can execute on as many as 40 hosts simultaneously. Many users may potentially use the same fast scratch disk on several machines. To eliminate the problem of overwriting **GPBEST-DCE** slave scratch decomposition files, the user simply needs to create a unique scratch working directory for his/her problem files.

Examples of SCRAtch directory use.

1) The user specifies the MAIN working directory, executes **GPBEST-DCE** on two hosts with unique scratch directories (/run/mlgreen/HAL and /isaac/mlgreen/ISAAC).

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /run/mlgreen/HAL
  MFLOp 42
  MEMOry-dce 256

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29

  MAIN /run/mlgreen/ELAS001
```

GPBEST-DCE command line execution

```
cd /run/mlgreen/ELAS001
  < elas001.dat best-dce   >& elas001.out &
```

2) The user configures two separate DCE systems and runs two **GPBEST-DCE** jobs concurrently.

JOB 1

GPBEST-DCE executes on two hosts, specify MAIN directory, specify RESULT directory.

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST hal.gpbest.com
  SCRAtch /run/mlgreen/HAL
  MFLOp 42

  HOST isaac.gpbest.com
  SCRAtch /isaac/mlgreen/ISAAC
  MFLOp 29
  MEMOry-dce 164

  MAIN /run/mlgreen/JOB1

  RESULT /run/mlgreen/RESULT.job1
```

GPBEST-DCE command line execution

```
cd /run/mlgreen/JOB1
  < job1.dat best-dce   >& /run/mlgreen/RESULT.job1/job1.out &
```

JOB 2

GPBEST-DCE executes on two hosts, specify MAIN directory, specify RESULT directory.

GPBEST-DCE input data contained in **GPBEST** data file

```
**DCE
  HOST bills.bestview.com
  SCRAtch /bills/mlgreen/BILLS
  MFLOp 19

  HOST ice.bestview.com
  SCRAtch /ice/mlgreen/ICE
  MFLOp 21

  MAIN /run/mlgreen/JOB2

  RESULT /run/mlgreen/RESULT.job2
```

GPBEST-DCE command line execution

```
cd /run/mlgreen/JOB2
  < job2.dat best-dce   >& /run/mlgreen/RESULT.job2/job2.out &
```

Note: This type of execution is valid since the working hosts are enrolled in different DCE systems. The DCE systems for each user cannot have overlapping hosts. When multi-users have DCE systems configured on the same hosts, be sure that the host scratch working directories are unique when jobs are executed.

The RESIDUAL system is solved by symbolically linking the decomposed GMR scratch files from the MAIN directory to the host scratch working directory. This eliminates copying the GMR scratch files to the host scratch directory and also increases the performance of **GPBEST** by

keeping the residual system scratch files on fast disk space. **GPBEST-DCE** chooses the host for solving the residual system by load-sensing the DCE system when all the GMRs have been decomposed. The host with the highest load rating according to the load rating formula presented in Section 5.8 ‘‘COURtesy’’ card is designated as the Residual Host. The Residual Host solves the residual system and returns the solution.

8.5 INSTRUCTIONS FOR USE

GPBEST-DCE

To execute **best-dce** as a background process, type in this c-shell

```
< data best-dce > & result &
```

Please Note:

In order to run **GPBEST-DCE** , you must have write permission in all of the working directories specified in the data file and the directory where the command line is executed.

Also, you must have read permission to the /gpbest/password file. This file is read in each time **GPBEST-DCE** is executed and contains **GPBEST-DCE** license information.

Refer to Section 5.8 for GPBEST-DCE input card definitions and examples.

SECTION	KEYWORD	PURPOSE
5.8.1 Definition of **GPBEST-DCE** Input Card		
	**DCE	
	HOST	defines required card for host-specific data
	MFLOp	defines required card for host-specific mega flop rating
	SCRAtch	defines required card for host-specific scratch working directory
	COURtesy	defines optional card for user's courtesy factor
	LIMIt	defines optional card for number of working hosts
	MAINdir	defines optional card for main working directory
	MAXTask	defines optional card for maximum number of allowable processes for the DCE configuration
	MEMOry-dce	defines optional card for host-specific dynamic memory allocation value
	MULTiple	defines optional card for host-specific number of processes
	PASSword	defines optional card for the **GPBEST-DCE** password file path and name
	PRIOrity	defines optional card for host-specific priority value
	RESUlt	defines optional card for result directory

8.6 GENERAL LIMITS OF GPBEST-DCE

It should be noted that there are certain limits which must be observed in the preparation of input for **GPBEST-DCE** . The present limits are summarized below. These limits will be improved in future versions of **GPBEST-DCE,** as computer workstation technology improves further.

ENTITY[1]	
GLOBAL PARAMETERS	
total geometric modeling regions	80
total degrees of freedom	80000
total residual degrees of freedom	20000
total geometric points (includes non-source points)	54000
total boundary source points	20000
total boundary elements	18000
total line elements	400
total enclosing elements	100
total discontinuity elements	100
REGION (GMR) PARAMETERS	
surfaces in any region	15
degrees of freedom in any region	12000
point sources/forces in any region	25
OTHER PARAMETERS	
time points for elastic analysis	20
table points for boundary condition and body force definition	20
temperature points for material properties	21
USER-SPECIFIED NUMBERING	
points	99999
elements	99999

[1] Definition of the terminology used in this table can be found in Section 4.5.

8.7 GPBEST-DCE STARTUP PROBLEMS

If the DCE system is manually killed or stopped abnormally, then by default **GPBEST-DCE** will not allow another parallel computational configuration to be started until the faulted configuration is removed. Check for the file /tmp/dced.<uid>, where <uid> represents your user identification number. If you do not know your <uid>, it can be found by the following command on most systems.

```
grep user_name /etc/passwd | awk -F: '{ print $3 }'
```

If this file exists, then a faulted DCE system is running and the DCE socket file name is located within this file. If this file is left behind, it will prevent DCE from starting a new **dce-d** daemon. Simply delete this file and remove any **dce-d** daemons that are running to recover from an abnormally stopped **GPBEST-DCE** job. You can find any **dce-d** daemon processes identification number <pid> and kill it by using the following commands.

```
ps -ef | grep dce-d | awk '{ print $2 }'
kill -9 <pid>
                or
ps -aux | grep dce-d | awk '{ print $2 }'
kill -9 <pid>
```

Repeat the process for all hosts that were configured when the DCE system abnormally exited.

