INTRODUCTION TO GPBEST: ELASTIC ANALYSIS

A Quick-Start Manual

Includes Instructions for Installing the GPBEST System

June, 1998 **Version 2.5**

Boundary Element Software Technology Corporation

P.O. Box 310 Getzville, NY 14068-0310 Phone: (716) 639-1920
E-Mail: support@gpbest.com Fax: (716) 639-1919

INTRODUCTION TO GPBEST: ELASTIC ANALYSIS

A Quick-Start Manual

Includes Instructions for Installing the GPBEST System

June, 1998 **Version 2.5**

Boundary Element Software Technology Corporation

P.O. Box 310 Getzville, NY 14068-0310 Phone: (716) 639-1920
E-Mail: support@gpbest.com Fax: (716) 639-1919

Please Note: This tutorial manual (**Version 2.5**) corresponds to the *GPBEST User's Manual, Volume 1* **(Version 5.2)** and the *Bestview User's Manual, Volume 2* **(Version 2.5)**.

GPBEST and BESTVIEW are trademarks of the Boundary Element Software Technology Corporation. MSC/PATRAN and MSC/NASTRAN are registered trademarks of the MacNeal-Schwendler Corporation. NASTRAN is a registered trademark of NASA. ABAQUS is a trademark of Hibbitt, Karlsson & Sorensen, Inc. I-DEAS is a trademark of Structural Dynamics Corporation. Pro/Engineer is a trademark of Parametric Technology Corporation. HyperMesh is a trademark of Altair Computing, Inc. FEMAP is a trademark of Enterprise Software Products, Inc. ANSYS is a trademark of Swanson Analysis Systems, Inc. All other company and product names are trademarks or registered trademarks of their respective owners.

Table Of Contents

Page

CHAPTER 1 - INTRODUCING THE GPBEST SYSTEM 1.1
Introduction 1.1
Introduction To The Boundary Element Method 1.1
Comparison Of BEM And FEM 1.1
Basic Theory Of The Boundary Element Method 1.3
Components Of The GPBEST System 1.5
Computer Programs 1.5
Documentation 1.5
Basic Terminology 1.6
What To Do If You Are Having Problems 1.8

CHAPTER 2 - INSTALLATION 2.1
Introduction 2.1
System Requirements 2.1
Note: Kernel Configuration for Hewlett Packard (HP-UX) Computers 2.2
Installation of GPBEST From Tape 2.3
Step 1: Determine Device Name Of Tape Drive 2.3
Step 2: Create A GPBEST Installation Directory And Install 2.4
Step 3: Remove Unnecessary Files 2.4
Step 4: Uncompress The Compressed Files 2.6
Step 5: Copy PATRAN Files (Optional) 2.6
Step 6: GPBEST Password 2.6
Step 7: Modify User Startup Files 2.8
Step 8: Test The GPBEST Installation 2.10
Step 9: Testing The BESTVIEW Installation 2.13
GPBEST File Structure 2.15

CHAPTER 3 - TUTORIAL #1: RUNNING AN EXAMPLE PROBLEM 3.1
Introduction 3.1
Searching The Example Problems 3.1
Looking At The GPBEST Data File 3.5
Running GPBEST 3.8
Checking A Completed Run 3.9
Checking Accuracy Of The Solution 3.12
Submitting GPBEST Jobs 3.13
CHECK Run 3.16
Units And Dimensions In GPBEST 3.17
Finding Other Example Problems 3.18
The GPBEST.FEATURES File 3.18
GPBEST Limits 3.21

CHAPTER 4 - TUTORIAL #2: 2-D PROBLEM 4.1
Introduction 4.1
Problem Specification 4.1
BESTVIEW Model 4.4
Step 1: Start BESTVIEW 4.5
Step 2: Define The Four Corner Points Of The Beam 4.6
Step 3: Define Four Geometrical Lines, One For Each Edge Of The Beam 4.7
Step 4: Define A GROUP 4.8
Step 5: Create A Boundary Element Mesh 4.9
Step 6: Display Boundary Element Mesh 4.9
Step 7: Prepare The GPBEST Data File (PREPARE ANALYSIS) 4.11
Step 8: Define The Boundary Conditions For The Model 4.13
Step 9: Run GPBEST To Analyze The Boundary Element Model 4.15
Step 10: Read The Neutral File Into BESTVIEW For Post-Processing 4.16
Step 11: Create A Deformed Shape Plot 4.16
Step 12: Create A Color Fringe Stress Plot 4.17
PATRAN Model 4.20
Step 1: Start PATRAN 4.20
Step 2: Define The Four Corner Points Of The Beam 4.21
Step 3: Define Four Geometrical Lines, One For Each Edge Of The Beam 4.22
Step 4: Define Groups 4.23
Step 5: Define Material Properties 4.23
Step 6: Create A Boundary Element Mesh 4.24
Step 7: Display Boundary Element Mesh 4.26
Step 8: Define The Boundary Conditions For The Model 4.28
Step 9: Run GPBEST To Analyze The Boundary Element Model 4.31
Step 10: Post-Processing The Results 4.32
Step 11: Create A Deformed Shape Plot 4.34
Step 12: Create A Color Fringe Stress Plot 4.35
I-DEAS Model 4.36
Step 1: Start I-DEAS 4.37
Step 2: Define The Geometry Of The Beam 4.38
Step 3: Define A GROUP 4.39
Step 4: Define Material Properties 4.39
Step 5: Create A Boundary Element Mesh 4.40
Step 6: Display Boundary Element Mesh 4.43
Step 7: Define The Boundary Conditions For The Model 4.44
Step 8: Run GPBEST To Analyze The Boundary Element Model 4.47
Step 9: Post-Processing The Results 4.48
Step 10: Create A Deformed Shape Plot 4.49
Step 11: Create A Color Fringe Stress Plot 4.51
HYPERMESH Model 4.52
Step 1: Start HYPERMESH 4.52
Step 2: Define Material Properties 4.53

Step 3: Define The Nodes of The Beam 4.54
Step 4: Define A GROUP 4.56
Step 5: Create A Boundary Element Mesh 4.56
Step 6: Display Boundary Element Mesh 4.59
Step 7: Define The Boundary Conditions For The Model 4.61
Step 8: Run GPBEST To Analyze The Boundary Element Model 4.65
Step 9: Post-Processing The Results 4.66
Step 10: Create A Deformed Shape Plot 4.67
Step 11: Create A Color Fringe Stress Plot 4.68
FEMAP Model 4.70
Step 1: Start FEMAP 4.70
Step 2: Define The Nodes 4.71
Step 3: Offset the Mid-Nodes 4.73
Step 4: Define A GROUP 4.73
Step 5: Define Material Properties 4.73
Step 6: Create A Boundary Element Mesh 4.74
Step 7: Display Boundary Element Mesh 4.77
Step 8: Define The Boundary Conditions For The Model 4.79
Step 9: Run GPBEST To Analyze The Boundary Element Model 4.81
Step 10: Post-Processing The Results 4.83
Step 11: Create A Deformed Shape Plot 4.84
Step 12: Create A Color Fringe Stress Plot 4.85

CHAPTER 5 - TUTORIAL #3: 3-D PROBLEM 5.1
Introduction 5.1
The GMR Concept 5.1
Overview 5.1
Characteristics Of GMRs 5.4
Reasons For Using Multi-GMR Models 5.5
Types Of Interfaces 5.5
GMR Considerations In Modeling Programs 5.6
Problem Specification 5.7
BESTVIEW Model 5.9
Step 1: Start BESTVIEW 5.9
Step 2: Define Four Points 5.9
Step 3: Define The Geometric Surfaces 5.11
Step 4: Define The Groups 5.14
Step 5: Create A Boundary Element Mesh 5.15
Step 6: Display Boundary Element Mesh 5.16
Step 7: Prepare The GPBEST Data File (PREPARE ANALYSIS) 5.17
Step 8: Define The Boundary Conditions For The Model 5.19
Step 9: Run GPBEST To Analyze The Boundary Element Model 5.21
Step 10: Post-Processing The Results 5.22
PATRAN Model 5.23
Step 1: Start PATRAN 5.23

Step 2: Define Four Points 5.23
Step 3: Define The Geometric Surfaces 5.25
Step 4: Define Material Properties 5.28
Step 5: Create A Boundary Element Mesh 5.28
Step 6: Define The Groups 5.30
Step 7: Display Boundary Element Mesh 5.31
Step 8: Define The Boundary Conditions For The Model 5.32
Step 9: Run GPBEST To Analyze The Boundary Element Model 5.36
Step 10: Post-Processing The Results 5.37
I-DEAS Model 5.39
Step 1: Start I-DEAS 5.39
Step 2: Define The Geometry Of The Beam 5.40
Step 3: Define Material Properties 5.41
Step 4: Create A Boundary Element Mesh 5.42
Step 5: Define The Groups 5.45
Step 6: Display Boundary Element Mesh 5.48
Step 7: Define The Boundary Conditions For The Model 5.49
Step 8: Run GPBEST To Analyze The Boundary Element Model 5.53
Step 9: Post-Processing The Results 5.54
HYPERMESH Model 5.56
Step 1: Start HYPERMESH 5.56
Step 2: Define Material Properties 5.56
Step 3: Define Eight Points 5.57
Step 4: Define The Geometric Surfaces 5.60
Step 5: Create A Boundary Element Mesh 5.62
Step 6: Define The Groups 5.64
Step 7: Display Boundary Element Mesh 5.66
Step 8: Define The Boundary Conditions For The Model 5.67
Step 9: Run GPBEST To Analyze The Boundary Element Model 5.71
Step 10: Post-Processing The Results 5.72
FEMAP Model 5.74
Step 1: Start FEMAP 5.74
Step 2: Create a Volume Defining a Cube 5.74
Step 3: Define Material Properties 5.76
Step 4: Create A Boundary Element Mesh 5.77
Step 5: Define The Groups 5.80
Step 6: Display Boundary Element Mesh 5.82
Step 7: Define The Boundary Conditions For The Model 5.83
Step 8: Run GPBEST To Analyze The Boundary Element Model 5.87
Step 9: Post-Processing The Results 5.88

CHAPTER 6 - ADDITIONAL HELPFUL INFORMATION 6.1
Introduction 6.1
Data Flow In And Out Of GPBEST 6.1
BESTVIEW 6.2

PATRAN 6.5
I-DEAS 6.8
HYPERMESH 6.11
FEMAP 6.14
PRO/Engineer 6.17
IGES 6.20
Other Systems 6.22
Auto-Meshing 6.23

APPENDIX A - REFERENCES A.1
General UNIX Literature A.1
Boundary Element Literature A.1

APPENDIX B - UNITS AND DIMENSIONS B.1
A Consistent Set Of Units B.1
A Table Of Suggested Units B.1
Some Notes About Units And Conversions B.4
Approximate Properties for Steel (including appropriate units) B.6
Approximate Properties for Aluminum (including appropriate units) B.7

APPENDIX C - USEFUL UNIX SCRIPTS C.1
Script: fromto.sh C.1
Script: runbest.sh C.1

APPENDIX D - GPBEST/BESTVIEW PASSWORD REQUEST D.1
GPBEST AND BESTVIEW PASSWORD REQUEST FORM D.2
Example Of The GPBEST password File D.3

APPENDIX E - GPBEST ELASTIC ANALYSIS KEYWORDS E.1
Introduction E.1
Section 5.1 - Case Control E.1
Section 5.2 - Material Property Definition E.4
Section 5.3 - Definition of Geometry E.5
Section 5.4 - Definition of GMR Compatibility (Interfaces) E.7
Section 5.5 - Definition of Boundary Conditions E.8
Section 5.6 - Body Force Definition E.10
Section 5.7 - Definition of Boundary Condition Changes E.11

CHAPTER 1 - INTRODUCING THE GPBEST SYSTEM

Introduction

This manual serves as an introduction to the GPBEST system. It contains all the information that is needed to install GPBEST and start running some basic problems. It will also introduce the various programs included in the GPBEST system.
This chapter presents an overview of boundary element analysis using the GPBEST system. Some users who are familiar with the boundary element method (BEM) and the GPBEST program may opt to skip this chapter in order to begin installation (Chapter 2) or run some simple problems (Chapter 3).

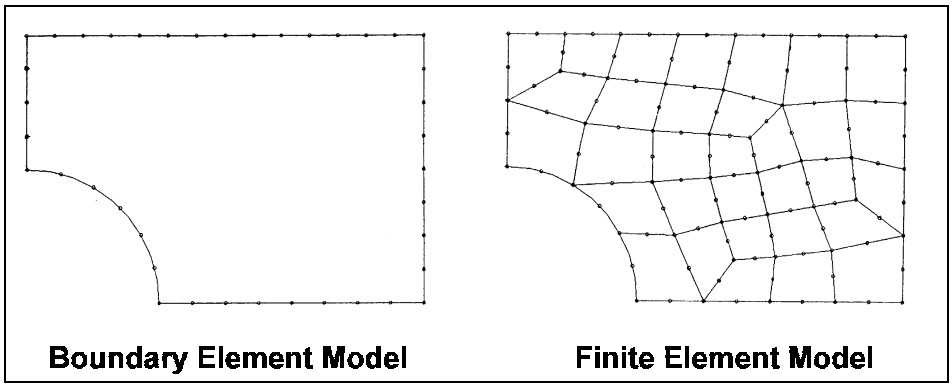
TIP: In this manual, sidebars such as this are used to provide additional information that may be of interest to the user.

Introduction To The Boundary Element Method

The boundary element method (BEM) may be new to you. Most engineers and analysts are familiar with the finite element method (FEM), where a model consisting of nodes and elements is used to describe the geometry of a component. The BEM uses a similar approach to modeling; except in the BEM, only a surface model needs to be constructed.

Comparison Of BEM And FEM

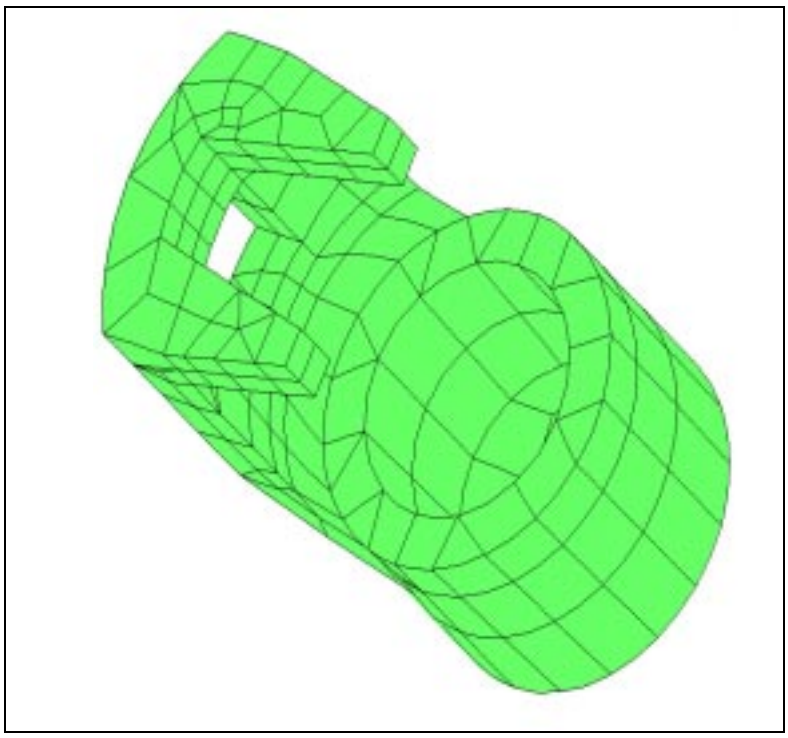
To illustrate the basic modeling difference between a BEM model and an FEM model, consider the basic problem of an elastic strip with an interior hole in tension. Since symmetry exists, a 2-D quarter model of the strip can be used for both the BEM and FEM models. The following figure illustrates what they might look like:



In particular, notice that the BEM model only requires nodes and elements around the surface of the quarter model of the strip. The FEM model requires a mesh over the entire region, which requires significantly more time and effort.

This modeling advantage of the BEM over FEM is magnified when we consider 3-D models. Consider the following model of an automotive brake caliper:

Model of an Automotive Brake Caliper



***TIP**:* Many books and articles have been published comparing the BEM and FEM. You can consult a basic BEM text, such as *The Boundary Element Methods in Engineering*, listed in Appendix A.

This mesh appears identical for the BEM and FEM models of this component. However, inside the surface they are completely different: the boundary element model is absent in the interior, while the finite element model requires a complete interior mesh. For real-world parts like the brake caliper shown above, creating a high-quality finite element mesh would be expensive and time-consuming. However, since the BEM model only requires a surface mesh, the modeling time is significantly reduced.

This modeling advantage is the primary advantage of the boundary element method. However, there are other advantages, such as superior accuracy of solution, infinite and semi-infinite models, crack and fracture problems involving stress concentrations, etc.

This is not to say that the BEM is a complete replacement for the FEM. It is useless to argue about which method is better. Each method has its own strengths. A knowledgeable user will be able to utilize both methods, and determine which one is the right tool for the job at hand.

Basic Theory Of The Boundary Element Method

The mathematics underlying the BEM theory and computer implementation is considered to be an order of magnitude more complex than the FEM. However, this need not be intimidating. This section will introduce the theory of the boundary element method without getting too involved in the finer points. The emphasis will be concept, not detail.

Consider a very large block of steel. Inside the block is a very large point force acting at a point. Around the point force, the deformation of the steel is very extensive. However, the deformation is negligible further from the point force.

***TIP**:* For more information on singularity functions and fundamental solutions, consult a basic BEM text, such as *The Boundary Element Methods in Engineering*, listed in Appendix A.

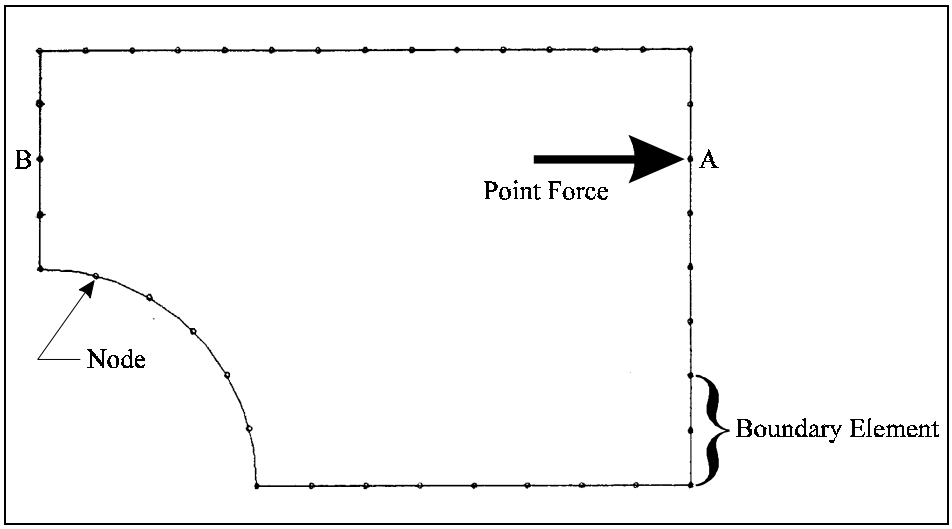
This basic behavior is captured mathematically in what are known as *singularity functions* and *singular solutions*, also known as *fundamental solutions*. The relevant singularity solutions for elasticity problems were first presented by Kelvin. Imagine an *infinite* steel block with a unit point force at a given location. The Kelvin solution, or elastic singular solution is the solution to the elasticity governing equations for this problem. It expresses the following basic ideas:

1) More deformation (displacement, stress, strain, etc.) occurs closer to the point force. Deformation is less when far away from the point force.

2) The direction of the point force affects the resulting deformation everywhere in the solid.

Once a fundamental solution is known for a particular problem (i.e., elasticity), a boundary element equation (also known as an *integral equation*) can be constructed. For instance, consider the following figure of the boundary element model for a simple problem:

Boundary Element Model For Quarter-Symmetric Model Of A Plate With A Hole



Imagine the following scenario: A unit point force is applied at point A. Monitoring equipment is set up at point B, and the displacement at B (due to the force at A) is measured. Now suppose, in addition to monitoring displacements at node B, displacements are measured at all the other nodes. Significant information would be accumulated. Now, suppose the point force is moved to another node, and the process is repeated for all nodes. When this process is finished, much data would be accumulated, but a complete solution to the problem would still be lacking.

***TIP**:* GPBEST automatically sets up the boundary integrals at each node and solves the system of equations. The user merely needs to provide a BEM model to GPBEST. GPBEST handles the complex mathematics.

Actually, this extensive amount of data is already available in the elastic fundamental solution. The problem is that the fundamental solution applies to an infinite solid, but not to the specific model presented here. This is where the boundary element method is helpful. It combines the extensive information available in the fundamental solution with a specific BEM model for the problem under consideration. The result of this procedure is a solution of a number of simultaneous equations yielding a solution (displacements, stresses, etc.) at every node on the boundary.

What if you really want to know what the deformation is *inside* the model? No problem! The BEM allows you to monitor the

deformation anywhere inside and outside of the model, even though only the boundary was modeled.

The detailed mathematics behind the boundary element method will not be given here. However, this section has introduced the concepts underlying the BEM. Consult a basic BEM text, such as those listed in Appendix A, for more details.

Components Of The GPBEST System

The GPBEST analysis system consists of various programs and manuals. As a new user, you will benefit greatly if you locate each of these items and become familiar with them.

Computer Programs

The GPBEST tape used to install the GPBEST system includes several computer programs. These will become available to you after GPBEST is installed:

> ***TIP***: GPBEST installation instructions are given in Chapter 2 of this manual.

1. The GPBEST program - This is the analysis program that takes a boundary element model as input and outputs results into a readable (ASCII) file and a (binary) GPBEST neutral file.
2. The BESTVIEW program - This is a graphical program that can be used to display, construct, and view results for GPBEST models.

> ***TIP***: The GPBEST example problems are extremely valuable for new users. They are discussed in Chapter 3 of this manual.

1. Example problems - The GPBEST system comes with hundreds of example problems that can be stored on your computer system. This enables users to quickly find example problems that may be similar to a problem they are trying to run. The example problems often allow users to get “up and running” very quickly.

> ***TIP***: The PATRAN, I-DEAS, HYPERMESH and FEMAP translators are demonstrated in the tutorials in Chapters 4-6 of this manual.