8.8 GPBEST-DCE EXAMPLE OUTPUT

To illustrate the correct procedure for suitably substructuring a **GPBEST-DCE** problem, Figure 4.8 in the **GPBEST** User Manual is analyzed. This problem contains approximately 24000 total DOF and is substructured into a seven-region model. Each region contains approximately 3500 to 4000 DOF and has been specifically chosen to optimize the **GPBEST** analysis.

The following **GPBEST-DCE** output log is annotated to provide a general user reference for **GPBEST-DCE** operational logs.

```
=================================================================

                      G P B E S T - D C E

       GENERAL PURPOSE BOUNDARY ELEMENT SOLUTION TECHNIQUE
                DISTRIBUTED COMPUTING ENVIRONMENT

                          VERSION 1.0
                            AUG 1995


       PROPERTY OF THE B.E.S.T. CORPORATION
       TO BE USED WITH PERMISSION ONLY
       ALL RIGHTS RESERVED

=================================================================




-----------------------------------------------------------------
            ****            DCE INPUT            ****
-----------------------------------------------------------------


                        ECHO OF DCE INPUT

 LINE            INPUT
    1     **DCE
    2       HOST hal.gpbest.com
    3       SCRATCH /run/mlgreen/HAL
    4       MFLOP 42
    5
    6       HOST bills.bestview.com
    7       SCRATCH /bills/mlgreen/BILLS
    8       MFLOP 18
    9
   10       HOST isaac.gpbest.com
   11       SCRATCH /isaac/mlgreen/ISAAC
```

```
   12       MFLOP 28
```

In this example, the user, ‘‘mlgreen’’ has choosen three machines for his host pool and used each machine’s fast scratch disk for decomposition of problem GMRs.

```
-----------------------------------------------------------------
            ****  START DCE SETUP MODE  ****
-----------------------------------------------------------------


SETTING MAIN WORKING DIRECTORY TO CURRENT
WORKING DIRECTORY: /run/mlgreen/MAIN

SETTING RESULT DIRECTORY TO CURRENT
WORKING DIRECTORY: /run/mlgreen/MAIN

CHECK ON HOST: hal.gpbest.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING

CHECK ON HOST: bills.bestview.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING

CHECK ON HOST: isaac.gpbest.com
WAS SUCCESSFUL. HOST IS ALIVE AND FUNCTIONING
```

The MAIN and RESULT directoriens are set by default to the CWD upon command line execution of **best-dce.**Each host included in the host pool is checked for correct internet host name and alive status.

```
CHECKING FOR EXISTING DCE CONFIGURATION AND
INITIALIZING SYSTEM FOR CURRENT DCE SETUP
```

The DCE system checks for an existing DCE configuration and initializes the current DCE setup.

```
CURRENT MACHINE USER AND LOADING STATISTICS:
HOSTNAME  1 MINUTE LOAD   5 MINUTE LOAD   15 MINUTE LOAD   # USERS
-----------------------------------------------------------------
hal              .16             .11              .07          1
bills            .00             .00              .00          0
isaac            .00             .01              .02          1

CURRENT MACHINE LOAD RATING:
HOSTNAME  MFLOP RATING  LOAD RATING
-----------------------------------------------------------------
hal               42       39.252
isaac             28       27.451
bills             18       18.000
```

The current 1, 5, and 15 machine loads and number of users per machine are compiled. According to the load rating factor formula, **GPBEST-DCE** calculates and sorts the host pool machines in descending order of load rating.

```
COPYING GPBEST DATA FILE: DATA.14653
TO WORKING DIRECTORY: /run/mlgreen/HAL

COPYING GPBEST DATA FILE: DATA.14653
TO WORKING DIRECTORY: /isaac/mlgreen/ISAAC

COPYING GPBEST DATA FILE: DATA.14653
TO WORKING DIRECTORY: /bills/mlgreen/BILLS
```

GPBEST-DCE selects the working hosts (which by default includes all hosts in the host pool) and copies the **GPBEST** data file to each working host's scratch directory.

```
ENROLLING MASTER PROGRAM IN DCE
MASTER PROGRAM TASK IDENTIFICATION NUMBER :  262145

-------------------------------------------------------------------
     ****  CURRENT DISTRIBUTED COMPUTING CONFIGURATION  ****
-------------------------------------------------------------------

THERE ARE  3 HOSTS CURRENTLY ENROLLED

THE PRESENT CONFIGURATION IS:

 # |        HOSTNAME              |  ARCH  |    TID
-------------------------------------------------------------------
  1 hal                            HPPA       262144
  2 isaac.gpbest.com               RS6K       524288
  3 bills.bestview.com             SGI        786432

CHECK ON DCE INPUT DATA COMPLETE
ALL HOSTS ARE NOW PRESENT IN DCE CONFIGURATION
```

The DCE system is started and the current distributed computing configuration is checked.

```
-------------------------------------------------------------------
       ****  START DATA CHECK FOR GPBEST FILES  ****
-------------------------------------------------------------------


SPAWNED GPBEST TASK FOR DATA CHECK
ON HOST: hal.gpbest.com
TID NUMBER:  262146

GPBEST DATA CHECK COMPLETE

COPYING DCELOG.DATCHECK TO MAIN DIRECTORY

DATA CHECK FOR GPBEST SUCCESSFUL

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES
-------------------------------------------------------------------
   1             3327        DCELOG.A101
   2             4167        DCELOG.A102
   3             3561        DCELOG.A103
   4             3309        DCELOG.A104
```

```
   5             4755        DCELOG.A105
   6             1914        DCELOG.A106
   7             2412        DCELOG.A107
```