1. The Translators - There are a series of translators available to convert from other CAD systems into GPBEST and subsequently to be able to view graphical results in other programs. For instance, a PATRAN user can use PATRAN to build a boundary element model. Then a translator can be used to translate this file into GPBEST format. After running the GPBEST analysis, the results can be translated back into PATRAN format for viewing in PATRAN. Since most CAD systems support IGES, this can also be used to transfer CAD geometry into the GPBEST system.

Documentation

The GPBEST system includes several documents.

1. *Introduction to GPBEST: Elastic Analysis*
   This is the manual you are reading. It is the best place to begin learning the basics of the GPBEST system. It contains several tutorials which introduce many different aspects of the system in a minimal amount of time.
2. *GPBEST User's Manual (Volume I)*
   This is the main reference manual for GPBEST analysis. Primarily, Chapter 5 contains the "cards" used to construct the GPBEST input file. However, other chapters provide important supporting documentation for running the GPBEST program. In particular, Chapter 6 contains some basic example problems which users can study.
3. *BESTVIEW User's Manual (Volume II)*
   This contains the reference manual for the BESTVIEW program. Consult Chapter 1 for a quick-start section. Chapter 8 has some tutorial examples which lead the user through the entire process of building a GPBEST model in BESTVIEW and viewing results from a GPBEST analysis in BESTVIEW. Also included are two chapters which may be of interest to all GPBEST users, whether they use BESTVIEW or not. Chapter 9 discusses the Translators for converting data and results from other CAD/FEM programs into and out of GPBEST. Chapter 10 describes the GPBEST Neutral File, which can be used to extract boundary element model information and results. Chapter 10 permits users to write their own translator programs to interface their CAD/FEM program with GPBEST.

Basic Terminology

***TIP**:* More detailed terminology and definitions are presented in Section 4.5.1 of the GPBEST User's Manual (Volume I).

It may be helpful to some common terms used in the GPBEST program and manuals.

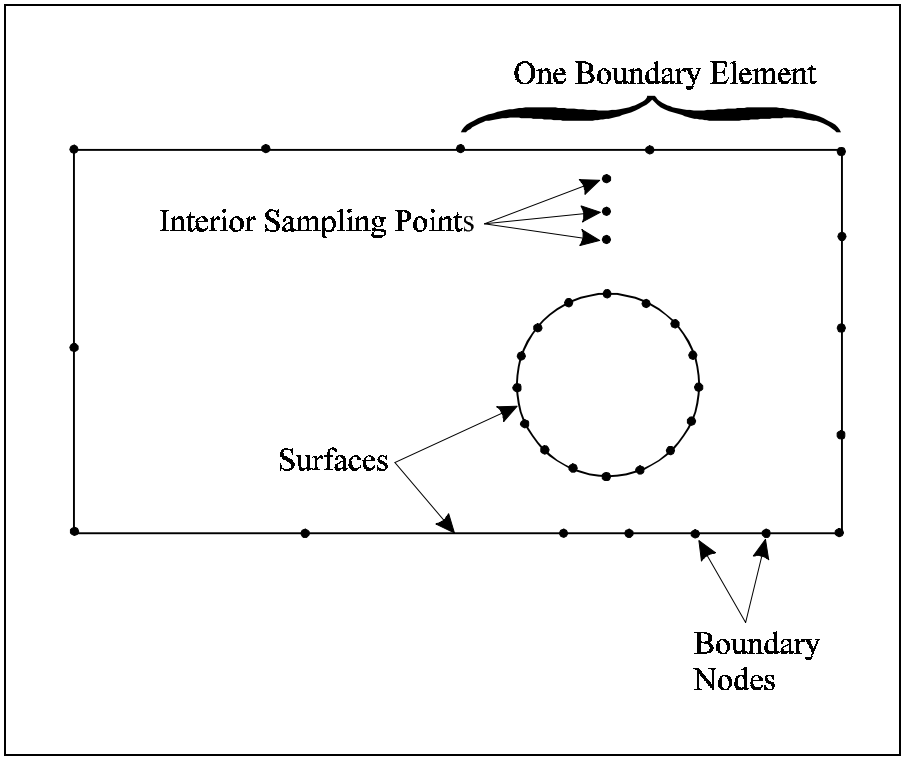
In the following picture, a 2-D boundary element model is used to illustrate some basic terminology. This is a model of a plate with an interior hole. The outer edge of the plate is called a *surface*, and is comprised of *boundary nodes* (or *boundary points*) and *boundary elements*. In this case, 3-noded (quadratic) boundary elements are used.

Also note that the interior hole is another surface. The two surfaces are independent of each other, hence their mesh refinement do not depend on each other. However, taken together, the two surfaces constitute a GMR (Geometric Modeling

Region), which results in an analysis of the stress, strain, displacements, etc. of the material *between* the hole surface and the outer edge surface of the plate.

Notice that there is no interior mesh. Only the boundaries of the component need to be modeled. However, if results are desired in the interior, *sampling points* can be used.

2-D Single-GMR BEM Model Of A Plate With An Interior Hole

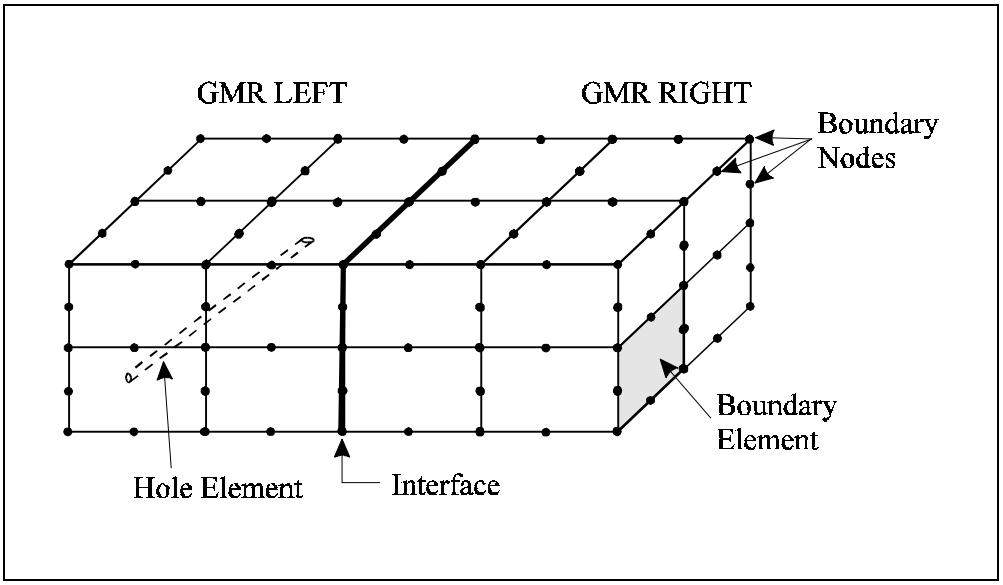


***TIP**:* GMRs are covered in more detail in Chapter 5 of this manual. The information presented here is merely intended to introduce some of the terminology used in GPBEST.

The 3-D model, shown below, illustrates how two GMRs can be used to break up a complex model into segments, joined at an *interface*. *Nodes* (or *points*) are used to define the outer surface of the block. In this case, 8-noded (quadratic) *boundary elements* (or *surface elements*) are used. The interior volume is not meshed, although *sampling points* could be used to capture results in the interior.

This figure also illustrates a powerful feature called *hole elements*. These permit quick modeling of cooling lines in injection molds and casting molds.

3-D Multi-GMR BEM Model



Other terminology will be introduced when appropriate. The terms described above should be sufficient to get started referring to the GPBEST manuals.

What To Do If You Are Having Problems

If you are experiencing problems using GPBEST or other programs in the GPBEST system, there are several steps that may help you:

Read as many diagnostic messages as possible. These could appear while the job is running, or they could be placed in the output files of various programs. Analyze these messages since they may lead you to the solution. For instance, the following message in the GPBEST results file indicates that a node has not been properly defined.

GPBEST Error Message Number And Description

```
-------------------------------------------------------
          ****            GMR INPUT            ****
-------------------------------------------------------

 *** FATAL ERROR  3044
    GMR GMR1
    NODE    994 IN ELEMENT      4 NOT FOUND
```

These message numbers will be very useful if you request technical support on a problem.

- Check the manuals. Although the GPBEST manuals are extensive, after using GPBEST for a while, you will find yourself referring to certain sections frequently. The more you do this, the better you will be able to diagnose problems when they arise.
- Contact the Boundary Element Software Technology Corporation for User assistance. Our phone numbers appear on the back of each manual.

  Phone: (716) 639-1920

  Fax: (716) 639-1919

  E-Mail: (support@gpbest.com)

CHAPTER 2 - INSTALLATION

Introduction

This chapter will lead you through the process of installing GPBEST from a distribution tape. In addition, the file structure of the GPBEST system will be described, including the extensive example problem directories.

System Requirements

There are certain computer system requirements that must be met in order to install and execute GPBEST. It would be futile to try to install GPBEST if the basic requirements are not met.

First, only the following engineering workstations and operating systems are supported:

- Digital Equipment (OSF1 3.2 or above)
- Hewlett-Packard (HPUX 9.XX)
- Hewlett-Packard (HPUX 10.XX or above)
- IBM RS6000 (AIX 3.2X or above)
- SUN Microsystems (SOLARIS 5.3 or above)
- Silicon Graphics (IRIX 5.XX or above)

A tape drive must be available to load the GPBEST system off of a tape. However, the workstation used to load up GPBEST does not need to be the same one that GPBEST is executed on at a later time. The above list of supported workstations only applies to the workstation that actually executes GPBEST. For example, if you had an XYZ brand computer, this could be used to install GPBEST off the installation tape, but a GPBEST-supported computer would be used to run problems.

In addition, it will take approximately 50 megabytes (Mb) of disk space just to load GPBEST. Additional space will be needed to uncompress many of the example files. Some of these files may be deleted or moved to a different disk afterward if you determine that they are not needed. However, to insure enough space for a complete installation on one disk, you should have at least 100 Mb of disk space available on the disk where GPBEST will be installed.

To *execute* GPBEST, it is important to have plenty of free disk space. The amount required will vary from problem to problem. However, it is probably a good idea to have 600 Mb of free disk

space (scratch space) available to run large, three-dimensional GPBEST problems. This disk space can reside on disks other than where GPBEST is installed. In addition, users will need several Mb of disk space to store recent data and results files before they are archived to tape.

It is also important to have enough main memory to run GPBEST. Although problems can be run with as little as 32 Mb of RAM, 64 Mb is the suggested amount of RAM for efficient execution. This will vary with problem complexity.

The BESTVIEW program is a graphical pre- and post-processor for GPBEST. It runs under the X-Windows graphics environment. Therefore, X-Windows is required to run BESTVIEW. However, some workstations have proprietary installations of X-Windows. For instance, SUN has '`openwin`'. BESTVIEW will run under this version of X-Windows.

The GPBEST user does not need to have specialized training or prior knowledge of the Boundary Element Method (BEM) in order to use GPBEST. However, knowledge of basic UNIX commands, the Finite Element Method (FEM), and computer graphics systems are helpful.

Note: 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 maximum 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 be 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.

Installation of GPBEST From Tape

Assuming that your system meets these basic requirements, you can proceed with the installation of GPBEST. Often, programs are installed by a system administrator. Whether the user or an outside person performs this step is immaterial. We will just refer to this person as the “system administrator” for the rest of this chapter.

Step 1: Determine Device Name Of Tape Drive

Before starting the installation, you must know the tape device name of the tape drive. Typically, on systems with only one tape drive, the following names are used:

Workstation	Tape Device Name
SUN systems	`/dev/rst0`
IBM RS6000 systems	`/dev/rmt0`
Silicon Graphics	`/dev/tape`
Hewlett Packard 9000/700 series	`/dev/rmt/0mn`
Digital DEC ALPHA	`/dev/rmt0h`

Step 2: Create A GPBEST Installation Directory And Install

> ***TIP:*** In this manual, the name of the GPBEST installation directory is called `/usr/local/gpbest`. This may differ from your system.

When installing programs, the system administrator typically will log in as a superuser, which gives permission to create directories at the root level.

GPBEST can be installed anywhere on the system. It could even be installed in a specific user's area, although this would be the exception. GPBEST is usually installed in a general area that can be accessed by all users. As the system administrator, you must decide where to install GPBEST. However, for the sake of this tutorial, we will assume that GPBEST is installed in the `/usr/local` directory. Thus, the GPBEST installation directory will be called `/usr/local/gpbest`. This name will appear frequently in the rest of this manual, so it is important to understand exactly where it is. The process of creating the GPBEST installation directory and loading the files from tape are illustrated here. The UNIX `tar` command will require the tape device name determined in **Step 1**. The following illustrates how installation may proceed on an HP workstation:

> ***TIP:*** In this manual, the dollar sign, $, will be used to indicate the UNIX prompt. User input will be displayed in **`bold`** type, while standard output from UNIX `will not be in bold type`.

```
login: james (superuser)
password: xyzabc
$ cd /usr/local
$ mkdir gpbest
$ chmod ugo+rx gpbest
$ cd gpbest
$ tar xvpf /dev/rst0
x ./README, 5140 bytes, 11 tape blocks
x ./examples/ex.large/CONTinjec.dat.Z, 44117 bytes, 87 tape blocks
x ./examples/ex.large/ACOUwall.dat.Z, 26878 bytes, 53 tape blocks
x ./examples/ex.large/sphere.dat.Z, 3533 bytes, 7 tape blocks
x ./examples/ex.large/ELASbrac.dat.Z, 172061 bytes, 337 tape blocks
x ./examples/ex.large/ELAScase.dat.Z, 171214 bytes, 335 tape blocks

 . . .

x ./translators/gpb/prob7.dat.Z, 518 bytes, 2 tape blocks
x ./translators/gpb/prob3.res.Z, 6128 bytes, 12 tape blocks
x ./translators/gpb/prob6.res.Z, 6750 bytes, 14 tape blocks
x ./translators/gpb/prob1.dat.Z, 4903 bytes, 10 tape blocks
$
```

In actuality, hundreds of files are installed in the above procedure. This will probably take less than 20 minutes.

Step 3: Remove Unnecessary Files

Although the entire GPBEST installation is copied from the tape to the hard disk, not all files may be necessary. Some files may be deleted if disk space is at a premium. In order to delete unwanted files, it is necessary to briefly describe the GPBEST installation directory structure.

In **Step 2**, the GPBEST installation copied files into the base GPBEST installation directory `/usr/local/gpbest`. (You may have chosen to use a different directory). Under the GPBEST installation directory there are several subdirectories. The following table describes the various GPBEST subdirectories, along with the workstation-specific subdirectories.

Subdirectory	Workstation (Operating System)
`./bin/ALPHA`	`DEC ALPHA (OSF1 3.2 or above)`
`./bin/HPPA9`	`Hewlett-Packard (HPUX 9.XX)`
`./bin/HPPA10`	`Hewlett-Packard (HPUX 10.XX or above)`
`./bin/RS6K`	`IBM RS6000 (AIX 3.2X or above)`
`./bin/SGI5`	`Silicon Graphics (IRIX 5.XX or above)`
`./bin/SUN4SOL2`	`SUN Microsystems (SOLARIS 5.3 or above)`
`./examples`	
`./translators`	

The system administrator should keep the subdirectory corresponding to their specific workstation and delete the rest to conserve disk space. For example, if GPBEST is run on an HP workstation (using HPUX 10.0), then the following session illustrates how the other unnecessary subdirectories can be deleted:

```
$ cd /usr/local/gpbest/bin
$ rm -r ALPHA
$ rm -r HPPA9
$ rm -r RS6K
$ rm -r SGI5
$ rm -r SUN4SOL2
$
```

This will leave the executables for the HP workstation version of GPBEST in the directory `/usr/local/gpbest/bin/HPPA10`. For the rest of this chapter, we will assume that the user is installing

TIP: A more detailed listing of the GPBEST distribution directories appears later in this chapter.

GPBEST on an HP workstation. Modify the remaining instructions for your particular workstation.

Step 4: Uncompress The Compressed Files

Many of the files in the GPBEST installation are stored in UNIX `compress` format. They are denoted by file names ending with `.Z`. Thus, `file1.dat` could be an uncompressed file, while the same file in compressed format would be `file1.dat.Z`. Compression is performed to save space on the installation tape and on the installation disk.

> ***TIP:*** It is most convenient for the users to `uncompress` all the files, since they will be readily available.

The system administrator may not need to `uncompress` all files. If disk space is limited, the system administrator may want to study the section "GPBEST File Structure" which appears later in this chapter. This may indicate which files may not be needed by the user, so they can be deleted or compressed. For the time being, we will assume that there is plenty of disk space and it is desirable to `uncompress` all files. The UNIX `find` command can be used to `uncompress` all the files at once. The `-exec` option executes the `uncompress` command on all compressed files:

```
$ cd /usr/local/gpbest
$ find . -name '*.Z' -exec uncompress {} \;
$
```

This may take about 20 minutes, and will increase the disk space taken up by the entire GPBEST installation three to four times. If you receive a system error complaining "`disk space full`", then you cannot `uncompress` all the files and keep them on this disk.

Step 5: Copy PATRAN Files (Optional)

> ***TIP:*** Users of PDA/PATRAN 2.5 do not need to carry out this step.

If users are using MSC/PATRAN to post-process GPBEST results, the system administrator must copy some files into the PATRAN installation area. Suppose that PATRAN was previously installed to the `/msc/patran75` directory. To copy the needed GPBEST files, execute the following command:

```
$ cp /usr/local/gpbest/templates/*tmpl /msc/patran75/res_templates/.
$
```

The actual directory names may be somewhat different for your system.

Step 6: GPBEST Password

A password file is needed in order to execute GPBEST and BESTVIEW. The password file contains a password for your specific installation and computer system ID. Instructions for obtaining the password file are contained in Appendix D.

> NOTE: A single password file can be used for GPBEST and BESTVIEW operating on a multiple-computer license. Let the Boundary Element Software Technology Corporation know your complete configuration when requesting a password. Of course, a license must be purchased for each component.

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 looks somewhat like the example shown in Appendix D.

```
$ vi password
  (strip off e-mail headings, etc.)
$
```

> ***TIP:*** Use only method 1) *or* 2). Do not use both.

Assuming the necessary password file has been obtained and edited, the system administrator can place it in one of two places:

1) In the directory `/gpbest`, which resides on the same disk as the GPBEST installation directory.

```
$ cd /
$ mkdir /gpbest
$ cp /etc/mail/password /gpbest/password
$ chmod ugo+r /gpbest/password
$
```

> ***TIP:*** The `password` file in these examples is assumed to reside in `/etc/mail`. Substitute your actual full file name (path + file) for `/etc/mail/password`.

2) In a different directory. Users must have this set as a variable in their startup file (see **Step 7**). If this approach is used, the password file can be renamed to be any name, as long as this name is properly specified in the startup file. For example, the system administrator placed the password file in the GPBEST *installation* directory, and its file name is `password`:

```
$ cd /usr/local/gpbest
$ cp /etc/mail/password .
$ chmod ugo+r password
$
```

Step 7: Modify User Startup Files

Depending on the specific UNIX shell being used, certain startup files must be modified in order to run GPBEST and BESTVIEW. The changes are:

1) The GPBEST `bin/.` directory, where the GPBEST and BESTVIEW executables are located, must be added to the user's existing path.
2) If the system administrator placed the GPBEST password file in a directory other than `/gpbest` in **Step 6**, then this directory and file name must be defined as an environment variable.
3) If the user will be using BESTVIEW for pre- and post-processing, the path to the BESTVIEW executable must be defined as a separate environment variable.

These three steps will now be enumerated for each UNIX shell. In each of the examples given below, we will assume that the GPBEST installation directory is `/usr/local/gpbest`, and the user is working on an HP workstation. Modify accordingly for your local installation.

User's UNIX Shell		Changes to Startup Files
c-shell (add to `$HOME/.cshrc` file)	1)	set path=(. /bin /usr/bin . . . /usr/local/gpbest/bin/HPPA10) *(The ". . ." represents other directories in the user's current path)*
	2)	setenv GPBEST_PASSWORD_FILE /usr/local/gpbest/password
	3)	setenv BVIEWDIR /usr/local/gpbest/bin/HPPA10
borne shell (add to `$HOME/.profile`)	1)	PATH= .:/bin:/usr/bin: . . . :/usr/local/gpbest/bin/HPPA10 export PATH
	2)	GPBEST_PASSWORD_FILE=/usr/local/gpbest/password export GPBEST_PASSWORD_FILE
	3)	BVIEWDIR=/usr/local/gpbest/bin/HPPA10 export BVIEWDIR
korn shell (add to `$HOME/.kshrc`)	1)	PATH= .:/bin:/usr/bin: . . . :/usr/local/gpbest/bin/HPPA10 export PATH
	2)	GPBEST_PASSWORD_FILE=/usr/local/gpbest/password export GPBEST_PASSWORD_FILE
	3)	BVIEWDIR=/usr/local/gpbest/bin/HPPA10 export BVIEWDIR

Alternately, instead of defining the GPBEST `bin/.` directory inside the path declaration, it can be *appended*. For example, if a user is using the c-shell on an HP workstation, inserting the following line *after the path has been defined* in `.cshrc` will append the GPBEST executable directory to the current path:

```
set path=($path /usr/local/gpbest/bin/HPPA10)
```

Similarly, if the user is using the korn shell or the borne shell, then insert the following lines after the path has been declared in the `.kshrc` file or `.profile` file, respectively:

```
PATH=$PATH:/usr/local/gpbest/bin/HPPA10
export PATH
```

Step 8: Test The GPBEST Installation

TIP: We encourage the system administrator to carry out Steps 8 and 9 to insure that GPBEST (and optionally BESTVIEW) have been properly installed.

If the installation is successful, the user should be able to run GPBEST, BESTVIEW, etc. To test the installation, first log out of the system, then log back in to make sure that the changes to the startup files take effect. Then run the GPBEST command (`best-s`), as shown below. Note that since you are not specifying any input file, the program will wait indefinitely. Hit `<Ctrl-C>` to break out of the infinite loop. (At this point, we are not interested in running a specific problem. We are just testing whether the system can locate the GPBEST executable and password file.)