GPBEST-DCE executes a data check on the best host (the host with the highest load rating factor) and presents the problem's GMR DOF and log names.

```
SORTING REGIONS IN DESCENDING DOF ORDER AND % WORKLOAD FIGURES

PRIORITY #    GMR DOF    GMR FILE LOG NAMES    % WORKLOAD
-----------------------------------------------------------------
   1             4755        DCELOG.A105          33.689
   2             4167        DCELOG.A102          22.673
   3             3561        DCELOG.A103          14.150
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197

NUMBER OF REGIONS IS GREATER THAN NUMBER OF HOSTS
A DYNAMIC LOADING SCHEME IS IMPLEMENTED
```

The GMRs are sorted with respect to the percent of total workload they represent.

```
CURRENT REGIONS BEING PROCESSED:
     HOSTNAME                |     GMR NAME
-----------------------------------------------------------------
hal.gpbest.com                 DCELOG.A105
isaac.gpbest.com               DCELOG.A102
bills.bestview.com             DCELOG.A103

NUMBER OF REGIONS COMPLETE:  0 OUT OF  7

PRIORITY #    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-----------------------------------------------------------------
   1             4755        DCELOG.A105          33.689
   2             4167        DCELOG.A102          22.673
   3             3561        DCELOG.A103          14.150
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197
```

The current GMRs being processed by each **GPBEST-DCE** machine and the completed GMRs are listed for the current problem.

```
REGION DECOMPOSITION COMPLETE
ON HOST: bills.bestview.com
TID NUMBER:  786433
GMR NAME: DCELOG.A103

COPYING REGION FILES TO MAIN DIRECTORY

SPAWNED GPBEST TASK FOR GMR DECOMPOSITION
ON HOST: bills.bestview.com
```

```
TID NUMBER:  786434
GMR NAME: DCELOG.A101
```

The GMR file log name DCELOG.A103 was completed on host bills.bestview.com and the GMR file log name DCELOG.A101 was spawned to bills.bestview.com.

```
CURRENT REGIONS BEING PROCESSED:

     HOSTNAME                |     GMR NAME
-------------------------------------------------------------------
hal.gpbest.com                 DCELOG.A105
isaac.gpbest.com               DCELOG.A102
bills.bestview.com             DCELOG.A101

NUMBER OF REGIONS COMPLETE:  1 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-------------------------------------------------------------------
   1             4755        DCELOG.A105          33.689
   2             4167        DCELOG.A102          22.673
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197

REGION DECOMPOSITION COMPLETE
ON HOST: hal.gpbest.com
TID NUMBER:  262148
GMR NAME: DCELOG.A105

COPYING REGION FILES TO MAIN DIRECTORY

SPAWNED GPBEST TASK FOR GMR DECOMPOSITION
ON HOST: hal.gpbest.com
TID NUMBER:  262149
GMR NAME: DCELOG.A104

CURRENT REGIONS BEING PROCESSED:

     HOSTNAME                |     GMR NAME
-------------------------------------------------------------------
hal.gpbest.com                 DCELOG.A104
isaac.gpbest.com               DCELOG.A102
bills.bestview.com             DCELOG.A101

NUMBER OF REGIONS COMPLETE:  2 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-------------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197
```

```
REGION DECOMPOSITION COMPLETE
ON HOST: isaac.gpbest.com
TID NUMBER:  524289
GMR NAME: DCELOG.A102

COPYING REGION FILES TO MAIN DIRECTORY

SPAWNED GPBEST TASK FOR GMR DECOMPOSITION
ON HOST: isaac.gpbest.com
TID NUMBER:  524290
GMR NAME: DCELOG.A107

CURRENT REGIONS BEING PROCESSED:

     HOSTNAME                |     GMR NAME
-----------------------------------------------------------------
hal.gpbest.com                 DCELOG.A104
isaac.gpbest.com               DCELOG.A107
bills.bestview.com             DCELOG.A101

NUMBER OF REGIONS COMPLETE:  3 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-----------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673    X
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197

REGION DECOMPOSITION COMPLETE
ON HOST: hal.gpbest.com
TID NUMBER:  262149
GMR NAME: DCELOG.A104

COPYING REGION FILES TO MAIN DIRECTORY

SPAWNED GPBEST TASK FOR GMR DECOMPOSITION
ON HOST: hal.gpbest.com
TID NUMBER:  262150
GMR NAME: DCELOG.A106

CURRENT REGIONS BEING PROCESSED:

     HOSTNAME                |     GMR NAME
-----------------------------------------------------------------
hal.gpbest.com                 DCELOG.A106
isaac.gpbest.com               DCELOG.A107
bills.bestview.com             DCELOG.A101

NUMBER OF REGIONS COMPLETE:  4 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-----------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673    X
```

```
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354    X
   6             2412        DCELOG.A107           4.397
   7             1914        DCELOG.A106           2.197

REGION DECOMPOSITION COMPLETE
ON HOST: isaac.gpbest.com
TID NUMBER:  524290
GMR NAME: DCELOG.A107

COPYING REGION FILES TO MAIN DIRECTORY

NUMBER OF REGIONS COMPLETE:  5 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-------------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673    X
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354    X
   6             2412        DCELOG.A107           4.397    X
   7             1914        DCELOG.A106           2.197

REGION DECOMPOSITION COMPLETE
ON HOST: hal.gpbest.com
TID NUMBER:  262150
GMR NAME: DCELOG.A106

COPYING REGION FILES TO MAIN DIRECTORY

NUMBER OF REGIONS COMPLETE:  6 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-------------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673    X
   3             3561        DCELOG.A103          14.150    X
   4             3327        DCELOG.A101          11.540
   5             3309        DCELOG.A104          11.354    X
   6             2412        DCELOG.A107           4.397    X
   7             1914        DCELOG.A106           2.197    X

REGION DECOMPOSITION COMPLETE
ON HOST: bills.bestview.com
TID NUMBER:  786434
GMR NAME: DCELOG.A101

COPYING REGION FILES TO MAIN DIRECTORY

NUMBER OF REGIONS COMPLETE:  7 OUT OF  7

GMR NUMBER    GMR DOF    GMR FILE LOG NAMES   % WORKLOAD COMPLETE
-------------------------------------------------------------------
   1             4755        DCELOG.A105          33.689    X
   2             4167        DCELOG.A102          22.673    X
   3             3561        DCELOG.A103          14.150    X
```

```
   4            3327        DCELOG.A101          11.540    X
   5            3309        DCELOG.A104          11.354    X
   6            2412        DCELOG.A107           4.397    X
   7            1914        DCELOG.A106           2.197    X


WORKLOAD DISTRIBUTION FOR THIS DCE CONFIGURATION:
     HOSTNAME                |   PERCENT PROCESSED
-------------------------------------------------------------------
hal.gpbest.com                   47.240
isaac.gpbest.com                 27.070
bills.bestview.com               25.690

ALL REGIONS ARE COMPLETE
PROCEED SOLVING THE RESIDUAL SYSTEM
```