A Successful GPBEST Test Run

```
$ cd $HOME
$ mkdir junkdir
$ cd junkdir
$ best-s

      ================================================================

                                G P B E S T

              GENERAL PURPOSE BOUNDARY ELEMENT SOLUTION TECHNIQUE

                                VERSION 5.0
                                 APRIL 1996


              PROPERTY OF THE B.E.S.T. CORPORATION
              TO BE USED WITH PERMISSION ONLY
              ALL RIGHTS RESERVED

      ================================================================

      GPBEST LICENSE INFORMATION:

  GPBEST IS LICENSED TO: Boundary Element Software Technology
Corporation
  LICENSE STARTING DATE:  August 1, 1995
  LICENSE ENDING DATE  :  December 31, 1999
  COMPUTER HOSTNAME/TYPE : jupiter Hewlett-Packard

      LICENSE AGREEMENT PERMITS EXECUTION OF THE FOLLOWING ANALYSES:
        ACOUSTIC           : X             FORCED VIBRATION   : X
        CONSOLIDATION      : X             FREE VIBRATION     : X
        CONCURRENT THERMAL : X             INHOMOGENEOUS      : X
        SCALAR POTENTIAL   : X             HEAT TRANSFER      : X
        ELASTIC            : X             PLASTICITY         : X
        FLUID DYNAMIC      : X             TRANSIENT DYNAMIC  : X
        DYN POROELASTIC    : X             DYN THERMOELASTIC  : X
        THERMAL DISTORTION : X

      LICENSE AGREEMENT PERMITS:
        TOTAL NUMBER OF REGIONS:           40
        TOTAL DEGREES OF FREEDOM:       24000
        -SEE YOUR GPBEST MANUAL FOR FURTHER DETAILS


(Hit Ctrl-C here to halt the program...)


*** TERMINATING best-s
*** Received signal 2 (SIGINT)

$
```

If a response similar to this has been received after executing the `best-s` command, then GPBEST has been successfully installed.

If the GPBEST session looks similar to the following screen, then GPBEST is not able to locate the correct password file:

GPBEST Could Not Find The Password File

```
$ cd $HOME
$ mkdir junkdir
$ cd junkdir
$ best-s

      ================================================================

                                G P B E S T

             GENERAL PURPOSE BOUNDARY ELEMENT SOLUTION TECHNIQUE

                                VERSION 5.0
                                 APRIL 1996


             PROPERTY OF THE B.E.S.T. CORPORATION
             TO BE USED WITH PERMISSION ONLY
             ALL RIGHTS RESERVED

      ================================================================

      GPBEST LICENSE INFORMATION:

cchar # (35 decimal) not found in function getval
cchar # (35 decimal) not found in function getval
cchar . (46 decimal) not found in function getval
cchar c (99 decimal) not found in function getval

 . . .

83 ctchar not found in function conval
FATAL CHECKSUM ERROR - LINE 1 - PASSWORD IS CORRUPT
$
```

What To Do If GPBEST Cannot Find The `password` File

Here are some things that can be checked if this problem occurs:

1. Check to make sure you are executing `best-s` on the correct workstation.
2. Make sure that the password file has been installed properly (**Step 6**). Does it exist? Where is it located? Are the file permissions correct?
3. Make sure the password directory is specified in the environment variable (**Steps 6 and 7**). Also, the path should refer to the path for the specific workstation you are using. For example, the path should not refer to the IBM directory (RS6K) if you are running on an HP workstation.
4. Make sure that the password file is up-to-date. Don't use an old password file. Has the license expired, and thus need updating? Contact the Boundary Element Software Technology Corporation to renew the license.
5. Make sure that the password file corresponds to the specific workstation you are using. The password contains encrypted information about the computer's system ID.

If GPBEST will not run, call the Boundary Element Software Technology Corporation, whose phone number, fax number and e-mail addresses are listed at the beginning of this manual.

Step 9: Testing The BESTVIEW Installation

TIP: BESTVIEW is licensed separate from GPBEST. You must be licensed to run BESTVIEW and have a proper `password` file to carry out **Step 9**.

At this point, the BESTVIEW installation can be tested. To test the BESTVIEW installation, you must be in the X-Windows environment. Execute the following:

```
$ cd $HOME
$ mkdir junkdir
$ cd junkdir
$ bestview
$
```

If everything works fine, you will see the following screen:

The Opening BESTVIEW Screen

Select the “Cancel” button with the mouse to exit this screen.

A problem may arise in the fonts. If the following message appears when executing BESTVIEW, then the font names must be modified:

```
 Graphics: Display :0.0 doesn't know font
 -misc-fixed-medium-r-normal--15-140-75-75-c-90-iso9999-1

 Run the Font Picker!
$
```

This message indicates that BESTVIEW started up trying to locate some fonts that were not on the system. Each installation of X-Windows has its own set of fonts. The two fonts needed by BESTVIEW are listed in the `BestviewFonts.inc` file located in the GPBEST installation directory. In this chapter (for a user on an HP workstation), the file is `/usr/local/gpbest/bin/HPPA10/BestviewFonts.inc` . This file consists of four lines, in which the actual fonts used by BESTVIEW are listed in lines 1 and 3. The following is an example of a `BestviewFonts.inc` file:

```
-misc-fixed-medium-r-normal--15-140-75-75-c-90-iso8859-1
font1
-adobe-courier-medium-r-normal--24-240-75-75-m-150-iso8859-1
font2
```

To list fonts that are supported on your system, use the `xlsfonts` utility:

```
$ xlsfonts
-adobe-courier-bold-o-normal--10-100-75-75-m-60-iso8859-1
-adobe-courier-bold-o-normal--12-120-75-75-m-70-iso8859-1
-adobe-courier-bold-o-normal--14-140-75-75-m-90-iso8859-1
-adobe-courier-bold-o-normal--18-180-75-75-m-110-iso8859-1
-adobe-courier-bold-o-normal--24-240-75-75-m-150-iso8859-1
-adobe-courier-bold-o-normal--8-80-75-75-m-50-iso8859-1
-adobe-courier-bold-r-normal--10-100-75-75-m-60-iso8859-1
-adobe-courier-bold-r-normal--12-120-75-75-m-70-iso8859-1
-adobe-courier-bold-r-normal--14-140-75-75-m-90-iso8859-1
 . . .
$
```

Choose two of these fonts and replace lines 1 and 3 of the `BestviewFonts.inc` file. Rerun BESTVIEW to make sure that the new fonts function properly. You may have to experiment with various fonts until two fonts are found that display properly in BESTVIEW.

TIP: Refer to Section 6.5 of the BESTVIEW User's Manual (Volume II) for more information on the `COLORBAR` file.

A `COLORBAR` file is included with Bestview. It is located in the GPBEST directory as `/usr/local/gpbest/bin/HPPA10/COLORBAR`. The `COLORBAR` file lists the colors used by BESTVIEW to display color fringe plots. If the standard colors are satisfactory, no further steps need to be taken. However, in order to define your own color spectrum for fringe plots, copy the `COLORBAR` file into your current working directory and edit the colors.

```
$ cp /usr/local/gpbest/bin/HPPA10/COLORBAR  $HOME/work/.
$
```

GPBEST File Structure

In **Step 2** of the GPBEST Installation procedure, the GPBEST distribution was placed under the directory `/usr/local/gpbest` . Since there are hundreds of files included in the distribution, it is helpful to describe some of the more useful files and subdirectories.

In order to obtain a list of all directories and files in the GPBEST distribution, use the UNIX `find` command:

```
$ find /usr/local/gpbest -print

/usr/local/gpbest
/usr/local/gpbest/README
/usr/local/gpbest/examples
/usr/local/gpbest/examples/ex.large
/usr/local/gpbest/examples/ex.large/CONTinjec.dat.Z
/usr/local/gpbest/examples/ex.large/ACOUwall.dat.Z

 . . .

/usr/local/gpbest/translators/ex.pre.pat3/pat05.dat.Z
/usr/local/gpbest/translators/ex.pre.pat3/pat05.log.Z
/usr/local/gpbest/translators/ex.pre.pat3/pat05.prt.Z
/usr/local/gpbest/translators/ex.pre.pat3/pat05.ses.Z
/usr/local/gpbest/translators/ex.pre.pat3/pat01.PATRAN.OUT.Z
/usr/local/gpbest/translators/ex.pre.pat3/pat03.PATRAN.OUT.Z
$
```

To help the system administrator and user become more familiar with the GPBEST distribution, a listing of the *primary* subdirectories and files are given here. In this chapter, it is assumed that the GPBEST installation directory is `/usr/local/gpbest` . The following would then be subdirectories of this installation directory:

Subdirectories and Files	**Description**

`./bin/ALPHA/`
`./bin/HPPA9/`
`./bin/HPPA10/`
`./bin/RS6K/`
`./bin/SGI5/`
`./bin/SUN4SOL2/`
	These directories contain workstation-specific files. It is recommended that the system administrator delete the subdirectories corresponding to unnecessary machines in order to save disk space. (See **Step 3**).

`./bin/*/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.
`./bin/*/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 the working directory and select it from within BESTVIEW (see the BESTVIEW manual for details).
`./bin/*/bestview`	BESTVIEW executable. BESTVIEW (Boundary Element Software Technology Viewer) is a graphic pre- and post-processor for generation and viewing of boundary element models as well as for preparation of input data for subsequent analysis through GPBEST.
`./bin/*/best-s`	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.).
`./bin/*/best-s60`	GPBEST (standard) executable (60,000 d.o.f.). For smaller problems use `best-s` for best efficiency.

Subdirectories and Files	Description
`./bin/*/best-t`	GPBEST executable for ALL analysis types. Standard analyses contained in `best-s` are also contained in `best-t`. However, it is recommended that `best-s` be used for standard analysis since `best-s` requires less computer resources to execute. `best-t` should be used for all other analyses not contained in `best-s`. These are periodic acoustics, structural acoustics, forced vibration and plasticity analysis (24,000 d.o.f.).
`./bin/*/iges`	IGES executable. IGES executable module is used by the BESTVIEW software to import IGES files.
`./bin/*/prebest`	GPBEST translator. Translates PATRAN neutral file, I-DEAS universal file, HYPEREMSH neutral file and FEMAP neutral file to GPBEST input data files.
`./bin/*/postbest`	GPBEST translator. Translates GPBEST neutral file NEUTRAL.OUT (created during GPBEST execution for post-processing GPBEST results).
`./bin/*/README`	Miscellaneous information about this directory. View this file for the most recent information.
`./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`**
`./examples/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`).

Subdirectories and Files	Description
`./examples/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** is 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:
`./examples/ex.small/*.`**`dat`**	The `.dat` extension signifies the GPBEST data file. Executing GPBEST with this input file creates the neutral file and result file.
`./examples/ex.small/*.`**`neu`**	GPBEST neutral file. Originally it is created by GPBEST as `NEUTRAL.OUT`. However, it is renamed with the `.neu` extension to specify the specific example it belongs to.
`./examples/ex.small/*.`**`res`**	GPBEST results file. Contains warning and error messages, information about GPBEST execution and results (displacements, stresses, temperatures, pressures, etc.).
`./examples/ex.small/`**`acou`**`*`	Acoustic analysis.
`./examples/ex.small/`**`aeig`**`*`	Acoustic eigenfrequency analysis.
`./examples/ex.small/`**`anis`**`*`	Elastic anisotropy (includes thermal and mechanical body forces).
`./examples/ex.small/`**`cons`**`*`	Consolidation.
`./examples/ex.small/`**`cpls`**`*`	Concurrent thermoplasticity (one-way coupling of heat conduction and thermoplasticity).
`./examples/ex.small/`**`cthe`**`*`	Concurrent thermoelasticity (one-way coupling of heat conduction and thermoelasticity).
`./examples/ex.small/`**`dist`**`*`	Thermal distortion with holes (heat from cooling holes causes distortion).
`./examples/ex.small/`**`elas`**`*`	Elasticity (includes thermal and mechanical body forces).
`./examples/ex.small/`**`forc`**`*`	Periodic elastodynamic analyses.

Subdirectories and Files	Description
`./examples/ex.small/`**`frac`**`*`	Fracture mechanics.
`./examples/ex.small/`**`frvb`**`*`	Free-vibration analysis.
`./examples/ex.small/`**`gaxis`**`*`	Generalized axisymmetric elastic analysis under nonaxisymmetric loading.
`./examples/ex.small/`**`heat`**`*`	Steady-state and transient heat conduction (includes problems with holes).
`./examples/ex.small/`**`plas`**`*`	Elastoplasticity.
`./examples/ex.small/`**`saco`**`*`	Structural acoustic analysis.
`./examples/ex.large/`	Contains a series of data for very large problems of elastic stress analysis involving complex geometries.
`./templates/*.`	Template files needed for MSC/PATRAN post-processing of GPBEST results.

With a knowledge of the purpose of each directory and file, the system administrator can intelligently determine if some files can be deleted or compressed. For instance, if the GPBEST users will only be running elasticity problems, then only the **`elas`** files may need to be kept in the `./examples/ex.small/` directory. For instance, if disk space was at a premium, the **`acou`**, **`heat`**, etc. files could be compressed or deleted from the system, while leaving the **`elas`** files on the system.

However, there is an advantage in keeping some of these files on the disk anyway. Sometimes a specific GPBEST capability may be better illustrated by an example problem from a different analysis type. For example, a particular cyclic symmetry option may not appear in any of the **`elas`** example problems. However, it could be extensively illustrated in the **`heat`** example problems. If there is sufficient disk space, it is recommended to keep all the example files on the system.

It is important to realize that the executables and files necessary to execute GPBEST, BESTVIEW, IGES, etc. are located in the

`./bin/*/` directory for the specific workstation. For instance, for an HP workstation, this directory is `/usr/local/gpbest/bin/HPPA10/`. All other directories and files could be moved to another disk if necessary.

CHAPTER 3 - TUTORIAL #1: RUNNING AN EXAMPLE PROBLEM

Introduction

This is the first tutorial. It involves the basic steps of finding an example problem in the GPBEST example directory and running GPBEST. It also illustrates basic concepts relevant to boundary element analysis, UNIX commands, input and output files, etc. After reading this chapter and following along by executing the given commands, the user should be able to search the example problems, submit GPBEST jobs and check the output.
This chapter assumes that the system administrator has installed GPBEST according to instructions given in Chapter 2. A simple GPBEST job was run in **Step 8** of Chapter 2 to test the installation. It was also mentioned in **Step 7** that each user must add certain statements to their UNIX startup files. Proceed with Chapter 3 only after the complete installation procedure has been carried out and GPBEST has been successfully invoked.

TIP: For some good books on UNIX, see Appendix A. These books contain many examples which enable the user to master UNIX commands quickly.

After logging into the workstation, you are probably at the standard UNIX prompt. In this manual, the `c-shell` is used to enter in UNIX commands. However, any UNIX shell will suffice. For example, if the user `john` logs in and runs the UNIX `date` command followed by the `echo` command, the following window might illustrate what this would look like:

```
login: john
password: xyzabc

$ date
Tue Mar 19 18:58:45EST 1996
$ echo $HOME
/users/john
$
```

`$HOME` is a standard UNIX variable that stands for the user's login directory. Hence, in this case, the user, `john`, could use `$HOME` and `/users/john` interchangeably. In the rest of this tutorial, `$HOME` will be used to signify the user's login directory.

Searching The Example Problems

To run an example problem, you must set up a working directory which can be used for the purpose of copying files and running problems. Any directory can be used. Typically you may wish to run problems in many different directories. For now, we will create a directory under the `$HOME` directory called `work`:

TIP: Common UNIX directory commands:
`mkdir` - create directory
`rmdir` - remove directory
`cd` - change directory

```
$ cd $HOME
$ mkdir work
$
```

The example problems were installed with GPBEST in Chapter 2. Now the user must find the GPBEST installation directory. In Chapter 2, it was named `/usr/local/gpbest`. Your particular installation may be different. For the purpose of illustration, the name `/usr/local/gpbest` will be used here.

TIP: If your GPBEST installation is not in `/usr/local/gpbest`, see your system administrator to find out where it was placed during the installation procedure.

Use `cd` to go into the `examples` directory and list the elasticity example data files in the directory with the `ls` command:

```
$ cd /usr/local/gpbest/examples/ex.small
$ ls elas*.dat
elas001.dat     elas068.dat     elas202.dat
elas002.dat     elas069.dat     elas203.dat
elas003.dat     elas070.dat     elas204.dat
elas004.dat     elas071.dat     elas205.dat
elas005.dat     elas072.dat     elas206.dat
 . . .
$
```

NOTE: If all the files have a `.Z` extension, then **Step 4** of the installation procedure (in Chapter 2) was not performed. Also, if there are no files in the examples directory, perhaps the system administrator deleted them or moved them to another directory.

TIP: For a description of each of the abbreviations used in the `examples` directory, refer to Chapter 2 of this manual or the GPBEST User's Manual (Volume I), Section 1.4. For example, `elas` stands for elasticity, `plas` stands for plasticity, etc.

The `examples` directory consists of hundreds of data files (`*.dat`), GPBEST neutral files (`*.neu`), and GPBEST results files (`*.res`). It is important to be able to search this directory efficiently to find files with specific capabilities. When encountering a new type of analysis or capability, it is usually most efficient to find an existing example problem with the desired capability. However, it would not be effective to view each file one by one until you find an example with the specific card or capability. One way to view the name and size of files is to obtain a 'long listing' (`ls -l`):

TIP: To list all the elasticity example files, (not just the 600 series):
`$ ls -l elas*`

```
$ ls -l elas6*.dat
-rw-r--r--  1 gpbest       5966 Nov  2  1993 elas603.dat
-rw-r--r--  1 gpbest       2682 Nov  2  1993 elas605.dat
-rw-r--r--  1 gpbest       3260 Nov  2  1993 elas606.dat
-rw-r--r--  1 gpbest       8004 Nov  2  1993 elas607.dat
-rw-r--r--  1 gpbest       3267 Nov  2  1993 elas608.dat
-rw-r--r--  1 gpbest      14877 Nov  2  1993 elas613.dat
 . . .
$
```

The first ten characters on each line indicate the file permissions. Suffice to say, if there are three r's (i.e., read permission) in the file permission section, this means that all users on the system can access them and copy them. If some of the files do not have read permission, then you may not be able to copy them. Consult your system administrator for assistance with this if there is a problem.

Another extremely important UNIX searching tool is the `grep` command. The importance of the `grep` command cannot be overstated. It allows the user to search through many files, looking for a string of characters. The basic syntax of the `grep` command is:

`grep` *`string`* `filenames`

TIP: For more information on the `grep` command, see the UNIX manual page:
`$ man grep`
or see an introductory UNIX book, such as *Morgan and McGilton* (Appendix A).

where the *`string`* is a single word or a series of characters inside single quotes. For example, to search the example directory for a file containing the phrase `SIMPLE CUBE` , use the following:

```
$ grep 'SIMPLE CUBE' *.dat
elas003.dat:$  SIMPLE CUBE IN TENSION IN Z-DIR
elas004.dat:$  SIMPLE CUBE IN TENSION,CONSTRAIN
elas605.dat:$  SIMPLE CUBE IN TENSION IN X-DIR
$
```

TIP: The `grep` command is *case-sensitive*. To search for `TOP` and `top`, use:
`grep -i top *`

This indicates that three example data files contain the phrase `SIMPLE CUBE` . The specific file name and line containing the phrase are shown.

Before leaving the subject of `grep`, the user should be aware of the power of the double-`grep` commands. These can be extremely useful for searching for specific features in GPBEST data files. Basically, there are two types of double-`grep` commands. First, suppose the user wants to find all the data files in the `examples` directory that contain the word `SIMPLE` and the

word CUBE, *occurring on the same line*. Instead of searching for the phrase SIMPLE CUBE, the two words can now be separated by other words.

```
$ grep 'SIMPLE' *.dat | grep 'CUBE'
elas003.dat:$   SIMPLE CUBE IN TENSION IN Z-DIR
elas004.dat:$   SIMPLE CUBE IN TENSION CONSTRAIN
elas605.dat:$   SIMPLE CUBE IN TENSION IN X-DIR
elas605.dat:  TITLE     A UNIT CUBE IN SIMPLE TENSION
$
```

Notice that this turned up the files from the previous grep command, but now included an additional line that was not caught by the phrase-grep command. In UNIX terminology, the output from the first grep command is *piped* (via the | character) to the second grep command.

To illustrate the second type of double-grep command, suppose the user wants to find all the data files in the examples directory that contain the word SIMPLE and the word SAMPLING, but *not necessarily occurring on the same line*.

TIP: The command inside the grave accents (`) is executed first by UNIX. Then its output is fed as input into the other grep command. Refer to a UNIX book to understand the grave accents better.

```
$ grep -l 'SIMPLE ' `grep -l 'SAMPLING' elas*.dat`
elas004.dat
elas011.dat
elas037.dat
elas038.dat
elas039.dat
elas040.dat
elas042.dat
elas605.dat
$
```

The backward quote (`) is called the grave accent. Make sure you don't use the other single quote ('). The character after the hyphen is the letter (l), not the number one (1).

TIP: Be careful! The grep command searches through all lines in a file, including comment lines ($).

In addition to using grep to search the example files, you can view the EX.summary file (/usr/local/gpbest/examples/EX.summary). It contains a brief listing of the GPBEST capabilities of the examples.