GPBEST-DCE has completed all GMRs for the current problem and reported workload distribution for this analysis.

```
                   SERIAL        DCE                        TIME
     REGION      MODE TIME    MODE TIME     SPEEDUP       SAVINGS
-------------------------------------------------------------------
DCELOG.DATCHECK         .8          1.0        .810           -.2
DCELOG.A103          124.3        124.5        .998           -.2
DCELOG.A105          268.7        145.5       1.847         123.2
DCELOG.A102          451.4        183.8       2.457         267.7
DCELOG.A104          513.3        211.5       2.427         301.8
DCELOG.A107          552.4        225.5       2.450         326.9
DCELOG.A106          567.4        228.5       2.483         338.9
DCELOG.A101          668.9        237.8       2.813         431.1

NOTE: TIME UNIT MINUTES
```

The timing, speedup and time savings for the current run were reported as **GPBEST-DCE** completed regions.

```
CURRENT MACHINE USER AND LOADING STATISTICS:
HOSTNAME  1 MINUTE LOAD   5 MINUTE LOAD   15 MINUTE LOAD   # USERS
-------------------------------------------------------------------
hal             2.05            1.79             1.32          0
isaac            .00             .06              .31          1
bills            .06             .36              .51          0

CURRENT MACHINE LOAD RATING:
HOSTNAME  MFLOP RATING  LOAD RATING
-------------------------------------------------------------------
isaac             28       21.374
hal               42       18.103
bills             18       11.921
```

GPBEST-DCE determined the host isaac.gpbest.com was the best suited host for the residual system solution and releases the remaining hosts from the **GPBEST-DCE** configuration.

```
CLEANING HOST: hal SCRATCH DIRECTORY
```

```
RELEASING HOST: hal FROM CURRENT DCE CONFIGURATION

CLEANING HOST: bills SCRATCH DIRECTORY

RELEASING HOST: bills FROM CURRENT DCE CONFIGURATION

-------------------------------------------------------------------
      **** START RESIDUAL SOLUTION FOR REGIONS  ****
-------------------------------------------------------------------

SPAWNED GPBEST TASK FOR RESIDUAL SYSTEM
ON HOST: isaac.gpbest.com
TID NUMBER:  524291

LINKING GMR FILES TO RESIDUAL HOST WORKING DIRECTORY

-------------------------------------------------------------------

GPBEST IS OPERATING IN A DISTRIBUTED COMPUTING ENVIRONMENT MODE


DCE PHASE 3: DECOMPOSITION OF RESIDUAL MATRIX AND OUTPUT OF SOLUTION
```

GPBEST-DCE output log is complete; the standard GPBEST residual matrix solution continues from here.

9.0 REFERENCES

1. BANERJEE, P.K., and BUTTERFIELD, R., (1975), Boundary Element Method in Geomechanics, Chapter 16 in Finite Element in Geomechanics, Ed. G. Gudehus, John Wiley.

2. BANERJEE, P.K., and BUTTERFIELD, R., (1981), Boundary Element Methods in Engineering Science, McGraw-Hill, London, Russian Translation, Mir Publishers, 1984, Moscow; Chinese Translation, National Defense Press, 1988, Beijing.

3. BANERJEE, P.K., (1994), The Boundary Element Methods in Engineering, McGraw-Hill, London.

4. BREBBIA, C.A., TELLES, J.C.F., and WROBEL, L.C., (1984), Boundary Element Techniques, Springer-Verlag.

5. LIGGETT, J., and LIU, P., (1983), Boundary Integral Equation for Porous Media Flow, Allen and Unwin, London.

6. CROUCH, S.L., and STARFIELD, A.M., (1983), Boundary Element Methods in Solid Mechanics, Allen and Unwin, London.

7. MUKHERJEE, S., (1982), Boundary Element Methods in Creep and Fracture, Applied Science Publishers, London.

8. BANERJEE, P.K., and BUTTERFIELD, R., Editors, (1979), Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking, Essex, U.K.

9. BANERJEE, P.K., and SHAW, R.P., Editors, (1982), Developments in Boundary Element Methods II, Elsevier Applied Publishers, Barking, Essex, U.K.

10. BANERJEE, P.K., and MUKHERJEE, S., Editors, (1984), Developments in Boundary Element Methods III, Elsevier Applied Science Publishers, Barking, Essex, U.K.

11. BANERJEE, P.K., and WATSON, J.O., Editors, (1986), Developments in Boundary Element Methods IV, Elsevier Applied Science Publishers, Barking, Essex, U.K.

12. BANERJEE, P.K., and WILSON, R.B., Editors, (1989), Industrial Applications of BEM, Elsevier Applied Science Publishers, Barking, Essex, U.K.

13. CRUSE, T.A., PIFKO, A., and ARMEN, H., Editors, (1985), Advanced Topics in Boundary Element Analysis, AMD-Vol. 72, ASME, New York.

14. RIZZO, F.J., and SHIPPY, D.J., (1977), An advanced boundary integral equation method for three-dimensional thermo-elasticity, Int. Jour. Numer. Methods in Engng., Vol. 11, pp. 1753-1768.

15. RIZZO, F.J., and SHIPPY, D.J., (1979), Recent advances of the boundary element method in thermoelasticity, Chapter VI in Banerjee, P.K. and Butterfield, R. (Eds.) Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking, Essex, U.K.

16. ZIENKIEWICZ, O.C., (1977) Finite Element Method in Engineering Science, McGraw Hill, London.

17. CRUSE, T.A., and WILSON, R.B., (1978), Boundary-integral equation methods for elastic fracture mechanics analysis, AFOSR-TR-77-0355.

18. BANERJEE, P.K., CATHIE, D.N., and DAVIES, T.G., (1979), Two and three dimensional problems of elastoplasticity, in Developments in Boundary Element Methods I, Banerjee, P.K. and Butterfield, R. (Eds.), Elsevier Applied Science Publishers, Barking, Essex, U.K.

19. BANERJEE, P.K., and CATHIE, D.N., (1980), A direct formulation and numerical implementation of the boundary element method for two-dimensional problems of elastoplasticity, Int. Jour. Mechanical Sciences, Vol. 22, pp. 233-245.

20. CATHIE, D.N., and BANERJEE, P.K., (1980), Boundary element methods in axisymmetric plasticity, Innovative Numerical Analysis for the Applied Engineering Sciences, Shaw, R.P., et al (Eds.), University of Virginia Press.

21. CATHIE, D.N., and BANERJEE, P.K., (1982), Boundary element methods for plasticity and creep including a viscoplastic approach, Res. Mechanica, Vol. 4, pp. 3-22.

22. KUMAR, V., and MUKHERJEE, S., (1977), A boundary-integral equation formulation for time-dependent inelastic deformation in metals, Int. Jour. Mechanical Sciences, Vol. 19, Vol. 713-724.

23. BANERJEE, P.K., and DAVIES, T.G., (1984), Advanced implementation of boundary element methods for three-dimensional problems of elastoplasticity and viscoplasticity, Chapter I in Developments in Boundary Element Methods III, Applied Science Publishers, London.

24. BANERJEE, P.K., and WILSON, R.B., (1984), Fundamental solutions of the elasticity equations for distributed hot spots and cavities, Unpublished work.

25. BANERJEE, P.K., and RAVEENDRA, S.T., (1986), Advanced boundary element method for two and three-dimensional problems of elastoplasticity, Int. Jour. Numer. Methods in Engng., Vol. 23, pp. 985-1002.

26. BANERJEE, P.K., and AHMAD, S., (1985), Advanced three- dimensional dynamic analysis by boundary element methods, AMD-Vol. 72, pp. 65-82, ASME, New York.

27. BANERJEE, P.K., AHMAD, S., and MANOLIS, G.D., (1986), Transient elastodynamic analysis of three-dimensional problems by boundary element method, Earthquake Engng. and Struct. Dynamics, Vol. 14, pp. 933-949.

28. AHMAD, S., and BANERJEE, P.K., (1988), Time domain transient elastodynamic analysis of 3-D solids by BEM, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 1709-1728.

29. AHMAD, S., and BANERJEE, P.K., (1986), Free-Vibration analysis by BEM using particular integrals, Journal of Engineering Mechanics, ASCE, Vol. 112, No. 7, pp. 682-695.

30. MANOLIS, G.D., and BANERJEE, P.K., (1986), Conforming and non-conforming boundary elements in 3-D elastostatics, Int. Jour. Numer. Methods in Engng., Vol. 23, pp. 1885-1904.

31. LACHAT, J.C., and WATSON, J.O., (1976), Effective numerical treatment of boundary integral equations: A formulation for three-dimensional elastostatics, Int. Jour. Numer. Methods in Engng., Vol. 10, pp. 991-1005.

32. WATSON, J.O., (1979), Advanced implementation of the boundary element method for two and three-dimensional elastostatics, in Banerjee, P.K. and Butterfield, R. (Eds.) Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking Essex, U.K., pp. 31-64.

33. BANERJEE, P.K., WILSON, R.B., and MILLER, N., (1985), Development of a large BEM system for three-dimensional inelastic analysis, AMD-Vol. 72, Edited by Cruse, T.A., Pifko, A., and Armen, H., pp. 1-20.

34. WILSON, R.B., SNOW, D.W., and BANERJEE, P.K., (1985), Stress analysis of gas turbine engine structures using boundary element methods, AMD-Vol. 72, pp. 45-64.

35. BANERJEE, P.K., WILSON, R.B., and MILLER, N., (1988), Advanced elastic and inelastic stress analysis of gas turbine engine structures by BEM, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 393-411.

36. BANERJEE, P.K., WILSON, R.B., and RAVEENDRA, S.T., (1987), Advanced applications of BEM to three-dimensional problems of monotonic and cyclic plasticity, Int. Jour. Mechanical Sciences, Vol. 29, No.9, pp. 637-653.

37. BANERJEE, P.K. and RAVEENDRA, S.T., (1987), A new boundary element formulation for two-dimensional elastoplastic analysis, Journal of Engineering Mechanics, ASCE, Vol. 113, No.2, pp. 252-265.

38. HENRY, D., PAPE, D. and BANERJEE, P.K., (1987), New axisymmetric BEM formulation for body forces using particular integrals, Journal of Engineering Mechanics, ASCE, Vol. 113, No.5, pp. 671-688.

39. HENRY, D. and BANERJEE, P.K., (1987), A thermoplastic BEM analysis for substructured axisymmetric bodies, Journal of Engineering Mechanics, ASCE, Vol. 113, No. 12, pp. 1880-1900.