We will return to the grep command, when it will be used to search through the GPBEST results file. For the time being, suppose we have found the example file (elas605.dat) we want and we want to run it using GPBEST. It must be copied to your working directory (which was defined previously):

TIP: Common UNIX file commands:
`cp - copy file`
`mv - move or rename file`
`rm - remove file`

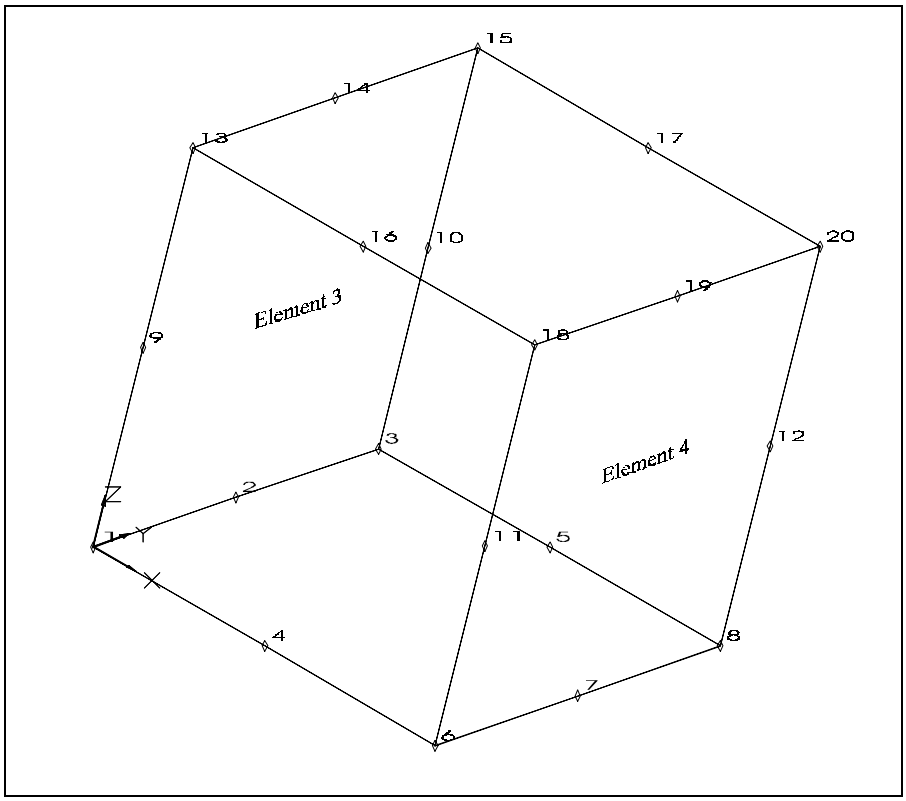
```
$ cp elas605.dat $HOME/work/.
$ cd $HOME/work
$ ls -l
total 2
-rw-r--r--  1 john         2682 Mar 19 19:06 elas605.dat
$
```

Looking At The GPBEST Data File

The model in `elas605.dat` is displayed in the following figure. The twenty node numbers are labeled. This particular problem contains six boundary elements (also known as surface elements), one for each face of the cube. There is no interior mesh; only a surface mesh. FEM users must adapt to the fact that an interior mesh using solid elements is not needed in the BEM model.

The boundary conditions are quite simple: element 3 is fixed, while element 4 has a uniform traction applied. Typically, in the BEM, boundary conditions and surface loads are applied on *surface elements*. This contrasts with the FEM, where most boundary conditions and loads are specified at the nodes.

Node And Element Numbers For Example Problem elas605.dat



TIP: Common UNIX commands to display a text file:
`cat - list out entire file`
`more - show one page at a time`
`vi - the standard UNIX text editor`
`emacs - a text editor`

The full text of file `elas605.dat` is included here. The user can probably identify at least the node and element numbers in the above figure from the lines in the data file. Note the basic sections indicated by **XXXX. To display a text file, issue either of the following commands:

```
cat elas605.dat
```

or

```
more elas605.dat
```

When using the `more` command, use the `<spacebar>` to display successive screens of information.

TIP: This problem is also listed in the GPBEST User Manual (Volume I) in Chapter 6. That chapter contains some selected problems in their entirety, including pictures of the model and results output. It would be wise to consult Chapter 6 when starting a new analysis type.

COMMENTS ($)

```
$ 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:
$
$
```

****CASE**

```
**CASE
TITLE     A UNIT CUBE IN SIMPLE TENSION
  ELASTICITY
  ECHO
```

****MATE**

```
**MATE
  ID MAT1
  TEMP  70.0
    EMOD 10.E6
    POIS 0.3
```

****GMR**

```
**GMR
  ID GMR1
  MAT MAT1
  TINT  70.0
  POINTS
   1       0.0      0.0      0.0
   2       0.0      0.5      0.0
   3       0.0      1.0      0.0
   4       0.5      0.0      0.0
   5       0.5      1.0      0.0
   6       1.0      0.0      0.0
   7       1.0      0.5      0.0
   8       1.0      1.0      0.0
   9       0.0      0.0      0.5
  10       0.0      1.0      0.5
  11       1.0      0.0      0.5
  12       1.0      1.0      0.5
  13       0.0      0.0      1.0
  14       0.0      0.5      1.0
  15       0.0      1.0      1.0
  16       0.5      0.0      1.0
  17       0.5      1.0      1.0
  18       1.0      0.0      1.0
  19       1.0      0.5      1.0
  20       1.0      1.0      1.0
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
  1    1  2  3  5  8  7  6  4
  2   13 14 15 17 20 19 18 16
  3    1  2  3 10 15 14 13  9
  4    6  7  8 12 20 19 18 11
  5    1  4  6 11 18 16 13  9
  6    3  5  8 12 20 17 15 10
  NORMAL  1 +
  SAMPLING POINTS
       21      0.5    0.0    0.5
       22      0.5    0.25   0.5
       23      0.5    0.5    0.5
       24      0.5    0.75   0.5
       25      0.5    1.0    0.5
```

****BCSET**

```
**BCSET
  ID DISP1
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 3
  DISP 1
    SPLIST ALL
    T 1     0.0

**BCSET
  ID DISP2
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 5
  DISP 2
    SPLIST ALL
    T 1     0.0

**BCSET
  ID DISP3
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 1
  DISP 3
    SPLIST ALL
    T 1     0.0

**BCSET
```

```
    ID TRAC1
    VALUE
    GMR GMR1
    SURFACE SURF1
      ELEMENTS 4
    TRAC 1
      SPLIST ALL
      T 1     1000.0

  $ END OF DATA
```

TIP: It is considered to be a good practice to comment GPBEST data files using $.

Regarding the structure of the above file, please note: the dollar sign, $, signifies a comment. GPBEST users are encouraged to use comments freely in data files. Comments may appear at the beginning of a line or anywhere on a line. If a $ sign appears in the middle of a line, anything after the $ sign is ignored by GPBEST.

```
**CASE    $ Analysis type, output requests,
          $ post-processing

**MATE    $ Material parameters (density, E)

**GMR     $ Geometry definition (regions,
          $ nodes, elements)

**BCSET   $ Boundary conditions, loads
```

The general structure of a GPBEST data file is given by the lines beginning with **. Each of these sections correspond to sections in Chapter 5 of the GPBEST User's Manual (Volume I).

Running GPBEST

This problem is ready to run. The following illustrates how to submit a GPBEST run in the background and how to check on its run status using the `ps` command:

TIP: To run GPBEST in the foreground, omit the &. This symbol causes UNIX to submit the given command to the background, returning the system prompt to the user to execute other commands.

```
$ < elas605.dat best-s > elas605.res &
$ ps
PID TTY      TIME COMD
 21155 term/a   0:00 ps
 20123 term/a   0:01 csh
 21154 term/a   0:03 best-s     (while job is running)
$ ls -l          (after job finishes)
total 92
-rw-r-----   1 john     2682 Mar 20 12:37 elas605.dat
-rw-r-----   1 john    43772 Mar 20 12:37 elas605.res
$
```

The `ps` command is one way to check if GPBEST is still running the problem. In the above example, `best-s` has been running for three seconds. Also, if the files are listed in the current directory while the problem is running (using the `ls` command), many `FT***` files (`FT001`, `FT003`, etc.) will appear. These are temporary files GPBEST uses while executing. If the job finishes without crashing, the `FT***` files will be deleted by GPBEST. As long as these files exist, GPBEST is probably still running. Use the `ps` command to be certain.

In the above example, the file `elas605.dat` is *redirected* (using the < symbol) to be standard input for GPBEST execution. The standard output is redirected into the text file `elas605.res`. This file is quite large, so it will not be printed here. However, it can be viewed using the `cat` or `more` command, or by using a text editor such as `vi` or `emacs`.

Checking A Completed Run

TIP: This section is extremely important. It contains the basic checks to perform *with every GPBEST run* to insure that the problem has run properly.

The question that may rise at this point is "Did this job run to completion, and are the results correct?" To answer this, it is best to use some basic UNIX tools and a knowledge of the GPBEST results file.

1. The size of the results file above is shown to be `43772 bytes`. This is a good sign. If this number were very small, say `0` or `200 bytes`, then the job did not run successfully.
2. The job took about six seconds to execute. This is typical for a small job. Any input errors probably would have caused the job to terminate within a second or two.
3. Use the `grep` command to search for words which may indicate that there might have been problems running the job. Searching for words such as `WARNING`, `FATAL`, `MESSAGE`, etc. can be helpful. Use the `grep` command to search for such words in the results file:

Searching The GPBEST Results File (elas605.res) For Possible Error Messages

```
$ grep "ERROR" elas605.res
          ****  ERROR SUMMARY AFTER INPUT PHASE  ****
 FATAL ERRORS:                0
 PERCENT ERROR  0.20569E-02   0.97736E-03   0.97549E-03
      THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS

$ grep "FATAL" elas605.res
      FATAL ERRORS:               0

$ grep "WARN" elas605.res
      WARNING MESSAGES:           0

$ grep "MESSAGE" elas605.res
      WARNING MESSAGES:           0
      INFORMATIONAL MESSAGES:     0
$
```

TIP: The UNIX script `runbest.sh` in Appendix C illustrates how some of these common `grep` commands can be executed as part of the job submittal. For the time being, the user can just type in the above `grep` commands manually.

1. In the foregoing session, it does not appear that there were any major problems running this job. However, this is not a *guarantee* of success.
2. A quick visual check of the output in the GPBEST file can give the user a feeling as to whether the job ran properly. Open the file `elas605.res` in a text editor. The following sections of output should be realized:
   a) ECHO of input file.
   b) GPBEST pre-processing information, summary of number of nodes, elements, boundary conditions, etc.
   c) GPBEST information output while executing the job.
   d) GPBEST output (results at nodes, elements, etc.)
3. If the results section contained all zeros, then there must have been an error. However, in this problem, the results are non-zero, which is a good sign. Of course, this does not guarantee that the results are correct. It merely indicates that a non-zero solution was obtained.

TIP: For more information on the L*oad Balance* calculation, see Section 4.5.5 and Section 5.1.10 of the GPBEST User's Manual (Volume I).

1. The *Load Balance* calculation can be one of the quickest and most useful ways to check a GPBEST run. Basically, the load balance takes the solution vector and calculates the surface load over each element. Then, these are added up over each GMR. The resulting load balance number is often quite small when compared with the magnitude of the applied load. This can be used to indicate the overall quality of the solution results. In addition, the

load balance calculation also outputs the surface area and volume of the boundary element model. This can often highlight problems with dimensions, units, or modeling errors. The “total applied load” can also be used to determine if the overall applied load is correct.

The following is an excerpt from the GPBEST results file. Notice that the surface area and volume calculations are correct for a unit cube. The percent error in load balance (in the *X*-direction) is small with respect to the total load. Also, the total applied load in the *X*-direction is 1000.0, which is also correct for this problem. These checks give us further confirmation that the job ran properly.

Listing of the Load Balance Portion of the GPBEST Results File (elas605.res)

```
     BOUNDARY LOAD    -0.20508E-01    0.97439E-02    0.97655E-02
     INTERFACE LOAD    0.00000E+00    0.00000E+00    0.00000E+00
     LOAD BALANCE     -0.20508E-01    0.97439E-02    0.97655E-02
 SURFACE AREA =0.6000E+01,          VOLUME =0.1000E+01

     TOTAL LOAD BALANCE AND APPLIED LOAD BALANCE:
                              X              Y              Z
       TOTAL BALANCE  -0.20508E-01    0.97439E-02    0.97655E-02
       PERCENT ERROR   0.20508E-02    0.97438E-03    0.97655E-03
       APPLIED LOAD    0.10000E+04    0.00000E+00    0.00000E+00

     THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS
     CALCULATED WITH RESPECT TO THE
     MAGNITUDE OF VECTOR OF APPLIED LOAD BALANCE =     0.10000E+04
```

The above is a good basic list of checks to run for *every* GPBEST job. It only takes a few seconds to run through these checks, but they can save the user hours of work if a spurious result goes undetected. For now we will assume that the run completed running without any major errors.

Checking Accuracy Of The Solution

> ***TIP:*** In the `vi` editor, to quickly find the results portion of the `*.res` file, search for the string `'ST RE'`. This will locate the character string 'GPBE**ST RE**SULTS'. For example, the following keystrokes will locate the results section in a `vi` session:
> `$ vi elas605.res <CR>`
> `/ST RE<CR>`

In order to determine if the solution is correct, it is necessary to scrutinize the results in file `elas605.res`, utilizing basic engineering skills and insight. In this case, the problem involves a simple elastic cube in uniform tension, so an analytical solution is available. For a beam of constant cross-section, the displacement is given by $\delta_x = \frac{F\,l}{AE}$, where F=1000.0, l=1, A=1, E=10^7. This results in a resulting deflection of $\delta_x = 0.0001$, which occurs everywhere on element 4. Now go into the `elas605.res` file and determine if all of the nodes connected to element 4 have this deflection. An excerpt of the results file is given here. Your section of the results file should look similar to this:

Listing Of A Portion Of The GPBEST Results File (elas605.res)

```
      ------------------------------------------------------------------
                 ****          GPBEST RESULTS           ****
      ------------------------------------------------------------------


           JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
           BOUNDARY SOLUTION AT TIME =      1.000000    FOR REGION = GMR1
ELEMENT  NODE NO.    X DISPL      Y DISPL      Z DISPL    X TRACTION ...
     1       1      .00000E+00   .00000E+00   .00000E+00   .00000E+00
     1       2      .00000E+00  -.15005E-04   .00000E+00   .00000E+00
     1       3      .00000E+00  -.30004E-04   .00000E+00   .00000E+00
  . . .
     4       6      .10000E-03   .00000E+00   .00000E+00   .10000E+04
     4       7      .10001E-03  -.14999E-04   .00000E+00   .10000E+04
     4       8      .99993E-04  -.29991E-04   .00000E+00   .10000E+04
     4      12      .10000E-03  -.29996E-04  -.15000E-04   .10000E+04
     4      20      .99985E-04  -.29994E-04  -.29994E-04   .10000E+04
     4      19      .10000E-03  -.15000E-04  -.29996E-04   .10000E+04
     4      18      .99993E-04   .00000E+00  -.29991E-04   .10000E+04
     4      11      .10001E-03   .00000E+00  -.14999E-04   .10000E+04
  . . .
```

As indicated in the boxed area, the GPBEST deflection agrees with the analytical solution. Even though this does not guarantee complete accuracy of the problem, the user has a high degree of confidence in the results at this point.

This completes the first part of this Chapter. At this point, you should feel comfortable searching the GPBEST examples directory with the `grep` command. You should also be able to run GPBEST and look at the results to determine if the job ran successfully. The remaining sections in this chapter contain other information that GPBEST users need to know.

Submitting GPBEST Jobs

There are several aspects of job submittal that the user should understand. First, there may be several versions of GPBEST installed on your system. The listing of the `/bin/` directory in the GPBEST File Structure section in Chapter 2 lists several versions of GPBEST executables.

For the most part, you will usually use the `best-s` command to execute GPBEST. However, it is useful to be aware of the other versions of GPBEST available for specific needs. In addition, you may also have a customized version of GPBEST on your system.

Versions Of GPBEST Executables

Version	Description
best-s	This includes all quasi-static analyses which involve the use of real arithmetic. 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.
best-s60	This is similar to `best-s`, except the problem size is increased to 60,000 degrees of freedom. Use this version when the problem exceeds 24,000 d.o.f.
best-t	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-s` will not run for the given problem.

When submitting jobs, you may use the procedure outlined earlier in this chapter (using redirection). However, as you become more familiar with GPBEST and UNIX, you may find yourself using repetitive sequences of commands every time you run GPBEST. For example, you could submit the job using `best-s`, search for error messages in the results file, note how big the results file is, etc.

TIP: The scripts `runbest.sh` and `fromto.sh` are not included in the GPBEST distribution.

To be able to execute these UNIX scripts, you must either place them in your working directory or place them in a directory that is called out in your PATH statement in your startup file.

Appendix C contains an example of a UNIX script, `runbest.sh`, which can be used to submit GPBEST jobs. This is not intended to be used for every job submittal. Rather, it is intended to *illustrate* how a UNIX shell script can be written to automate repetitive tasks. Modify it to suit your needs.

This particular script creates an abbreviation file (`*.abr`), which contains a summary of the results file (`*.res`). By listing the abbreviation file after the job runs, the user can quickly perform several checks on the results file. A typical job submittal using `runbest.sh` is as follows:

```
$ cd $HOME/work
$ ls -l
total 6
-rw-r--r--   1 john          2682 Mar 19 19:06 elas605.dat

$ runbest.sh elas605
Running gp-best with input file: elas605.dat

$ ls -l
total 98
-rw-r--r--   1 john          1013 Mar 19 20:13 elas605.abr
-rw-r--r--   1 john          2682 Mar 19 19:06 elas605.dat
-rw-r--r--   1 john         44580 Mar 19 20:13 elas605.res

$ cat elas605.abr
                ****  ERROR SUMMARY AFTER INPUT PHASE  ****
      FATAL ERRORS:                0
        PERCENT ERROR   0.20508E-02    0.97438E-03    0.97655E-03
      THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS
      FATAL ERRORS:                0
      WARNING MESSAGES:            0
      INFORMATIONAL MESSAGES:      0
      TOTAL LOAD BALANCE AND APPLIED LOAD BALANCE:

                              X              Y              Z
        TOTAL BALANCE  -0.20508E-01    0.97439E-02    0.97655E-02
         APPLIED LOAD   0.10000E+04    0.00000E+00    0.00000E+00
        PERCENT ERROR   0.20508E-02    0.97438E-03    0.97655E-03

      THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS
      CALCULATED WITH RESPECT TO THE
      MAGNITUDE OF VECTOR OF APPLIED LOAD BALANCE =     0.10000E+04



           JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION

Start: Tue Mar 19 20:13:05 EDT 1996
End:   Tue Mar 19 20:13:16 EDT 1996

real       11.0
user        5.8
sys         2.4
total 96
-rw-r--r--   1 john          2682 Mar 19 19:06 elas605.dat
-rw-r--r--   1 john         44580 Mar 19 20:13 elas605.res
-rw-r--r--   1 john          1013 Mar 19 20:13 elas605.abr
$
```

A reference is made to another script, `fromto.sh`, in the script `runbest.sh` . The `fromto.sh` script is similar to the UNIX `grep` command, except `fromto.sh` searches through a file for all the text between lines containing *string1* and *string2*. For instance, to extract the lines containing the “Load Balance” portion of the results file:

```
$ fromto.sh 'TOTAL LOAD BALANCE' 'JOB TITLE' elas605.res

      TOTAL LOAD BALANCE AND APPLIED LOAD BALANCE:

                              X              Y              Z
        TOTAL BALANCE  -0.20508E-01    0.97439E-02    0.97655E-02
         APPLIED LOAD   0.10000E+04    0.00000E+00    0.00000E+00
        PERCENT ERROR   0.20508E-02    0.97438E-03    0.97655E-03

      THE PERCENTAGE ERROR IN THE TOTAL BALANCE IS
      CALCULATED WITH RESPECT TO THE
      MAGNITUDE OF VECTOR OF APPLIED LOAD BALANCE =     0.10000E+04



            JOB TITLE:      A UNIT CUBE IN SIMPLE TENSION
$
```

The `fromto.sh` script can be used any time to search through text files and extract lines between two search strings. It is a very useful general purpose UNIX script.

CHECK Run

In the case of the previous problem, you were able to look at the GPBEST data file (`elas605.dat`), but were not able to view it graphically on your computer. For such a small problem, this is not a great concern. However, for larger problems, it is useful to be able to view the geometry graphically before running it in GPBEST. In many cases, you may not understand the problem until you see a picture of the geometry.

TIP: The `CHECK` keyword causes GPBEST to stop just before the solver phase. The `NEUTRAL` keyword causes model geometry to be written to a GPBEST Neutral file.

The `CHECK` command can be added to the `**CASE` section of the GPBEST data file. This is called a "Check Run." Adding the `CHECK` line will cause GPBEST to:

1. Read in the data file.
2. Perform initial checks on the data.
3. Output model geometry to the GPBEST Neutral file.

Here are the first few lines of the file `elas605.dat` with the inclusion of the `CHECK` and `NEUTRAL` lines:

Example Of GPBEST Input File To Perform A "Check Run"

```
**CASE
  TITLE      A UNIT CUBE IN SIMPLE TENSION
  ELASTICITY
  ECHO
  CHECK
  NEUTRAL

**MATE
  ID MAT1
  TEMP  70.0
  EMOD 10.E6
  ...
```

TIP: It is a good practice to run a "Check Run" before submitting a problem for a complete GPBEST analysis. This permits you to view the model geometry first, as well as alerting you to possible problems with the input data.

GPBEST stops before actually solving the model. The "Check Run" will typically take only a few seconds to complete.

A benefit of doing a "Check Run" is that the GPBEST Neutral file will contain geometry that can be viewed in a graphics package. BESTVIEW can read in a GPBEST Neutral file directly. PATRAN and I-DEAS require the POSTBEST translator to create the geometry files these systems recognize. The tutorials in Chapters 4 and 5 illustrate how to read the GPBEST Neutral file into BESTVIEW and how to translate it using POSTBEST for PATRAN and I-DEAS. Chapter 6 indicates how other CAD systems interface with the GPBEST Neutral file.

Units And Dimensions In GPBEST

As a GPBEST user, you have immense flexibility in choosing units you want to work with. If you desire SI units, that is fine. If you prefer English units, that is fine also. The only requirement that you must be aware of is that the units must be consistent from one entity to another. For instance, the units on model dimensions, pressure loads and Young's modulus must all be consistent.

The danger with mixing units is that GPBEST will execute the job flawlessly without alerting the user to possible problems. **It is the user's responsibility to insure that the units are consistent.**

TIP: Please turn to Appendix B now. You will want to refer to it often until you become comfortable with units used in GPBEST input.

Appendix B is supplied to provide two sets of consistent units. However, the user can use whatever system of units are preferred. The table in Appendix B provides a starting point. We suggest that you refer to it often. Also, as a new GPBEST user, you may want to run a few simple problems with known solutions (like the block in `elas605.dat`), thus allowing you to check to make sure the units (and results) are correct.

Finding Other Example Problems

As mentioned previously, the UNIX `grep` command is extremely useful in finding problems in the GPBEST `examples` directory. You may want to develop some confidence in searching the `examples` directory, since this is a useful starting point when using a new feature or capability in GPBEST.