40. HENRY, D. and BANERJEE, P.K., (1988), A variable stiffness type boundary element formulation for axisymmetric elastoplastic media, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 1005-1027.

41. BANERJEE, P.K., AHMAD, S. and CHEN, K. (1987), Advanced application of BEM to wave barriers in multi-layered three-dimensional soil media, Earthquake Engng. and Struct. Dynamics, Vol. 16, pp. 1041-1060.

42. DARGUSH, G. F. and BANERJEE, P.K. (1989), Advanced BEM for steady state heat tranfer analysis, Int. Jour. Num. Methods in Engng., Vol. 28, pp. 2113-2142.

43. BANERJEE, P.K., HENRY, D. and RAVEENDRA, S.T. (1989), Advanced BEM for inelastic analysis of solids, Int. Jour. Mechanical Sciences, Vol. 31, pp. 309-322.

44. DARGUSH, G. F. and BANERJEE, P.K. (1989), Development of BEM for transient poroelasticity, Int. Jour. Numer. Methods in Engng., Vol. 28, pp. 2423-2449.

45. DARGUSH, G. F. and BANERJEE, P.K. (1989), The boundary element method for plane problems of thermoelasticity, Int. Jour. Solids and Structures, Vol. 25, pp. 999-1021.

46. DARGUSH, G. F. and BANERJEE, P.K. (1990), BEM analysis for three-dimensional problems of transient thermoelasticity, Int. Jour. Solids and Structures, Vol. 26, pp. 199-216.

47. BANERJEE, P.K., AHMAD, S. and WANG, H.C. (1988), A new BEM for the acoustic eigenfrequency analysis, Int. Jour. Numer. methods in Engng., Vol. 26, pp. 1299-1309.

48. WANG, H.C. and BANERJEE, P.K. (1987), Axisymmetric free-vibration analysis by BEM, Journal of Applied Mechanics, Vol. 55, pp. 437-444.

49. WANG, H.C. and BANERJEE, P.K. (1990), Multi-domain Generalized axisymmetric analysis by BEM, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 985-1001.

50. HENRY, D. and BANERJEE, P.K. (1988), A new boundary element formulation for two and three-dimensional problems of thermoelasticity using particular integrals, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 2061-2077.

51. HENRY, D. and BANERJEE, P.K. (1988), A new boundary element formulation for two and three-dimensional problems of elastoplasticity using particular integrals, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 2079-2096.

52. MROZ, Z. (1967) On the description of anisotropic work hardening, Jour. Mech. Phys. Solids, Vol. 15, pp. 163-175.

53. KRIEG, R.D. (1975) A practical two surface plasticity theory, Journal of Applied Mechanics, ASME, Vol. E42, pp. 641-646.

54. DAFALIAS, Y.F. and POPOV, E.P. (1974) A model of nonlinearly hardening material for complex loadings, Proc. 7th U.S. National Congress in applied Mechanics, Boulder, Acta Mech., Vol. 21, pp. 173-192.

55. ROSCOE, K.H. and BURLAND, J.B. (1968) On the generalized stress-strain behavior of wet clay, Engineering Plasticity, Hayman, J.V. and Leckie, F.A., eds., Cambridge University Press, pp. 535-609.

56. BANERJEE, P.K. and YOUSIF, N.B. (1986) A plasticity model for the anisotropically consolidated clay, Int. Jour. Numer. Analyt. Methods in Geomechanics, Vol. 10, pp. 521-541.

57. WALKER, K.P. (1981) Research and Development Program for Nonlinear Structural Modeling with Advanced Time-Temperature Dependent Constitutive Relationships, NASA CR-165533.

58. CASSENTI, B.N. and THOMPSON, R.L. (1983) Material Response Predictions for Hot Section Gas Turbine Engine Components, AIAA-83-2020, presented at the AIAA/SAE/ASME 19th Joint Propulsion Conference, Seattle, Washington, June 27-29.

59. AHMAD, S. and BANERJEE, P.K. (1988), Multi-domain BEM for two-dimensional problems of elastodynamics, Int. Jour. Numer. Methods in Engng., Vol. 26, pp. 891-911.

60. WANG, H.C. and BANERJEE, P.K. (1989), General axisymmetric periodic dynamic stress analysis by BEM, Int. Jour. Numer. Methods in Engng., Vol. 28, pp. 2065-2083.

61. WANG, H.C. and BANERJEE, P.K. (1990), Axisymmetric transient elastodynamics analysis by BEM, Int. Jour. Solids and Structures, Vol. 26, pp. 401-415.

62. PAPE, D. and BANERJEE, P.K. (1987), Treatment of body forces in 2D elastostatic BEM using particular integrals, Journal of Applied Mechanics, Vol. 54, pp. 871-886.

63. WILSON, R.B., MILLER, N.M. and BANERJEE, P.K. (1990), Calculations of natural frequencies and mode shapes for three-dimensional structures by BEM, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 1737-1757.

64. BANERJEE, P.K., STIPHO, A.S. and YOUSIF, N.B. (1984), A theoretical and experimental investigation of the behavior of anisotropically consolidated clay, Chapter I in Development in Soil Mech. Foundation Engrg., Vol. 2, Elsevier Applied Science, London.

65. DARGUSH, G and BANERJEE, P.K. (1988), Development of an Integrated BEM for Hot Fluid Structure Interaction, Earth to Orbit Propulsion Systems, NASA Conf. Publication No. 3012.

66. DARGUSH, G. and BANERJEE, P.K. (1989), Developments of an integrated BEM for hot fluid structure interaction, Gas Turbine and Aeroengine Congress, ASME, Toronto, Canada, also Journal of Engineering for Gas Turbine and Power, Trans ASME, Vol. 112, pp. 243-250, April 1990.