Use the UNIX `cd` command to go to the `examples` directory. Using the directory name from Chapter 2, this is (it may be different for your system):

```
$ cd /usr/local/gpbest/examples/ex.small
$
```

Perhaps you can search for the following capabilities, using `grep` commands such as those shown:

Search for:	Using the command:
Data files containing hole elements	`grep ”HOLE” *.dat`
2-D Elastic analysis (`PLANE`)	`grep ”PLAN” elas*.dat`
Axisymmetric model (`AXISYMMETRIC`)	`grep ”AXIS” *.dat`
2-D elastic analysis with traction load (`TRACTION`)	`grep -l ’PLAN’ `grep -l ’TRAC’ elas*.dat` `
Anisotropic material properties (`ANISOTROPIC`)	`grep ”ANIS” *.dat`
Gravity loading (`INERTIA`)	`grep ”INER” *.dat`

The GPBEST.FEATURES File

A new user of GPBEST is exposed to a seemingly endless number of cards and options, primarily in Chapter 5 of the GPBEST User’s Manual. Many new users wonder which options are supported for their analysis. Relevant to this tutorial manual, one may ask: Which GPBEST cards are supported for elastic analysis? Can anisotropic material properties be specified in an elastic analysis? What about quartic boundary elements? How

about gravitational body forces? Most of these questions can be answered through a brief discussion of the GPBEST.FEATURES file.

TIP: It is highly recommended that a new user create the GPBEST.FEATURES file and print it out for ready reference.

If the PRIN FEAT (PRINT 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 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:

Example Of GPBEST Input File Used To Create GPBEST.FEATURES File

```
$ cat feat.dat
  **CASE
    TITLE      JUST PRINT OUT GPBEST.FEATURES FILE
    PRINT FEATURES
    CHECK
  $ END OF FILE
$ < feat.dat best-s > feat.res
$
```

Once the above file is input to GPBEST, the file GPBEST.FEATURES is created. Since it is 132 characters wide, it should be printed on wide paper or by 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, the following is an excerpt from 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 the following table:

Status Levels Of Special Features In GPBEST.FEATURES File

```
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 RELEVANT TO THIS ANALYSIS  (FATAL ERROR)
```

The second section that is output to the `GPBEST.FEATURES` file is listed in the next table. Each analysis type is assigned an alphabetical letter. For example, ‘P’ is used for ELASTOSTATICS ANALYSIS.

Analysis Types In GPBEST.FEATURES File

```
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 ANAYSIS
_
E = THERMOVISCOUS FLUID - TRANSIENT, LINEAR ANALYSIS
_
F = THERMOVISCOUS FLUID - TRANSIENT, NON-LINEAR ANALYSIS
```

The four basic cross-reference tables included in `GPBEST.FEATURES` are

1. Two-dimensional analysis
2. Three-dimensional analysis
3. Axisymmetric analysis
4. Generalized axisymmetric analysis

The following table shows another excerpt from `GPBEST.FEATURES`. In particular, the features (or options) are listed in the left 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

ELASTOSTATIC ANALYSIS (P) because the implementation status of these two items is a ‘1’. 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. Most options are supported for ELASTOSTATIC ANALYSIS.

Listing Of A Portion Of The GPBEST.FEATURES File

```
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
 . . .
```

In addition to the `GPBEST.FEATURES` file, Appendix E lists the keywords supported by elastic analysis in GPBEST. This is a condensed listing of the pages in Chapter 5 of the GPBEST User’s Manual (Volume I). If you are only running elastic analyses, then the listing in Appendix E will prove very helpful, since it is not cluttered with options that are not supported.

GPBEST Limits

Another important listing to have for ready reference is the list of current GPBEST limits. This will help the user determine how many nodes can be included 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 (`*.res`) when `PRINT LIMITS` is included in `**CASE`.

The following table lists the current limits for the basic version of GPBEST (i.e., **`best-s`**) which is limited to 24,000 d.o.f. Some users have enhanced versions of GPBEST which permit larger problems to be run. Users with special versions of GPBEST should print out their own limits, since they will likely be different from those shown in this table.

To print out the limits for your current version of GPBEST, run the following short input file:

Example Of GPBEST Input File To Print Out Program Limits

```
$ cat limits.dat
  **CASE
    TITLE  PRINT OUT CURRENT GPBEST LIMITS TO RESULTS FILE
    PRINT LIMITS
    CHECK
  $ END OF FILE
$ < limits.dat best-s > limits.res
$
```

Listing Of Current GPBEST Limits (best-s Version)

(Your particular version of GPBEST may vary somewhat)

```
            -------------------------------------
              GPBEST LIMITS FOR CURRENT VERSION
            -------------------------------------

    GLOBAL PARAMETERS
    -----------------
    TOTAL GEOMETRIC MODELING REGIONS           =   40
    TOTAL DEGREES OF FREEDOM                   =   24000
    TOTAL POINTS (INCLUDING NON-SOURCE POINTS) =   16000
    TOTAL BOUNDARY SOURCE POINTS               =   8000
    TOTAL VOLUME POINTS
      (VOLUME SOURCE POINTS + SAMPLING POINTS) =   8000
    TOTAL VOLUME SOURCE POINTS                 =   4000
    TOTAL VOLUME SOURCE POINTS (FOR FLUIDS)    =   100
    TOTAL BOUNDARY ELEMENTS                    =   4000
    TOTAL HOLE AND INSERT ELEMENTS             =   400
    DISCONTINUITY ELEMENTS                     =   100
    ENCLOSING ELEMENTS                         =   100
    CYCLIC SYMMETRY INTERFACES                 =   20

    REGION (GMR) PARAMETERS
    -----------------------
    SURFACES IN ANY REGION                     =   15
    REGION DEGREES OF FREEDOM                  =   10000
    POINTS IN ANY REGION                       =   9000
    BOUNDARY SOURCE POINTS IN ANY REGION       =   8000
    VOLUME SOURCE POINTS IN ANY REGION         =   4000
    BOUNDARY ELEMENTS IN ANY REGION            =   4000
    HOLE AND INSERT ELEMENTS IN ANY REGION     =   400
    VOLUME CELLS IN ANY REGION                 =   1333

    OTHER PARAMETERS
    ----------------
    DEGREES OF FREEDOM IN RESIDUAL SYSTEM      =   10000
    TABLE POINTS (B.C./BODY FORCES/ETC.)       =   20
    TEMPERATURE POINTS FOR MATERIAL PROPERTIES =   21

    FIXED PARAMETERS
    ----------------
    SPATIAL DIMENSIONS                         =   3
    DEGREES OF FREEDOM PER SOURCE POINT        =   4
    DEGREES OF FREEDOM FOR FLUIDS              =   3
    TOTAL STRESS COMPONENTS                    =   6
    TOTAL COUPLED "STRESS" COMPONENTS          =   7
    POINTS IN AN ELEMENT                       =   25
    POINTS IN A 2D ELEMENT                     =   5
    POINTS IN A HOLE OR INSERT ELEMENT         =   9
    POINTS IN A CELL                           =   25
    POINTS IN A 2D CELL                        =   25

    SPECIAL FIXED PARAMETERS
    ------------------------
    SOURCE POINTS PER 3D ELEMENT IN MODEL      =   3
    USER DEFINED HOLE OR INSERT POINTS         =   3
    LANCZOS TIME POINTS                        =   50
    ERROR MESSAGES STORED                      =   500


WORK SPACE ALLOCATION
---------------------

 TEMPORARY STORAGE ARRAY FOR ASSEMBLY, ETC.
    ALLOCATED =   2148000
    TYPE      = STATIC ALLOCATION

 TEMPORARY STORAGE ARRAY FOR GEOMETRY AND MATERIAL
    ALLOCATED =   320000
    TYPE      = STATIC ALLOCATION

 DEFAULT ARRAY SIZE FOR SOLVING EQUATION SYSTEM
    ALLOCATED =   24 MEGABYTES
    TYPE      = DYNAMIC ALLOCATION
```

CHAPTER 4 - TUTORIAL #2: 2-D PROBLEM

Introduction

This chapter contains a very simple 2-D example which illustrates the key steps of how to create a boundary element model in a CAD program, output a GPBEST data file for running a boundary element analysis, and post-processing the GPBEST results in a graphical program. After going through this tutorial, you should be able to pre-process and post-process simple boundary element models.

It Is Estimated That It Will Take About One Hour To Go Through The Tutorial For Each Modeling Program In This Chapter

The procedure is presented in several steps: First, the BESTVIEW program is illustrated, then the PATRAN program, then the I-DEAS program, then the HYPERMESH program, and finally the FEMAP program. Each of these tutorials are presented in parallel, so the user can easily see how the steps relate in the various programs.

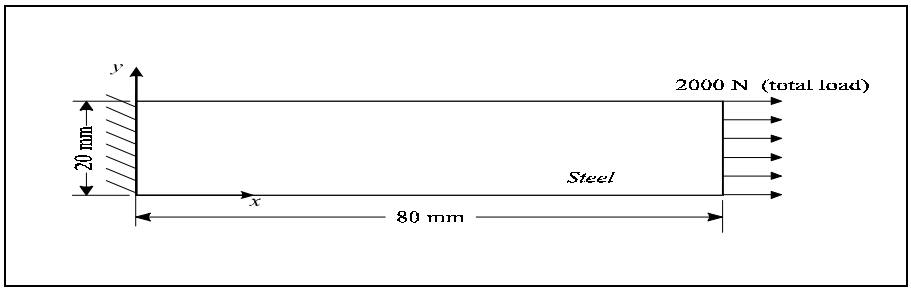
Even if you use a different CAD program for pre-processing (geometry creation, meshing, etc.), it is still recommended that users become familiar with the BESTVIEW program. BESTVIEW is highly integrated with the GPBEST command. Many users find some functions much more efficient in BESTVIEW as opposed to some finite element pre- and post-processors. In particular, defining boundary conditions and post-processing results are very easily accomplished in BESTVIEW. (Of course, users must have a BESTVIEW license to run BESTVIEW.)

BESTVIEW can be used in conjunction with other programs. For instance, PATRAN could be used to create the geometry and build a mesh. The PATRAN neutral file could be read into BESTVIEW and boundary conditions could be applied in BESTVIEW.

Problem Specification

A cantilever beam is shown in the following figure. It is fixed at the wall and is free to move at the free end (tip). A static force of 2000 N acts in the outward (*+X*) direction at the tip. The object is to determine the stress and deformation state of the beam under the given load.

Beam Geometry



A Note About The Units Used In GPBEST

This is an appropriate point to make mention of units used in conjunction with GPBEST. The user can use any system of units and dimensions that they desire. However, they must be consistent. Units for length, mass, loads, displacements, and so forth must all be consistent. For instance, if mass is specified using a length unit of meters while the model dimensions are specified in mm, the results will not be correct.

This bookkeeping can be a burden for a new user who may be merely trying to learn how to run the GPBEST program. For this reason, we have included a preferred set of units for English and Metric systems in Appendix B. Feel free to refer to this list often, as it will prove helpful in specifying consistent units.

For the sake of checking the GPBEST results, it is useful to note the following beam and geometry calculations.

Analytical Solution

Maximum deflection at the tip of the beam (approximate):

$$\delta_{tip} = \frac{FL}{AE} = \frac{(2000N)(80mm)}{(20mm \times 1mm)(207000\,N/mm^2)} = 0.03865mm$$

Stress:

$$\sigma_{xx} = \frac{F}{A} = \frac{2000N}{20mm \times 1mm} = 100\,{}^{N}\!/_{mm^2}, \quad (everywhere)$$

Length around region:

$$L = 80 + 20 + 80 + 20 = 200mm$$

Planar area (or planar volume):

$$V = 80 \times 20 = 1600 mm^2$$

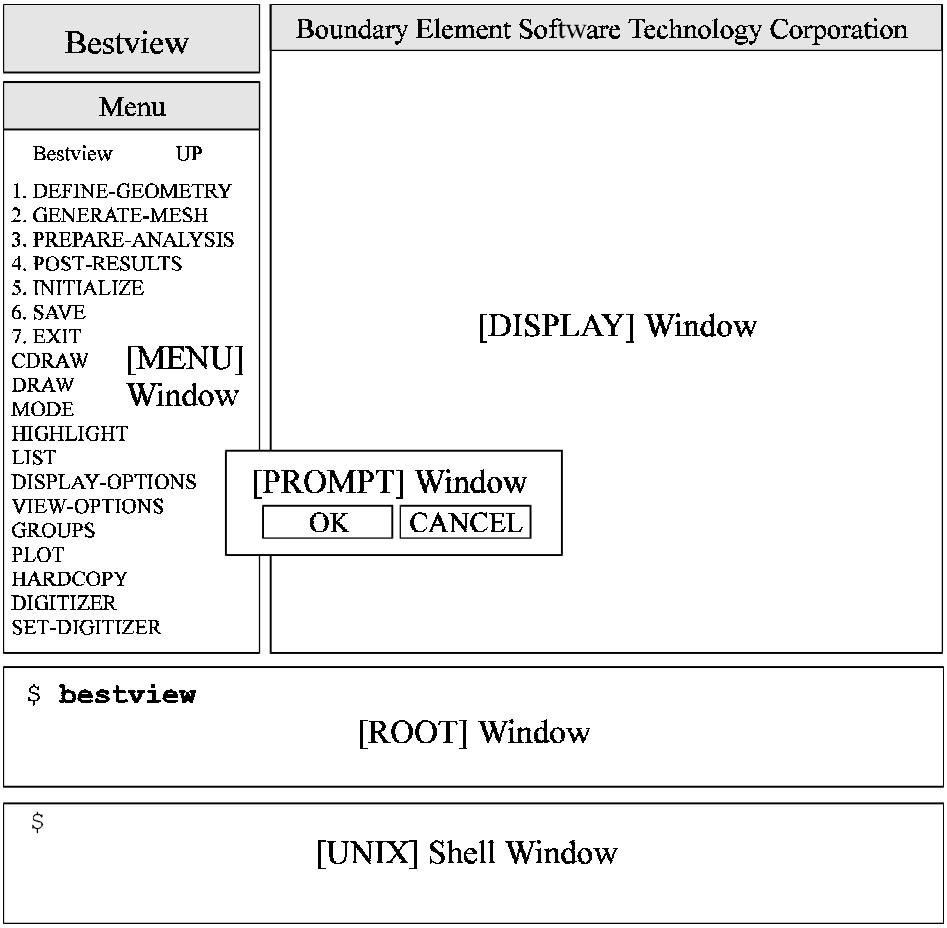
BESTVIEW Model

The BESTVIEW program has been designed to integrate well with the GPBEST program. This allows users to create boundary element models, import geometry from other CAD systems, and display results.

TIP: This tutorial covers BESTVIEW 2.5 or higher.

The following figure illustrates the complete BESTVIEW screen. It also highlights the names for the various windows. These names will be used throughout this tutorial to indicate the specific window to access and where to type in commands and responses.

BESTVIEW Window Names



The purpose of each of these BESTVIEW windows is summarized in this table:

The Purpose Of Each BESTVIEW Window

BESTVIEW Window Name	Window Function
[MENU]	This is the main command window. Some menu commands will result in another submenu displayed in this area. Select commands with a mouse click.
[DISPLAY]	This is the main display window where the graphics, model, coordinate axes, etc. are displayed. Certain entities can be selected directly from this window.
[PROMPT]	This window appears when the user must enter specific data. Sometimes the choice is a simple YES/NO response. At other times, filenames, geometrical coordinates, rotation angles, etc. must be entered here.
[ROOT]	This is the UNIX window used to start the BESTVIEW program. It is suggested that this window be placed at the bottom of the screen so that it is visible. It is used to display listings of data, to prompt the user to carry out a specific action, or to accept data input.
[UNIX]	This is a UNIX shell window independent of the BESTVIEW program. UNIX commands, such as the print command (`lpr`), can be entered here.

Step 1: Start BESTVIEW

In this step, you will start the BESTVIEW program and name the database you will be working on. If BESTVIEW does not display properly, review the BESTVIEW installation steps in Chapter 2. A common problem involves not specifying the GPBEST `password` file properly. If problems persist, call the Boundary Element Software Technology Corporation for assistance.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[ROOT]	**`bestview`**	Start BESTVIEW.
[PROMPT]	`<ESC>` **`tut2a`** `[OK]`	Name of new database.
[MENU]	`SET-DIGITIZER`	Turn DIGITIZER mode OFF.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[PROMPT]	[OFF]		

Step 2: Define The Four Corner Points Of The Beam

The geometry of the beam is defined by four corner points and four lines defining the edge of the beam. These are merely geometry and have nothing to do with the nodes and elements. However, the points and lines will be used to construct a boundary element mesh in subsequent steps.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	DEFINE-GEOMETRY		
[MENU]	POINT		
[MENU]	ENTER		
[PROMPT]	<ESC> 1 [OK]		First POINT number is 1.
[PROMPT]	0,0,0 <CR>		Point 1.
[PROMPT]	80,0,0 <CR>		Point 2 (Length is 80 mm).
[PROMPT]	80,20,0 <CR>		Point 3 (Height is 20 mm).
[PROMPT]	0,20,0 <CR>		Point 4.
[PROMPT]	<ESC>		
[MENU]	DEFI.POIN	UP	Move up one menu.
[MENU]	CDRAW		At this point, the [DISPLAY] window should appear as shown in the following figure.

Geometric Points Defining The Corners Of The Beam

Step 3: Define Four Geometrical Lines, One For Each Edge Of The Beam

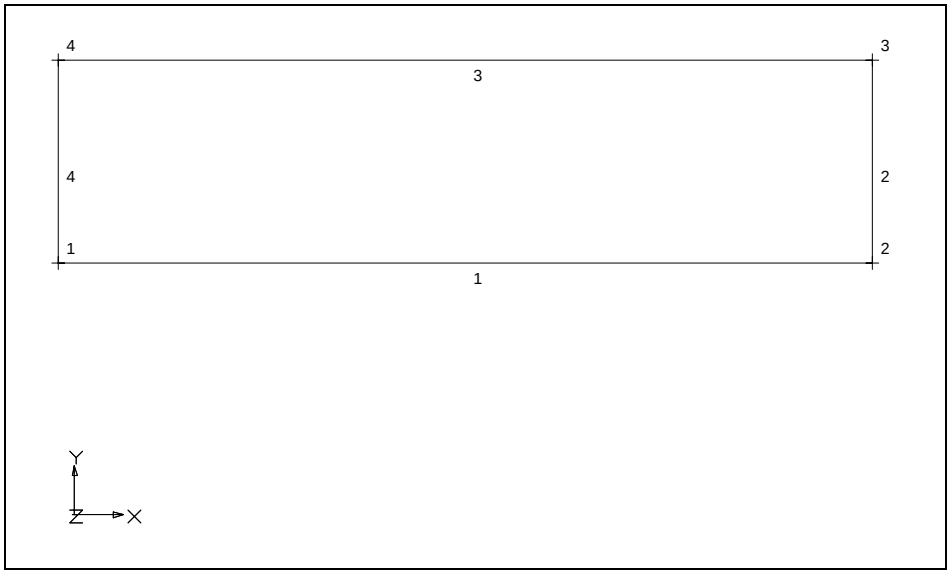
The four points defined in the previous step can be used to construct geometrical lines. The lines are defined using the 2-point option.

These lines are not boundary elements. They are merely geometrical entities which will be used to create boundary elements later.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`LINE`	
[MENU]	`ENTER`	
[PROMPT]	`<ESC> 1 [OK]`	First LINE number is 1.
[PROMPT]	`1,2 <CR>`	Line 1 appears in [DISPLAY] window.
[PROMPT]	`2,3 <CR>`	Line 2.
[PROMPT]	`3,4 <CR>`	Line 3.
[PROMPT]	`4,1 <CR>`	Line 4.
[PROMPT]	`<ESC>`	End input. The [DISPLAY] window should appear as in the figure below.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`DEFI.LINE` `UP`	Move up one menu.
[MENU]	`DEFI` `UP`	Move up one menu.

Geometric Points And Lines



Step 4: Define A GROUP

Groups are useful for meshing purposes. After the four lines are placed in a group, the group can be used to create a boundary element mesh.

Groups are also useful for defining multiple-region models. Groups in BESTVIEW correspond to GMRs in GPBEST. These concepts will be explained further in Chapter 5.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`GROUPS`	
[MENU]	`ADD ENTITY`	
[PROMPT]	`beam1 [OK]`	Name of GMR.
[MENU]	`LINE`	
[PROMPT]	`<ESC> 1,4 [OK]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[PROMPT]	`YES`		Save current changes.
[PROMPT]	`<ESC> MAGENTA [OK]`		Color of group. Color choices are displayed in the [ROOT] window.
[MENU]	`GLOB.GROU`	`UP`	Move up one menu.

Step 5: Create A Boundary Element Mesh

The group, `beam1`, created in the previous step, is now used to generate the boundary element mesh. The boundary element mesh consists of boundary elements and nodes. Only one boundary element will be created on each edge of the beam. Each boundary element will contain three nodes (two end nodes and one mid node), hence, it is called a *quadratic* element. Note that equivalencing of nodes is unnecessary. BESTVIEW takes care of this detail automatically.

Note that an interior mesh is not needed; a mesh is only required on the boundary of the region. This is a tremendous modeling advantage inherent in the boundary element method.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	`GENERATE MESH`		
[MENU]	`FAST-MESH`		
[ROOT]	`0.005 <CR>`		A message appears in the [ROOT] window to “Give search radius:”.
[ROOT]	`1 <CR>`		A message appears in the [ROOT] window to “Give mesh density for patches <n>”. Specify only one element in each direction.
[MENU]	`GENE`	`UP`	Move up one menu. The mesh has been created.

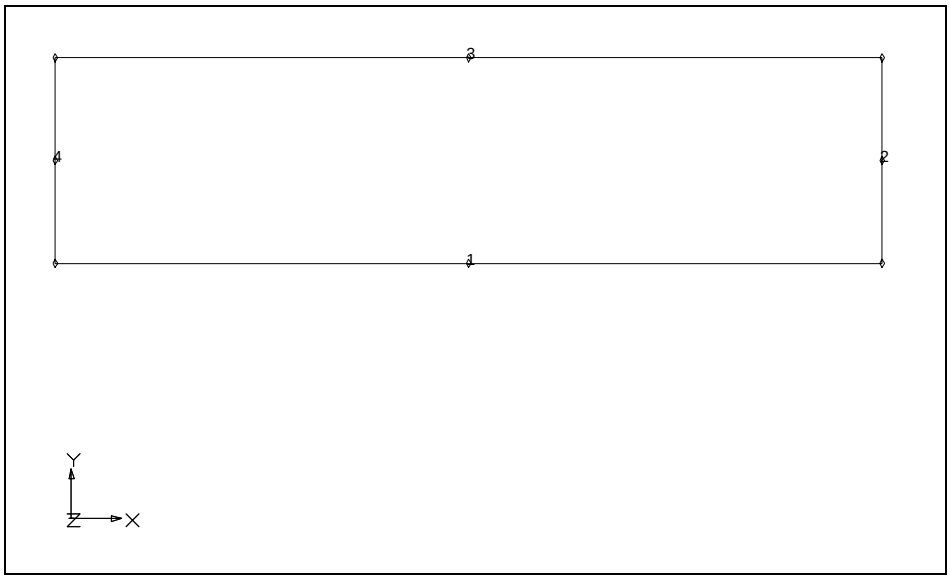
Step 6: Display Boundary Element Mesh

At this point, it is useful to learn how to manipulate the display in BESTVIEW. In this step, specific entities will be selected to be displayed. The point and line display will be turned off since these entities are no longer needed. A plot of the element numbers and node numbers will be useful.

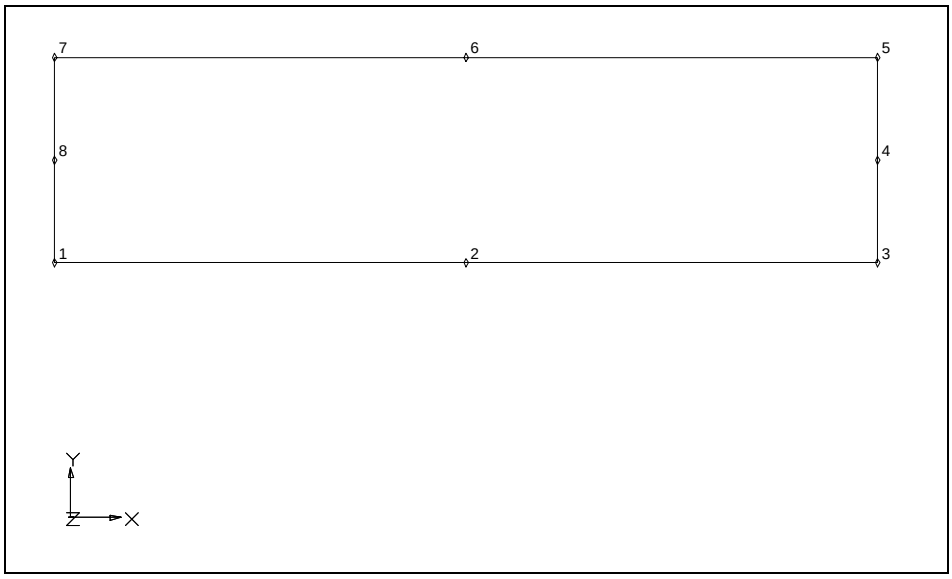
INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	DISPLAY-OPTIONS		
[MENU]	ENTITY		
[MENU]	ALL		
[PROMPT]	[OFF]		Turn everything OFF.
[MENU]	MESH		
[PROMPT]	[ON]		Turn only MESH items ON (NODES and ELEMENTS).
[MENU]	GLOB.DISP.ENTI	UP	Move up one menu.
[MENU]	LABEL		
[MENU]	MESH		
[PROMPT]	OFF		
[MENU]	ELEMENT		
[MENU]	ON		
[MENU]	CDRAW		See [DISPLAY] window in figure below. Only element numbers are displayed.
[MENU]	ALL		
[MENU]	OFF		
[MENU]	NODE		
[PROMPT]	ON		
[MENU]	CDRAW		See [DISPLAY] window in figure below. Only node numbers are displayed.
[MENU]	GLOB.DISP.LABE	UP	Move up one menu.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`GLOB.DISP` `UP`	Move up one menu.

Element Numbers



Node Numbers



Step 7: Prepare The GPBEST Data File (PREPARE ANALYSIS)

The boundary element model has been defined in terms of nodes and elements. Proceed to prepare the GPBEST data file. As

noted in Chapter 3, the GPBEST data file consists of four basic sections:

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

These four sections are constructed in this same order in BESTVIEW.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	PREPARE-ANALYSIS	
[ROOT]	<CR>	Press <CR> to continue with "Prepare Analysis".
[MENU]	CASE	
[MENU]	TITLE	
[ROOT]	SIMPLE BEAM <CR>	
[MENU]	DIMENSION	
[MENU]	2. PSTS : PLANE STRESS	
[MENU]	1. NONE	No symmetry.
[MENU]	PREP.CASE.DIME UP	
[MENU]	ANALYSIS TYPE	
[MENU]	ELASTIC	
[MENU]	PREP.CASE.ANAL UP	
[MENU]	PREP.CASE UP	**CASE input has been completed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	MATERIAL	Define material parameters.
[MENU]	DEFINE	
[ROOT]	STEEL <CR>	Name of the beam material.
[ROOT]	I <CR>	Isotropic.
[ROOT]	0 <CR>	Temperature T=0.
[ROOT]	207000.0 <CR>	Young's Modulus $\left(\frac{N}{mm^2}\right)$.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`0.3 <CR>`	Poisson's Ratio.
[ROOT]	`7.833E-9 <CR>`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen that are consistent with dimensions of the model which were mm.
[MENU]	`PREP.MATE UP`	Move up one menu. **MATE input has been completed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`GMR`	
[MENU]	`NO`	Not an axisymmetric model.
[ROOT]	`1 <CR>`	Thickness.
[ROOT]	`0 <CR>`	Reference temperature T=0.
[MENU]	`NEXT GMR`	End of only GMR in the model.
[PROMPT]	`[NO]`	No reference points to be defined. **GMR input has been completed.

Step 8: Define The Boundary Conditions For The Model

While in PREPARE DATA, the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X* and *Y* degrees of freedom on element 4.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST. Typically, loads in GPBEST are specified as distributed loads, or tractions.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	BCSET	
[ROOT]	WALL <CR>	Name of boundary condition set. Fix the displacements at the left edge of the beam.
[ROOT]	G <CR>	Global value.
[ROOT]	S <CR>	Selected elements.
[PROMPT]	[NO]	Don't use digitizer to select.
[PROMPT]	<ESC> A <CR>	Add entities.
[ROOT]	4 <CR>	Element 4.
[ROOT]	. <CR>	Period (end input).
[ROOT]	C <CR>	In [DISPLAY] window, element 4 is highlighted. Confirm that this is correct.
[ROOT]	A <CR>	All nodes on element 4.
[ROOT]	N <CR>	No time variation.
[ROOT]	D <CR>	Displacement.
[ROOT]	1 <CR>	Constrain *X*-direction.
[ROOT]	0 <CR>	Zero magnitude.
[ROOT]	C <CR>	Confirm.
[ROOT]	Y <CR>	Proceed to constrain *Y*-direction.
[ROOT]	G <CR>	Global value.
[ROOT]	N <CR>	No time variation.
[ROOT]	D <CR>	Displacement.
[ROOT]	2 <CR>	Constrain *Y*-direction.
[ROOT]	0 <CR>	Zero magnitude.
[ROOT]	C <CR>	Confirm.
[ROOT]	N <CR>	No more BCs in WALL BCSET.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	BCSET		Proceed to define the distributed load (traction) at the right edge.
[ROOT]	TIP <CR>		Name of boundary condition set.
[ROOT]	G <CR>		Global value.
[ROOT]	S <CR>		Selected elements.
[PROMPT]	[NO]		Don't use digitizer to select.
[PROMPT]	<ESC> A <CR>		Add entities.
[ROOT]	2 <CR>		Element 2.
[ROOT]	. <CR>		Period.
[ROOT]	C <CR>		In [DISPLAY] window, element 2 is highlighted.
[ROOT]	A <CR>		All nodes on element 2.
[ROOT]	N <CR>		No time variation.
[ROOT]	T <CR>		Traction.
[ROOT]	1 <CR>		X-direction.
[ROOT]	100. <CR>		Magnitude. Note that this is a force per unit area. The total force is 2000 N. The dimension of the tip is 20 mm (and unit thickness). Therefore, 2000/20 = 100. N/mm.
[ROOT]	C <CR>		Confirm.
[ROOT]	N <CR>		No more BCs.
[MENU]	PREP	UP	Move up one menu.
			**BCSET input has been completed.
[MENU]	EXIT		Proceed to BESTVIEW exit.
[PROMPT]	[YES]		Save this database.
[PROMPT]	[YES]		Exit BESTVIEW now.

Step 9: Run GPBEST To Analyze The Boundary Element Model

> ***TIP:*** The GPBEST neutral file contains BEM model information and results. It is used primarily for post-processing.

At this point, the GPBEST data file has been created by BESTVIEW. Its name is `tut2a.DAT`. Proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut2a.DAT best-s > tut2a.res`**
[UNIX]	**`mv NEUTRAL.OUT tut2a.neu`**

> ***TIP:*** Check your GPBEST results file before post-processing. It does not do much good to view the results graphically if fatal errors were encountered in the GPBEST run.

Step 10: Read The Neutral File Into BESTVIEW For Post-Processing

You can view the GPBEST results file (`tut2a.res`) and determine if GPBEST ran successfully using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, proceed to view the results in BESTVIEW. For this step, it is not necessary to open up the previously saved database. The GPBEST neutral file contains the boundary element model and results information; which is all that is needed.

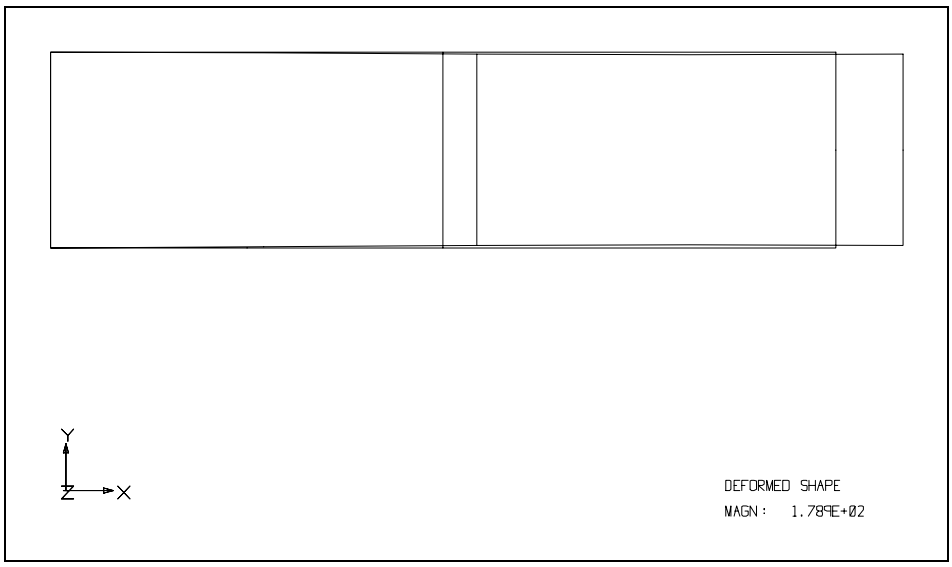
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`bestview`**	Start BESTVIEW.
[PROMPT]	`<ESC> tut0.db [OK]`	Specify a new name for the database.
[MENU]	`POST-RESULTS`	
[MENU]	`READ-GPBEST-RES.`	Read in NEUTRAL file.
[ROOT]	`tut2a.neu`	Specify the name of the NEUTRAL file from the GPBEST run.
[MENU]	`CDRAW`	This displays the boundary element model in the [DISPLAY] window.

Step 11: Create A Deformed Shape Plot

Plot the deformed beam along with the original (undeformed) geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`DEFORMED-GEOMETRY`	Still in the POST-RESULTS menu.
[PROMPT]	`[ON]`	Turn deformed plot ON.
[PROMPT]	`MANUAL [OK]`	
[PROMPT]	`[OK]`	Accept the default scale factor for the deformed shape.
[MENU]	`CDRAW`	
[PROMPT]	`[YES]`	Display both undeformed and deformed geometry. At this point, the [DISPLAY] window appears as shown in the following figure.

Deformed Geometry Plot



TIP: Note that there is an extra vertical line in the middle of the beam. This is merely added by GPBEST to aid the user in visualizing interior behavior while post-processing 2-D models. It is a direct consequence of the `CONTOUR` card in the `**CASE` block in the GPBEST file.

Step 12: Create A Color Fringe Stress Plot

The Von Mises stress plot is easily displayed by changing the MODE. For this problem, the stress is uniform in the interior. Note however, that the interior stress is displayed even though there really is no mesh in the interior. This is because GPBEST automatically creates a temporary interior mesh for 2-D problems for interior post-processing purposes. The CONTOUR card in the **CASE data block triggers this step. It is not necessary for the user to create an interior mesh for post-processing purposes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	`MODE`		
[MENU]	`FRINGE-PLOT`		
[MENU]	`CDRAW`		The Von Mises stress is displayed on a fringe plot.
[MENU]	`POST`	`UP`	Proceed to exit BESTVIEW.
[MENU]	`EXIT`		
[PROMPT]	`[NO]`		Do not save this session.
[PROMPT]	`[YES]`		

The Data File
`tut2a.DAT`

```
**CASE
 TITLE  SIMPLE BEAM
 PLANE STRESS
 ELASTIC
 TIME  1.0
 ECHO
 NEUTRAL
 CONT

**MATE
ID   STEEL
TEMP    0.0000000000E+00
EMOD     207000.0000
POIS    0.3000000119
DENS    0.7833000026E-08

**GMR
ID beam1
THIC     1.000000000
MATE   STEEL
TREF    0.0000000000E+00
TINT    0.0000000000E+00
 POIN
    1      .000000E+00      .000000E+00      .000000E+00
    2      .400000E+02      .000000E+00      .000000E+00
    3      .800000E+02      .000000E+00      .000000E+00
    4      .800000E+02      .100000E+02      .000000E+00
    5      .800000E+02      .200000E+02      .000000E+00
    6      .400000E+02      .200000E+02      .000000E+00
    7      .000000E+00      .200000E+02      .000000E+00
    8      .000000E+00      .100000E+02      .000000E+00
SURF beam1
 TYPE QUAD SERE
 ELEM
     1     1     2     3
     2     3     4     5
     3     5     6     7
     4     7     8     1
NORM    +

**BCSET
ID  WALL
 VALUE
GMR beam1
SURF beam1
ELEM              4
DISP     1
SPLIST ALL
T  1    0.0000000000E+00
DISP     2
SPLIST ALL
T  1    0.0000000000E+00

**BCSET
ID  TIP
 VALUE
GMR beam1
SURF beam1
ELEM              2
TRAC     1
SPLIST ALL
T  1     100.0000000
```

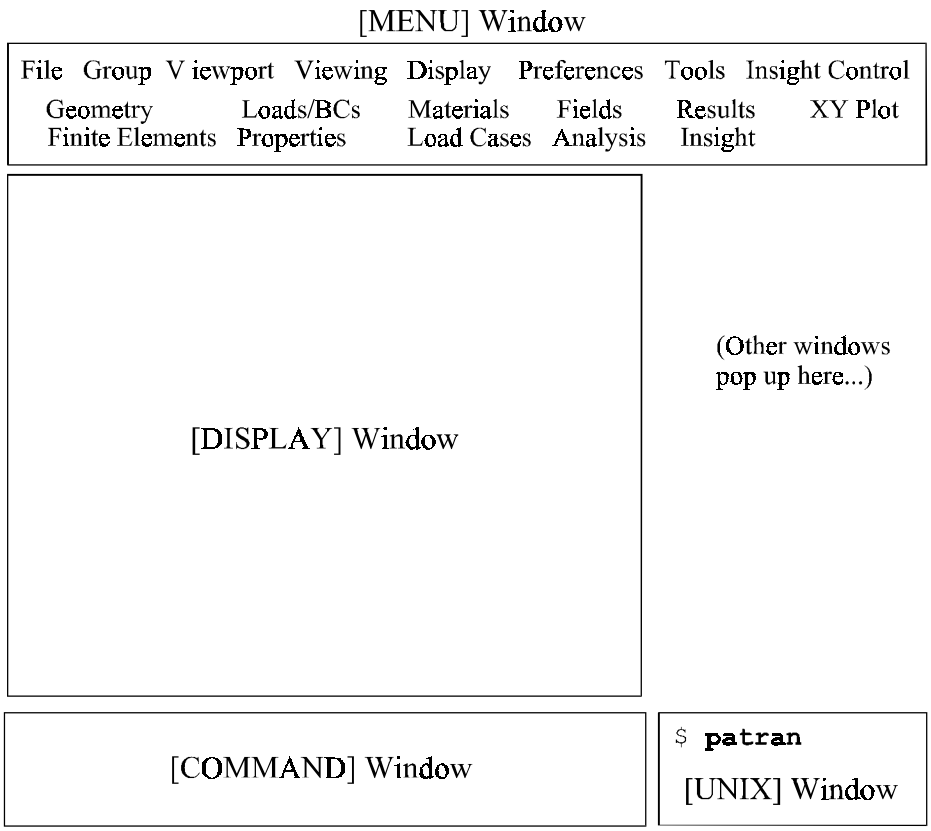
PATRAN Model

It is assumed that the PATRAN user is familiar with geometry creation, meshing, etc. in PATRAN. This tutorial is not intended to teach how to use PATRAN. It is intended to show an experienced PATRAN user how to interface with the GPBEST program. Some steps in the modeling process, such as boundary condition specification, require special attention. This tutorial will illustrate the special steps to follow to create a GPBEST boundary element model and view the results of a GPBEST analysis in PATRAN.

TIP: This tutorial covers MSC/PATRAN 7.5 or higher.

The names of the PATRAN windows are given in the following figure:

The PATRAN Window Names



Step 1: Start PATRAN

In this step, you will start the PATRAN program and name the database that will be used..

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**patran**	Start PATRAN.
[MENU]	File	
[MENU]	New...	
[New Database]	New Database Name: tut2b [OK]	This will name the database tut2b.db.
[New Model Preferences]	[OK]	Accept defaults.

Step 2: Define The Four Corner Points Of The Beam

The geometry of the beam is defined by four corner points and four lines defining the edge of the beam. These are merely geometry and have nothing to do with nodes and elements. However, the lines will be used to construct a boundary element mesh in subsequent steps.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	Geometry	Two windows are displayed.
[Geometry]	Action: Create Object: Point Method: XYZ	
[Geometry]	[Apply]	Accept (0,0,0) for first point.
[Geometry]	Point Coordinates List: [80,0,0] [Apply]	Point 2.
[Geometry]	Point Coordinates List: [80,20,0] [Apply]	Point 3.
[Geometry]	Point Coordinates List: [0,20,0] [Apply]	Point 4.
		The four points should appear in the [DISPLAY] window.

Geometric Points Defining The Corners Of The Beam

Step 3: Define Four Geometrical Lines, One For Each Edge Of The Beam

The four points defined in the previous step can be used to construct geometrical lines.

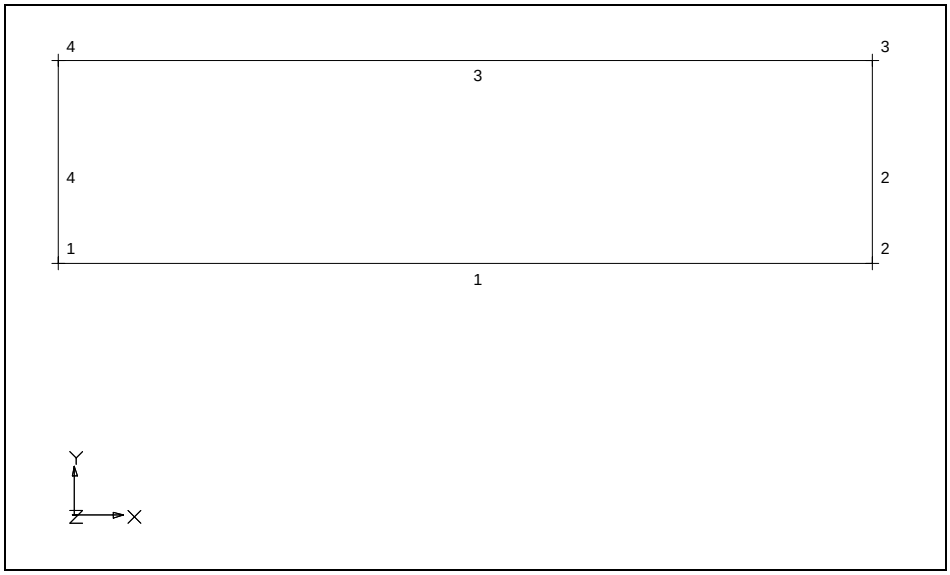
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Geometry]	`Object: Curve`
	`Method: Point`
	`Option: 2 Point`	

[Geometry]	`Starting Point List:` `Point 1`	As an alternative to this approach, you could also pick the points in the [DISPLAY] window using the mouse.
[Geometry]	`Ending Point List:` `Point 2`	
[Geometry]	`[Apply]`	LINE 1 is created.
[Geometry]	`Starting Point List:` `Point 2`	
[Geometry]	`Ending Point List:` `Point 3`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Geometry]	`[Apply]`	LINE 2 is created.
[Geometry]	`Starting Point List:` `Point 3`	
[Geometry]	`Ending Point List:` `Point 4`	
[Geometry]	`[Apply]`	LINE 3 is created.
[Geometry]	`Starting Point List:` `Point 4`	
[Geometry]	`Ending Point List:` `Point 1`	
[Geometry]	`[Apply]`	LINE 4 is created.

Geometric Points And Lines



Step 4: Define Groups

Note that for this simple problem, it is not necessary to define a GROUP in PATRAN. More complex models consisting of multiple regions require the use of GROUPS. This will be examined in the following chapter.

Step 5: Define Material Properties

Material properties are now defined. Start by defining the STEEL material property. Note that the material property only needs to be

defined in PATRAN. It does not need to be associated with any elements. The PREBEST translator will extract the material properties from the neutral file.

In a finite element model consisting of beam elements, element properties would usually be defined also. However, this step is unnecessary for a GPBEST model since the only information needed about the boundary elements is their nodal connectivity. Therefore, element properties, such as area, thickness, etc. are not needed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Materials`	Define material parameters.
[Materials]	`Action: Create`	
	`Object: Isotropic`	
	`Method: Manual Input`	
[Materials]	`Material Name: Steel`	
[Materials]	`Input Properties...`	
[Input Options]	`Constitutive Model:`	
	`Linear Elastic`	
[Input Options]	`Elastic Modulus:207000.`	Young's modulus $\left(\frac{N}{mm^2}\right)$.
[Input Options]	`Poisson Ratio: 0.3`	
[Input Options]	`Density: 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.
[Input Options]	`[Apply]`	
[Input Options]	`[Cancel]`	Note that it is not necessary to assign "Property" information (Area, moment of inertia, beam orientation, etc.) to elements.