67. ZIENKIEWICZ, O.C., BAUER, J., MORGAN, K. and ONATE, E. (1977), A simple and efficient element for axisymmetric shells, Int. Jour. Numer. Methods in Engng., Vol. 11, pp. 1545-1558.

68. DELPAK, R. (1975), Role of the Curved Parametric Element in Linear Analysis of Thin Rotational Shells, Ph.D. Thesis, Department of Civil Engineering and Building, The Polytechnic of Wales.

69. GIANNI, M. and MILES, G.A. (1970), A curved element approximation in the analysis of axisymmetric thin shells, Int. Jour. Numer. Methods in Engng., Vol. 2, pp. 459-476.

70. MUKHERJEE, S. and PODDAR, B., An Integral Equation Formulation for Elastic and Inelastic Shell Analysis, Proceedings of the International Conference, Beijing, China, (October 1986).

71. KRAUS, H. (1967), Thin Elastic Shells, John Wiley and Sons, New York, pp. 445-447.

72. BANERJEE, P.K. and HENRY, D. (1991), Analysis of three-dimensional solids with holes by BEM, Int. Jour. Numer. Methods in Engng, Vol. 31, pp. 369-384.

73. BANERJEE, P.K. and HENRY, D. (1992), Analysis of 3-dimensional solids with inclusions by BEM, Int, Jour. Solids Struct., Vol.29. pp. 2423-2440.

74. LEISSA, A. and ZHANG, Z. (1983), On the three-dimensional vibrations of the cantilevered rectangular parallelepiped, Jour. of the Acoustical Soc. of America., Vol. 73, No. 6, pp. 2013-2021.

75. MacBAIN, J.C. and KIELB, R.E., and LEISSA, A.W. (1984), Vibrations of twisted cantilevered plates - experimental investigation, ASME Paper 84-GT-96.

76. KIELB, R.E., LEISSA, A.W., MacBAIN, J.C. and CARNEY, K.S. (1985), Joint Research Effort on Vibrations of Twisted Plates - Phase I: Final Results, NASA Reference Publication No. 1150.

77. WANG, H.C. and BANERJEE, P.K. (1990), Generalized axisymmetric free-vibration analysis by BEM, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 985-1001.

78. DARGUSH, G. and BANERJEE, P.K. (1991), Application of the Boundary Element Method to Transient Heat Conduction, Int. Jour. Numer. Methods in Engng., Vol. 31.

79. CARLSLAW, H.S. and JAEGER, J.C. (1947), Conduction of Heat in Solids, Clarendon Press, Oxford.

80. HAJI-SHEIKH, A. and SPARROW, E.M. (1966), Transient heat conduction in a prolate spheroidal solid, Journal of Heat Transfer, ASME, Vol. 88, pp. 331-333.

81. ISRAIL, A.S.M. and BANERJEE, P.K. (1990), Advanced development of time-domain BEM for two-dimensional scalar wave propagation, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 1003-1020.

82. SHUKU, T. and ISHIHARA, K (1973), The analysis of the acoustic field in irregularly shaped room by the finite element method, Journal of Sound and Vibration, Vol. 29, pp. 67-76.

83. NEFSKE, D. J., WOLF, J. A. and HOWELL, L. J. (1982), Structural-acoustic finite element analysis of the automobile passenger compartment: a review of current practice, Journal of Sound and Vibration, Vol. 80, pp. 247-266.

84. ISRAIL, A.S.M. and BANERJEE, P.K. (1990), Effects of geometrical and material properties on the vertical vibration of 3-D foundations by BEM, Int. Jour. Numer. Analyt. Methods in Geomechanics, Vol. 14, pp. 49-70.

85. WOODS, R. D. (1968), Screening of surface waves in soils, Journal of Soil Mech. and Found. Division, ASCE, Vol. 94, SM 4, July, pp. 951-979.

86. CHEN, C.H. and PENZIEN, J. (1986), Dynamic modelling of axisymmetric foundations, Earthquake Engng. and Struct. Dynamics, Vol. 14, pp. 823-840.

87. BURGGRAF, O. (1966), Analytical and numerical studies of the structure of steady separated flows, Journal of Fluid Mechanics, Vol. 24, Part 1, pp. 113-151.

88. GHIA, U., GHIA, K.N. and SHIN, C.T. (1982), High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, Journal of Comp. Physics, Vol. 48, pp. 387-411.

89. DARGUSH, G. and BANERJEE, P.K. (1989), Advanced Development of BEM for Steady Incompressible Thermoviscous Flow, Chapter III in Development in BEM, Vol. 5, Elsevier Applied Science Publishers.

90. MILLSAPS, K. and POHLHAUSEN, K. (1953), Thermal distributions in Jeffery-Hamel flows between nonparallel plane walls, Journal of Aeronautical Sciences, March, pp. 187-196.

91. GARTLING, D.K., NICKELL, R.E. and TANNER, R.E. (1977), A finite element convergence study for accelerating flow problems, Int. Jour. Numer. Methods in Engng., Vol. 11, pp. 1155-1174.

92. ECKERT, E.R.G. and SOEHNGEN, E. (1952), Distribution of heat-transfer coefficients around circular cylinders in crossflow at Reynolds numbers from 20 to 500, Transactions of the ASME, Vol. 74, pp. 343-347.

93. SZABO, B.A. (1988), Solution of electrostatic problems by the p-version of the finite element method, Communications in Applied Numer. Methods, Vol. 4, pp. 565-572.

94. COTTRELL, P.E. and BUTURLA, E.M. (1985), VLSI wiring capacitance, IBM Journal of Research and Development, Vol. 29, No. 3, pp. 277-288.

95. ISRAIL, A.S.M. and BANERJEE, P.K., (1990), Advanced time-domain formulation of BEM for two-dimensional transient elastodynamics, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 1421-1440.

96. AHMAD, S. and BANERJEE, P.K. (1990), Inelastic transient dynamic analysis of three-dimensional problems by BEM, Int. Jour. Numer. Methods in Engng., Vol. 29, pp. 371-390.