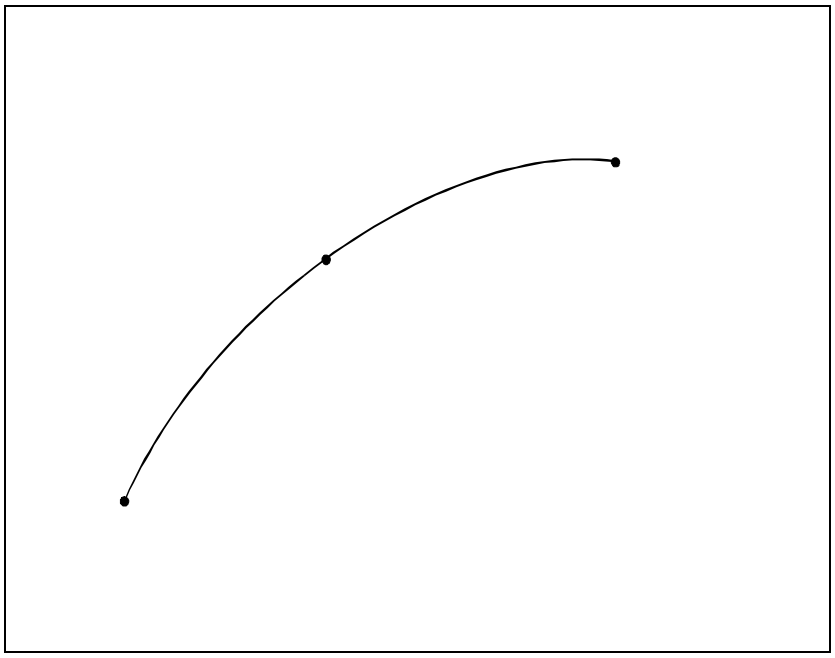
Step 6: Create A Boundary Element Mesh

The boundary element mesh is now created. It consists of boundary elements and nodes. Only one boundary element will

be created on each edge of the beam. Each boundary element will contain three nodes (two end nodes and one mid node), hence it is called a *quadratic* element.

Although PATRAN does not support boundary elements directly, boundary elements are simulated using beam elements in 2-D. These beam elements do not require properties, such as area, orientation, etc., since they are not true beams. The translator step (PREBEST) later converts them into GPBEST boundary elements.

2-D Boundary Element



As illustrated in this figure, make sure you use 3-noded beam elements in PATRAN. Although 2-noded beam elements are more popular in finite element modeling, it is preferable to use 3-noded elements in GPBEST for 2-D problems.

Equivalencing and renumbering of nodes and elements are not necessary if a mesh is created in BESTVIEW. BESTVIEW automatically equivalences and renumbers entities. The user must include this extra step when building a model in PATRAN.

Note that an interior mesh is not needed; a mesh is only required on the boundary of the region. This is a tremendous modeling advantage inherent in the boundary element method.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Finite Elements`	
[Finite Elements]	`Action: Create`	
	`Object: Mesh Seed`	
	`Type: Uniform`	
[Finite Elements]	`Number of Elements`	
	`Number=1`	
[Finite Elements]	`Curve List: Curve 1:4` `[Apply]`	
[Finite Elements]	`Object: Mesh`	
[Finite Elements]	`Type: Curve`	
[Finite Elements]	`Element Topology: Bar3`	
[Finite Elements]	`Curve List: Curve 1:4` `[Apply]`	
[Finite Elements]	`Action: Equivalence`	Equivalence coincident nodes.
	`Object: All`	
[Finite Elements]	`Equivalencing Tolerance: .005`	
[Finite Elements]	`[Apply]`	
[Finite Elements]	`Action: Renumber`	Renumbering is not required. It is merely done for the user's convenience.
[Finite Elements]	`Start ID: 1`	
	`Node List: Node 1:#`	
[Finite Elements]	`[Apply]`	Nodes are renumbered from 1 to 8.

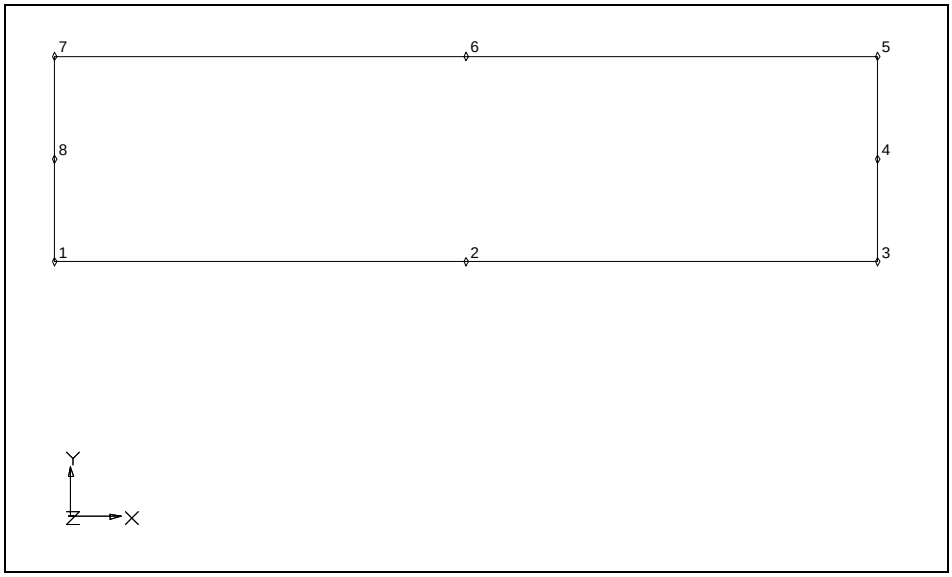
Step 7: Display Boundary Element Mesh

At this point, the model display in PATRAN can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Display`	
[MENU]	`Plot/Erase...`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Plot/Erase]	`[Erase All Geometry]`	The points and lines no longer need to be displayed.
[Plot/Erase]	`[OK]`	
[MENU]	`Display`	
[MENU]	`Finite Elements ...`	
[FEM Attributes]	`(next to BAR elements, click on the LABEL box)`	This turns OFF bar labels. Only node labels remain ON.
[FEM Attributes]	`[Apply] [Cancel]`	The model, along with node number labels, appears in the [DISPLAY] window.

Node Numbers



Step 8: Define The Boundary Conditions For The Model

> ***TIP**:* Note that GPBEST boundary conditions are not specified using the traditional FEM approach in PATRAN. While working with 2-D models in PATRAN, the GPBEST user must use nodal displacements to specify both loads and boundary conditions. You must specify all nodes on an element to apply a boundary condition to the element in 2-D models.

Now the boundary conditions can be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X* and *Y* degrees of freedom on element 4.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual has an extensive description on specifying GPBEST boundary conditions in PATRAN. It would be best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, in 2-D, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 110 is used at the wall, since both *X* and *Y* displacements are zero there. Load set 101 is used at the tip of the beam, since the *X*-traction is non-zero there. For 2-D problems, loads and boundary conditions are specified using *nodal constraints*.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Load Cases`	First, define the load cases applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.
[Load Cases]	`Action: Create`	
[Load Cases]	`Load Case Name: 101`	*X*-component of traction is non-zero at tip of beam.
[Load Cases]	`[Apply]`	
[Load Cases]	`Load Case Name: 110`	*X*- and *Y*-components of displacement are zero at the wall.
[Load Cases]	`[Apply]`	
[Load Cases]		All load cases that may be used are listed in the [Existing Load Cases] section.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Load/BCs`	

[Load/Boundary Condition]	`Action: Create`
	`Object: Displacement`
	`Type: Nodal`
	`Analysis type: Structural`	

[Load/Boundary Condition]	`Current Load Case: (Click on this box to select next load set.)`	Work on the WALL BC first.
[Change Current Load Case]	`(Select load case 110)` `[OK]`	110=zero displacement in the *X*- and *Y*-directions.
[Load/Boundary Condition]	`New Set Name: WALL`	
[Load/Boundary Condition]	`[Input Data...]`	
[Input Data]	`Translations: < 0 >` `[OK]`	Only one value is needed inside the brackets.
[Input Data]	`[Select Appl Region...]`	
[Select Application Region]	`Geometry Filter: FEM`	Turn on FEM filter.
[DISPLAY]	`(draw a box around the left edge of the beam)`	It is necessary to select *all* nodes on a given element where you want to apply boundary conditions. If only one node is selected, it will be ignored by the PREBEST translator. At this point, nodes 1, 7, and 8 have been selected.
[Select Application Region]	`[Add]` `[OK]`	In [Application Region], nodes 1,7, and 8 are listed.
[Load/Boundary Condition]	`[Apply]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Load/Boundary Condition]	`Current Load Case: (Click on this box to select next load set.)`	Now work on the traction load at the tip of the beam.
[Change Current Load Case]	`(Select load case 101)` `[OK]`	101=non-zero traction.
[Load/Boundary Condition]	`New Set Name: TIP`	
[Load/Boundary Condition]	`[Input Data...]`	
[Input Data]	`Translations: < 100 >` `[OK]`	Only one value is needed inside the brackets. Here, the traction magnitude is specified as 100 N/mm. Direction is specified by the Load set ID 101.
[Input Data]	`[Select Appl Region...]`	
[Select Application Region]	`Geometry Filter: FEM`	Turn on FEM filter.
[DISPLAY]	`(draw a box around the right edge of the beam)`	It is necessary to select *all* nodes on a given element where you want to apply boundary conditions. At this point, nodes 3, 4, and 5 have been selected.
[Select Application Region]	`[Add]` `[OK]`	Node 3:5 appears in [Application Region].
[Load/Boundary Condition]	`[Apply]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File`	
[MENU]	`Export...`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Export]	`Format: Neutral` `Create New File: /users/john/work/tut2b.pat`	This creates a PATRAN neutral file named `tut2b.pat`.
[Export]	`[Apply]`	
[Warnings]	`[OK]`	Ignore warning message.
[MENU]	`File`	
[MENU]	`Quit`	Exit from PATRAN. It is sometimes advantageous to examine the PATRAN "Journal File" (`tut2b.db.jou`). This file lists the commands and messages given in a PATRAN session.

> ***TIP:*** Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 9: Run GPBEST To Analyze The Boundary Element Model

At this point, the PATRAN neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`3 <CR>`	PATRAN.
[PREBEST]	`tut2b.pat <CR>`	PATRAN neutral file name.
[PREBEST]	`Y <CR>`	Yes - this is the correct file.
[PREBEST]	`tut2b.dat<CR>`	Write to `tut2b.dat` file.
[PREBEST]	`2 <CR>`	2D - Plane Stress.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`1 <CR>`	Default thickness is 1 mm.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`N <CR>`	No body forces.
		The PREBEST program ends, and the file `tut2b.dat` has been created. It can be run directly in GPBEST.
		Note that it is sometimes useful to examine the `patbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining if the model was completely translated or not.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut2b.dat best-s > tut2b.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut2b.neu`**

Step 10: Post-Processing The Results

TIP: Check your GPBEST results file before post-processing. It does not do much good to view the results graphically if fatal errors were encountered in the GPBEST run.

You can view the GPBEST results file (`tut2b.res`) and determine if GPBEST ran successfully using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, then proceed to view the results in PATRAN. For this step, it is best to start PATRAN with a new database.

Since PATRAN cannot read in the GPBEST neutral file directly, it is necessary to run the POSTBEST translator to create files that PATRAN will recognize.

Results could be imported directly into the original model database. However, here a new database will be created for results post-processing.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**postbest**<CR>	
[POSTBEST]	3 <CR>	PATRAN-3.
[POSTBEST]	tut2b.neu <CR>	Name of GPBEST neutral file.
[POSTBEST]	N <CR>	Do not renumber the nodes.
		Three files are created: PATRAN.GEOM PATRAN.DIS.1 PATRAN.NOD.1

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**patran**	Start PATRAN.
[MENU]	File	Specify a new name for the database.
[MENU]	New...	
[New Database]	New Database: tut0	
	[OK]	
[New Model Preferences]	[OK]	
[MENU]	File	Import model.
[MENU]	Import...	
[Import]	Object: Model	
	Source: Neutral	
	Import file: /users/john/work/PATRAN.GEOM	
[Import]	[Apply]	
[Message]	[YES]	Continue importing neutral file.
[Message]	[YES]	Commit to PATRAN database.
[MENU]	File	Import results.
[MENU]	Import...	
[Import]	Object: Results	
	Format: PATRAN 2.nod	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Template for PATRAN 2.5 Import Results]	`Files: (select gpbest_nod.res_tmpl)` `[OK]`	If this template does not appear in the selection window, it means that the system administrator omitted **Step 5** in the installation procedure (Chapter 2).
[Import]	`Import File: /users/john/work/PATRAN.NOD.1` `[Apply]`	
[Warnings]	`[OK]`	
[MENU]	`Display`	Turn off labels for post-processing.
[Display]	`Finite Elements ...`	

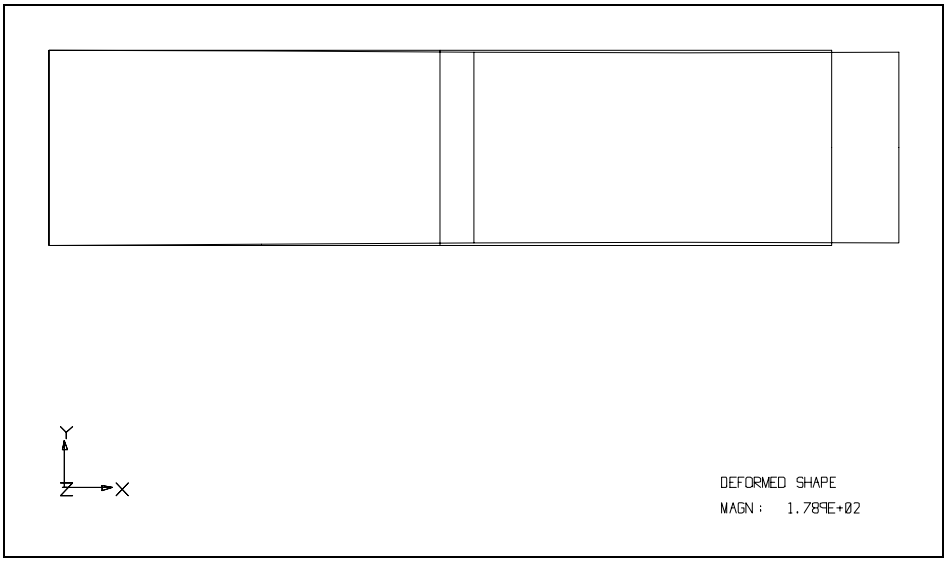
[FEM Attributes]	`[Hide All FEM Labels]`
	`[Apply]`
	`[Cancel]`	

Step 11: Create A Deformed Shape Plot

Plot the deformed beam along with the original (undeformed) geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Results`	
[Results]	`Action: Create` `Object: Deformation`	
[Results]	`Select Result:` `1.1 Displacement Components`	
[Results]	`Select Deformation Result: Displacement, Components`	
[Results]	`[Apply]`	

Deformed Geometry Plot



Step 12: Create A Color Fringe Stress Plot

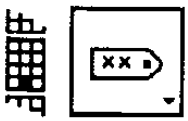
The Von Mises stress plot is easily displayed by changing the Results Display. For this problem, the stress is uniform in the interior. Note however, that the interior stress is displayed even though there really is no mesh in the interior. This is because GPBEST automatically creates a temporary interior mesh for 2-D problems for interior post-processing purposes. The CONTOUR card in the **CASE data block triggers this step. The user does not have to create an interior mesh for post-processing purposes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Results]	`Object: Fringe`	
[Results]	`Select Fringe Result: Equivalent (Von Mises) Stress`	
[Results]	`[Apply]`	
[MENU]	`File`	Proceed to exit PATRAN.
[MENU]	`Quit`	

I-DEAS Model

TIP: This tutorial covers I-DEAS Master Series 6 or higher.

It is assumed that the I-DEAS user is familiar with geometry creation, meshing, etc. in I-DEAS. This tutorial is not intended to teach how to use I-DEAS. It is intended to show an experienced I-DEAS user how to interface with the GPBEST program. Some steps in the modeling process, such as boundary condition specification, require special attention. This tutorial illustrates the special steps to follow in order to create a GPBEST boundary element model and view the results of a GPBEST analysis in I-DEAS.



Some of the icons that appear on the screen actually sit on top of other available icons. Since the needed icons in this tutorial may be hidden behind some other icons, we are using the helpful icons given in the *I-DEAS Student Guide*, Second Edition. These give the position of the icon in the [Icons] section in addition to the icon button. An example of one of these icons is shown here in the left margin.

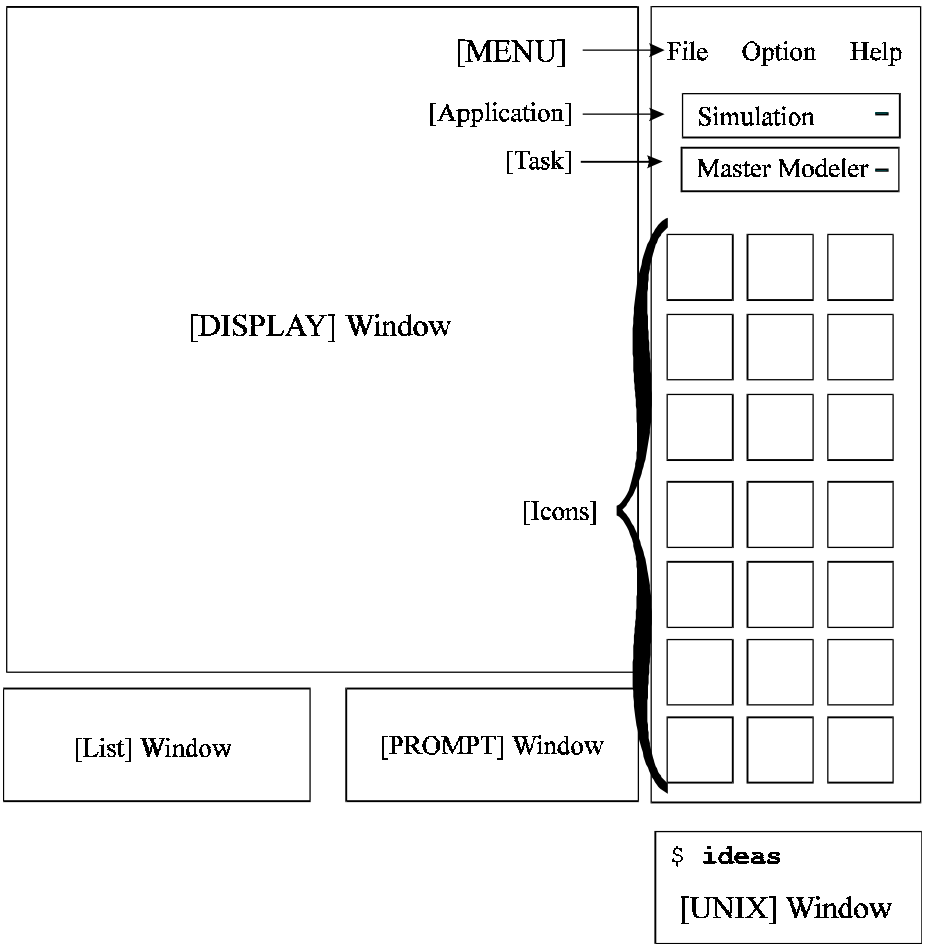
An Important Note About Units In I-DEAS

A word of explanation is required in regard to units used in I-DEAS. I-DEAS allows users the freedom to specify any set of units they want to work with. On the surface, this may seem to be an extremely useful feature. However, there is a drawback: internally, I-DEAS stores *all* data in SI (Meter-Newton) units. And more importantly, when exporting data for use with other programs, I-DEAS converts everything into SI units, even if that is not what you wanted.

The I-DEAS tutorials in this manual will assume that users can keep track of their own units. The units used throughout are the preferred set of units given in Appendix B (mm-Newton-sec). *However, in order to tell I-DEAS to turn off the units conversion, you must specify SI as the units you want to work with.* When you specify SI units, no conversion is made. Even though SI units may be suggested while using the program, just enter the desired dimensions, material properties, loads, etc. that you desire, such as mm, Newtons, etc. This tutorial should clarify the procedure.

The names of the I-DEAS windows are shown in the following figure:

The I-DEAS MS Window Names



Step 1: Start I-DEAS

In this step, you will start the I-DEAS program and name the database that will be used. It is also good to set some view settings before proceeding. Make sure you understand the information given above about units in I-DEAS.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`ideas`**	Start I-DEAS.
[I-DEAS Start]	`Project: john`	
	`Model File Name: tut2c.mf1`	
	`Application: Simulation`	
	`Task: Master Modeler`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[I-DEAS Start]	`[OK] [OK]`	Create model database.
[MENU]	`[Options]`	
[MENU]	`[Units]` `Meter (newton)`	Specify SI units in order to turn off unit conversion. See note, above.
[MENU]	`[Options]` `[View Settings]`	Set viewport diameter.
[View Settings]	`Viewport Diameter: 120` `[OK]`	

Step 2: Define The Geometry Of The Beam

I-DEAS allows the user to create very detailed geometry in 2-D and 3-D. In this case, the 2-D beam will be modeled by a “plane” of the appropriate dimensions. This is merely geometry and has nothing to do with nodes and elements.

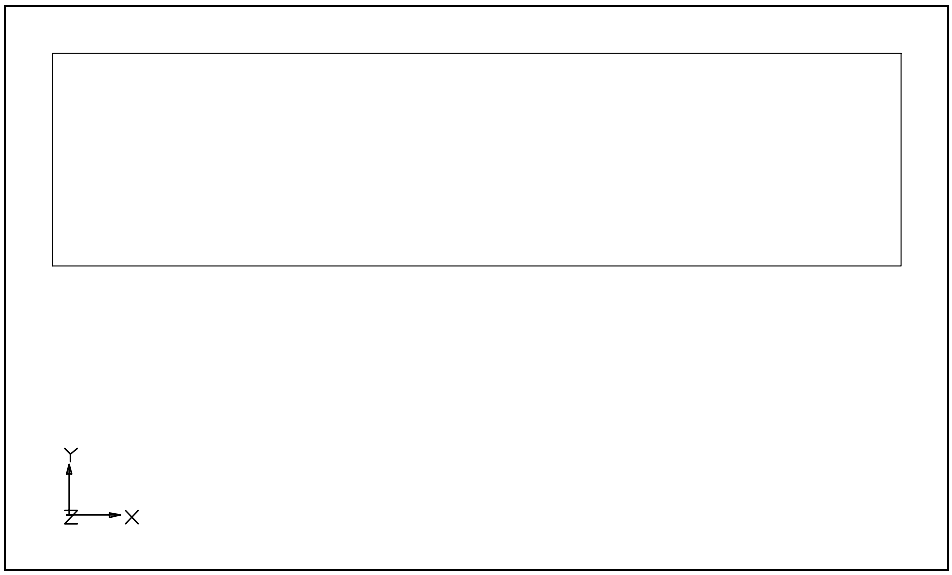
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Parts...`	

[Part Catalog]	`[Plane]`
	`X-Dimension: 80`
	`Y-Dimension: 20`
		The dimensions of the beam are 80mm x 20mm. See the note above about units in I-DEAS.