97. ISRAIL, A.S.M. and BANERJEE, P.K. (1992), Advanced development of boundary element method for two-dimensional dynamic elasto-plasticity, Int. Jour. Solids and Structures, Vol. 29, No. 11, pp. 1433-1451.

98. ISRAIL, A.S.M. and BANERJEE, P.K. (1991). Interior stress calculations in 2-D time-domain transient BEM analysis, Int. Journal of Solids and Structures, Vol. 28, No. 7, Feb 1991.

99. CRUSE, T.A. (1973). Application of the boundary integral equation method to three-dimensional stress analysis, Computers and Structures, Vol. 3, pp. 509-527.

100. LEKHNITSKII, S.G. (1968). Anisotopic Plates, Gordon and Breach Science Publishers.

101. MALEN, K. (1971). A unified six-dimensional treatment of elastic Green's functions and dislocations, Phys. Stat. Sol., Vol. 44, pp. 661-672.

102. NOWACKI, W. (1986). Thermoelasticity, 2 edn., PWN - Polish Scientific Publishers, Warszawa.

103. PADOVAN, J. (1986). Anisotropic thermal stress analysis, in R.B. Hetnarski (ed.), Thermal Stresses I, Elsevier Science Publishers B.V., Amsterdam, pp. 143-262.

104. SNYDER, M.D. and CRUSE, T.A. (1975). Boundary-integral equation analysis of cracked anisotropic plates, Int. Jour. Fracture, Vol. 11, pp. 315-328.

105. SYNGE, J.L. (1957). The Hypercircle in Mathematical Physics, Cambridge University Press, pp. 411-413.

106. DEB, A., HENRY, D. and WILSON, R.B. (1990). An alternate BEM for anisotropic problems of thermoelasticity, Int. Jour. Solids Structures, Vol.27, No.13, pp. 1721-1738.

107. WILSON, R.B. and CRUSE, T.A. (1978). Efficient implementation of anisotropic three-dimensional boundary integral equation stress analysis, Int. J. Numer. Methods in Engng., Vol. 12, pp. 1383-1397.

108. ISRAIL, A.S.M. and DARGUSH, G.F. (1991). Dynamic fracture mechanics studies by time-domain BEM, Engineering Fracture Mechanics, Vol. 39, pp. 315-328.

109. BLANDFORD, G.E., INGRAFFEA, A.R. and LIGGET, J.A. (1981). Two-dimensional stress intensity factor computations using the boundary element method, Int. Journal for Numerical Methods in Engineering, Vol. 17, pp. 387-404.

110. DEB, A., Advanced Development of BEM for Linear and Nonlinear Analyses of Anisotropic Solids, Ph.D. dissertation, State University of New York at Buffalo, Buffalo, NY, USA.

111. HILL, R. (1981). The Mathematical Theory of Plasticity, Oxford University Press.

112. TSAI, S. W. and WU, E. M. (1971). A general theory of strength for anisotropic materials, Journal Composite Materials, Vol. 5, pp. 387-404.

113. KENAGA, D., DOYLE, D. F. and SUN, C. T. (1987). The characterization of B/A1 composite in the nonlinear range as an orthotropic elastic-plastic material, Journal Composite Materials, Vol. 21, pp. 516-531.

114. ZYTINSKI, M., RANDOLPH, M. F. and NOVA, R. (1978). On modelling the unloading-reloading behaviour of soils, Int. Jour. Numer. Analyt. Methods in Geomechanics, Vol. 2, No. 1, pp. 87-93.

115. CHOPRA, M. B. (1992). Development of numerical solutions (FEM and BEM) to problems with material and geometric nonlinearities, Ph.D. dissertation, State University of New York at Buffalo, Buffalo, New York.

116. CHEREPANOV, G. P. (1979), Mechanics of Brittle Fracture, Mc-Graw-Hill, New York.

117. RAVEENDRA, S.T. and CRUSE, T. A. (1989), BEM analysis of problems of fracture mechanics, Developments in Boundary Element Methods-5 (Eds. P. K. Banerjee and R. B. Wilson), Elsevier Applied Science, London, pp. 187-204.

118. RAVEENDRA, S.T. and BANERJEE, P.K. (1991), Computation of stress intensity factors for interfacial cracks, Engrg. Frac. Mech., Vol. 40, No.1, pp. 89-103.

119. RICE, J. R. (1988), Elastic fracture mechanics concepts for interfacial cracks, Transactions of the ASME, 55, pp. 98-103.

120. ISIDA, M. (1981), Rotating disk containing an internal crack located at an arbitrary position, Engng Frac. Mech. 14, pp549-555.

121. LOVE, A. E. H. (1944), A Treatise on the Mathematical Theory of Elasticity, Dover Publications, New York.

122. RAVEENDRA, S.T. and BANERJEE, P.K., (1991), Analysis of rotating solids with cracks by the boundary element method, Int. J. Solids and Structures, Vol.28, No.9, pp 1155-1170.

123. DEB, A. and BANERJEE, P.K., (1989), A comparison between Isoparametric and Lagrangian elements in 2D BEM, Int. Journal for Numerical Methods in Engineering, Vol. 28, pp. 1539-1555.

124. BURTON, A.J. and MILLER, G. F., (1971), The application of Integral Equation Methods to the numerical solution of some Exterior Problems, Proc. Roy. Soc. Ser. A, 323, pp. 201-210.

125. WANG, C. (1995). Advanced development of boundary element methods in material nonlinear analysis, Ph.D. dissertation, State University of New York at Buffalo, Buffalo, New York.

126. ROARK, R.J., and YOUNG, W.C., (1975), Formulas for Stress and Strain, Fifth Edition, McGraw-Hill, New York.

127. BARONE, M.R. and CAULK, D.A., (1985), Special boundary integral equations for approximate solution of potential problems in three-dimensional regions with slender cavities of circular cross-section, IMA J. Appl. Math., 35, pp. 311-325.