[Part Catalog]	`[OK]`	A plane defining the beam geometry has been created.
[Icons]	`Name` `Parts...`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Name]	`(move cursor until it is near the beam face F1 and it appears highlighted; click mouse)`	
[Name]	`Name: Beam1 [OK]`	

The Named Part Beam1

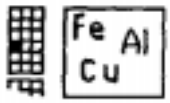


Step 3: Define A GROUP

For this simple problem, it is not necessary to define a GROUP in I-DEAS. More complex models consisting of multiple regions require the use of GROUPS. This will be examined in the next Chapter.

Step 4: Define Material Properties

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Task]	`[Master Modeler]` `change to [Meshing]`	Switch to FEM meshing mode.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Materials...`	Define material properties.
[Materials]	`Material Label: 1` `Name: Steel1`	
[Materials]	`[Quick Create]`	

[Quick Create]	`[Modulus of Elasticity]`
	`207000`
	`[Modify Value]`
		Young's Modulus $\left(\frac{N}{mm^2}\right)$.
		Ignore the units given by I-DEAS, since SI units are not being used. See the note at the beginning of this section.

[Quick Create]	`[Poisson ratio]`
	`0.3`
	`[Modify Value]`	

[Quick Create]	`[Mass Density]`
	`7.833E-9`
	`[Modify Value]`
		Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$.
		Notice that units are chosen consistent with dimensions of the model (mm).

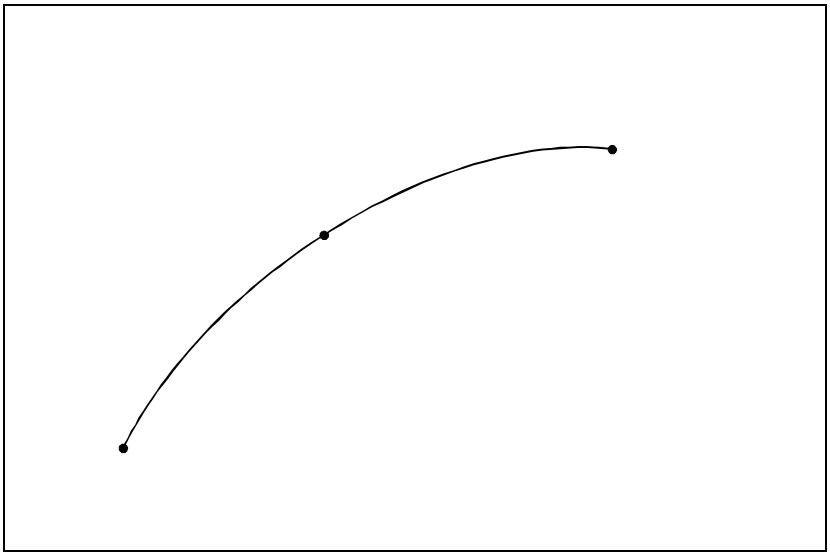
[Quick Create]	`[OK]`	
[Materials]	`[OK]`	

Step 5: Create A Boundary Element Mesh

The boundary element mesh is now created. It consists of boundary elements and nodes. Only one boundary element will be created on each edge of the beam. Each boundary element will contain three nodes (two end nodes and one mid node). Hence, it is called a *quadratic* element.

Although I-DEAS does not support boundary elements directly, boundary elements are simulated using beam elements in 2-D. These beam elements do not require properties, such as area, orientation, etc., since they are not true beams. The translator step (PREBEST) later converts them into GPBEST boundary elements.

2-D Boundary Element



As illustrated in this figure, make sure you use 3-noded beam elements in I-DEAS. Although 2-noded beam elements are more popular in finite element modeling, it is preferable to use 3-noded element in GPBEST when solving 2-D problems.

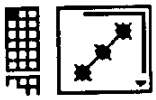
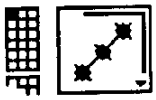
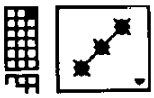
Equivalencing and renumbering of nodes and elements are not necessary if a mesh is created in I-DEAS. I-DEAS automatically equivalences and renumbers entities. However, for larger problems, the user may have to perform this step manually.

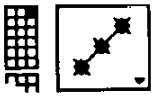
Note that an interior mesh is not needed; a mesh is only required on the boundary of the region. This is a tremendous modeling advantage inherent in the boundary element method.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Create FE Model...`	Proceed to generate the mesh.

[FE Model Create]	`Part: Beam1`
	`Material: Steel1`
	`FE Model Name: Fem1`	

[FE Model Create]	`[OK]`	

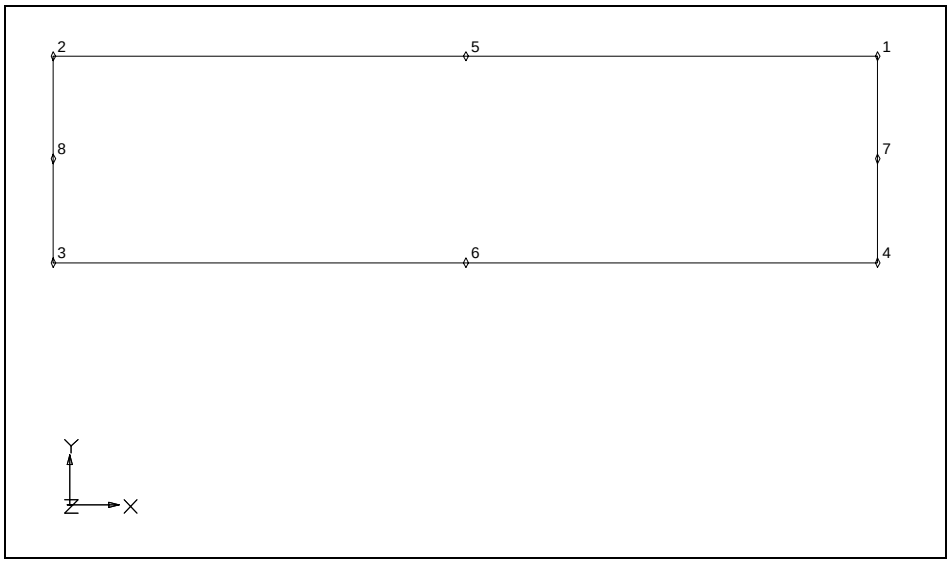
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Define Beam Mesh...`	Top and bottom edges.
[DISPLAY]	`(Hold down [Shift] key and select bottom and top edges)`	Holding down the [Shift] key allows you to select multiple entities in I-DEAS.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Define Mesh]	`Mesh Type: Free`	
	`Element Family: Beam`	
	`Element Type: 3-noded`	
	`Element Length: 80`	
[Define Mesh]	`[Apply] [OK]`	
[Icons]	 `Define Beam Mesh...`	Left and right edges.
[DISPLAY]	`(Hold down [Shift] key and select left and right edges)`	
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Define Mesh]	`Mesh Type: Free`	
	`Element Family: Beam`	
	`Element Type: 3-noded`	
	`Element Length: 20`	
[Define Mesh]	`[Apply] [OK]`	
[Icons]	 `Beam Mesh...`	Top and bottom edges.
[DISPLAY]	`(Hold down [Shift] key and select top and bottom edges of beam)`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[DISPLAY]	`[YES]`	Keep this mesh.
[Icons]	 `Beam Mesh...`	Left and right edges.
[DISPLAY]	`(Hold down [Shift] key and select left and right edges of beam)`	
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[DISPLAY]	`[YES]`	Keep this mesh. NOTE: These nodes were equivalenced automatically. In a larger model, they may have to be equivalenced manually.

Step 6: Display Boundary Element Mesh

At this point, you can manipulate the model display in I-DEAS. It is a good idea to check the model, connectivity, etc.

Node Numbers



***TIP**:* Note that the node numbers may differ from the model shown here. This will not affect the results of the tutorial.

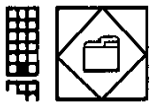
Step 7: Define The Boundary Conditions For The Model

> ***TIP**:* Note that GPBEST boundary conditions are not specified using the traditional FEM approach in I-DEAS. While working in I-DEAS, the GPBEST user must use distributed element loads to specify both loads and boundary conditions. Typically the loads and boundary conditions are distributed over elements, not at individual nodes. Please be aware of these aspects.

Now the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X* and *Y* degrees of freedom on the element at the wall.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

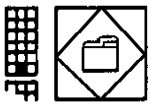
Chapter 9 of the BESTVIEW manual has an extensive description on specifying GPBEST boundary conditions in I-DEAS. It would be best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way that finite element boundary conditions are.* In particular, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 110 is used at the wall, since both *X* and *Y* displacements are zero there. Load set 101 is used at the tip of the beam, since the *X*-traction is non-zero there.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Task]	`[Meshing]` `change to [Boundary Conditions]`	First, define the `Load Sets` applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.
[Icons]	 `Sets ...`	Define the proper Load Set number.

[Set Management]	`Type: Load`
	`Set: 110`
	`[Create]`
	`[Make Current]`
	`[Dismiss]`
		Load set 110=zero displacement components in *X*- and *Y*-directions.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
		At this point, the proper Load set number has been defined. Now proceed to apply the fixed boundary condition at the wall.
[PROMPT]	**`/create add beam`** `<CR>`	This is an abbreviation for /Create Additional Beam Loads.
[DISPLAY]	`(click on left edge)`	Apply the boundary condition at the wall.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[DISPLAY]	`[Global/Beam Length]`	
[DISPLAY]	`[Constant]`	
[PROMPT]	`999 <CR>`	This is a dummy X-force. It should be non-zero.
[PROMPT]	`<CR> <CR> <CR> <CR> <CR> <CR>`	Just hit <CR> to accept default answers, which in this case is zero force and moment magnitude and vector color=orange.

At this point the fixed boundary condition at the left end of the beam (the wall) has been defined. Now proceed to apply the traction load to the right end of the beam (the tip). Begin by defining the proper Load Set number.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Sets ...`	Define the proper Load Set number. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.

[Set Management]	`Type: Load`
	`Set: 101`
	`[Create]`
	`[Make Current]`
	`[Dismiss]`
		Load set 101=non-zero *X*-traction.
		NOTE: Before typing in the Load Set number, be sure to erase whatever label appears on the "`Set`" line.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[PROMPT]	**`/create add beam`** `<CR>`	This is an abbreviation for /Create Additional Beam Loads.
[DISPLAY]	`(click on right edge)`	Apply the boundary condition at the tip of the beam.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[DISPLAY]	`[Global/Beam Length]`	
[DISPLAY]	`[Constant]`	
[PROMPT]	`100 <CR>`	Here, the traction magnitude is specified as 100 N/mm. Direction is specified using Set ID 101 (*X*-direction).
[PROMPT]	`<CR> <CR> <CR> <CR> <CR> <CR>`	Just hit <CR> to accept default answers, which in this case is zero force and moment magnitude and vector color=orange.

At this point, the boundary conditions and loads have been defined. Now you can exit from the IDEAS program.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File`	The database should be saved before the model is exported.
[MENU]	`Save`	
[MENU]	`File`	
[MENU]	`Export...`	
[Export Selections]	`I-DEAS Simulation Universal File`	
[Export Selections]	`[OK]`	
[Export Universal File]	`File Name: tut2c`	This creates an I-DEAS neutral file named `tut2c.unv`.
[Export Universal File]	`(select Fem1)`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Message]	`[OK]`	Create a new Universal File.
[Export Universal File]	`[OK]`	Watch for messages in the [List] window. They can help determine what was written to the Universal File.
[MENU]	`File`	
[MENU]	`Exit`	Exit from I-DEAS.
[Message]	`[YES]`	Save file before exiting.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 8: Run GPBEST To Analyze The Boundary Element Model

At this point, the I-DEAS universal file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`5 <CR>`	I-DEAS MS (R3-R6).
[PREBEST]	`tut2c.unv <CR>`	I-DEAS universal file name.
[PREBEST]	`tut2c.dat <CR>`	Write to `tut2c.dat` file (default).
[PREBEST]	`2 <CR>`	2D - Plane Stress.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[PREBEST]	`N <CR>`	Do not scale the data. It is already in the desired form (mm-Newton). See the note at the beginning of this section regarding the units in I-DEAS.
[PREBEST]	`1 <CR>`	Default thickness is 1 mm.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`N <CR>`	No body forces.
		The PREBEST program ends, and the file `tut2c.dat` has been created. It can be run directly in GPBEST.
		Note that it is sometimes useful to examine the `ideasbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining if the model was completely translated or not.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut2c.dat best-s > tut2c.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut2c.neu <CR>`**

Step 9: Post-Processing The Results

TIP: Check your GPBEST results file before post-processing. It does not do much good to view the results graphically if fatal errors were encountered in the GPBEST run.

You can view the GPBEST results file (`tut2c.res`) and determine if GPBEST ran successfully using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, then proceed to view the results in I-DEAS. For this step, it is best to start I-DEAS with a new database. The neutral file contains everything that is needed: the boundary element model and results information.

Since I-DEAS cannot read in the GPBEST neutral file directly, it is necessary to run the POSTBEST translator to create files that I-DEAS will recognize.

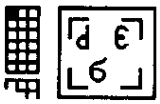
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**postbest**<CR>	
[POSTBEST]	5 <CR>	I-DEAS MS (R3-R6).
[POSTBEST]	tut2c.neu <CR>	Name of GPBEST neutral file.
[POSTBEST]	N <CR>	Do not renumber the nodes.
[POSTBEST]	N <CR>	Do not scale the data. It is already in the desired form (mm-Newton). See the note at the beginning of this section.
[UNIX]	**cp IDEAS.unv tut0.unv** <CR>	This universal file is created: IDEAS.unv It may be useful to browse the bestideas.log file for any error messages.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**ideas**	Start I-DEAS. This is merely a post-processing session.
[I-DEAS Start]	Project: john	
	Model File Name: tut0	
	Application: Simulation	
	Task: Post processing	
[I-DEAS Start]	[OK] [OK]	Create model database.
[MENU]	File	
[MENU]	Import...	
[Import Selections]	[I-DEAS Simulation Universal File]	
	[OK]	

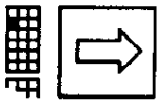
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[File Name Input]	`Universal file name: tut0.unv` `[OK]`	
[I-DEAS List]		While the universal file is being read in, notice the messages in this window. They provide some useful diagnostics in the event that something should go wrong.

Step 10: Create A Deformed Shape Plot

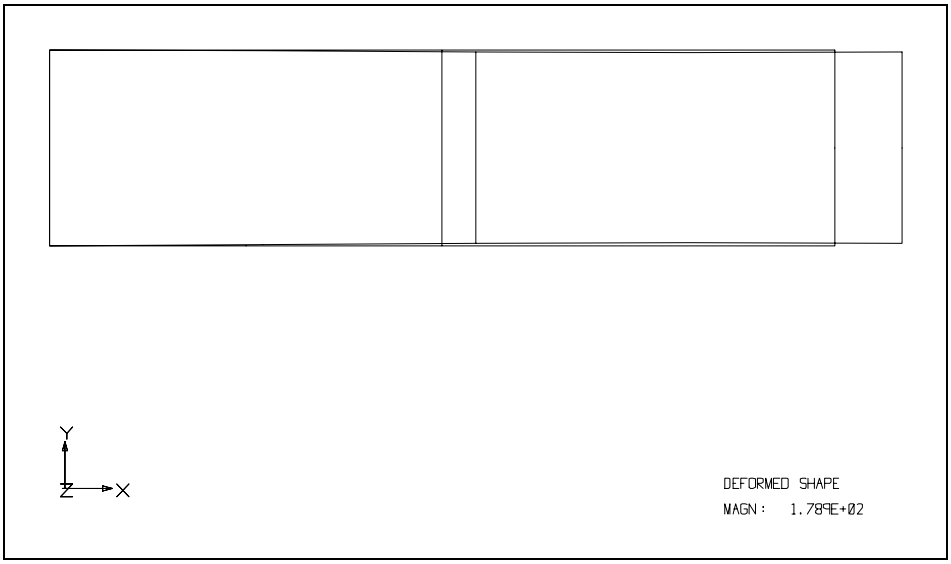
Plot the deformed beam along with the original (undeformed) geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Results...`	
[Results Selection]	`Display Results [Clear]`	This turns off color fringe plot.

[Results Selection]	`Deformation Results:`
	`1-NODAL DISPLACEMENTS`
	`Component: Magnitude`
		This already appears in the window.

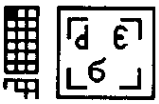
[Results Selection]	`[OK]`	
[Icons]	 `Display`	
[DISPLAY]	`Draw a box around the entire model. Then click Middle Mouse button to confirm this selection.`	Deformed shape plot will appear in the [DISPLAY].

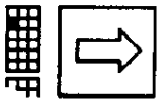
Deformed Geometry Plot



Step 11: Create A Color Fringe Stress Plot

The Von Mises stress plot is easily displayed by changing the Results Display. For this problem, the stress is uniform in the interior. Note however, that the interior stress is displayed even though there is really no mesh in the interior. This is because GPBEST automatically creates a temporary interior mesh for 2-D problems for interior post-processing purposes. The CONTOUR card in the **CASE data block triggers this step. The user does not need to create an interior mesh for post-processing purposes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Results...`	
[Results Display]	`Click mouse on [3-NODAL VON-MISES STRESS]`	This highlights the component of stress to use in the color fringe plot.
[Results Selection]	`Display Results: Click right arrow.`	This puts the stress component into the Display Results window.
[Results Selection]	`Deformation Results: [Clear]`	This turns off the deformed shape plot.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Results Selection]	`[OK]`	
[Icons]	 `Display`	
[DISPLAY]	`Draw a box around the entire model. Then click Middle Mouse button to confirm this selection.`	Color fringe plot of stress will appear in the [DISPLAY].
[MENU]	`File`	Proceed to exit out of I-DEAS.
[MENU]	`Exit`	
	`[NO]`	Do not save this database. It was only used for post-processing.

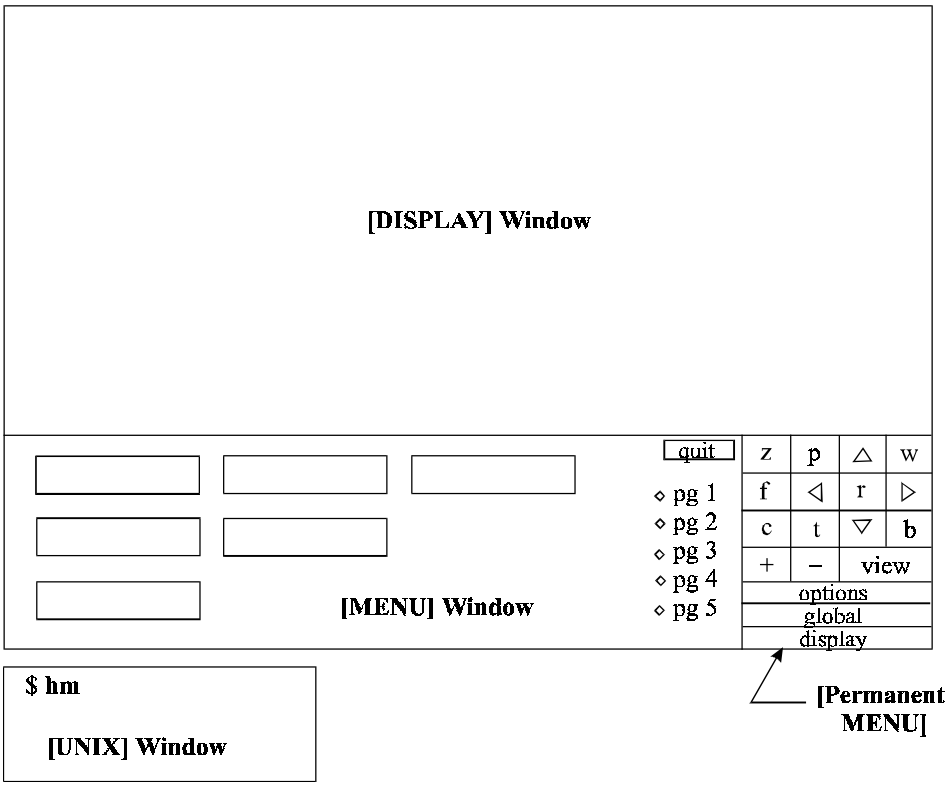
HYPERMESH Model

It is assumed that the HYPERMESH user is familiar with geometry creation, meshing, etc. in HYPERMESH. This tutorial is not intended to teach how to use HYPERMESH. It is intended to show an experienced HYPERMESH user how to interface with the GPBEST program. Some steps in the modeling process, such as boundary condition specification, require special attention. This tutorial will illustrate the special steps to follow to create a GPBEST boundary element model and view the results of a GPBEST analysis in HYPERMESH.

TIP: This tutorial covers HYPERMESH 2.1 or higher.

The HYPERMESH Window Names

The names of the HYPERMESH windows are given in the following figure:



Step 1: Start HYPERMESH

In this step, you will start the HYPERMESH program, name the database that will be used, and define a collector. It is probably easiest to define material properties right away.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**hm**	Start HYPERMESH.
[MENU]	◆ `pg 1`	Go to Page 1 of the Menus.
[MENU-pg 1]	`[files]`	

[Files]	`File= tut2d.hm`
	`[save]`
	`[return]`
		This will name the database `tut2d.hm`.

[Permanent Menu]	`[global]`	
[Global]	`Template file: …/hm21a/template/feoutput/hm/general`	
[Global]	`[return]`	

Step 2: Define Material Properties

[MENU-pg 1]	`[collectors]`	Hypermesh requires a collector to be defined. Its concept is similar to “groups” in BESTVIEW and GMRs in GPBEST.
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲ `[mats]`	Click on the input collector switch, ▲. Proceed to define the material properties.
[Collectors: Create]	`[name =] STEEL`	This is the name of the material.
[Collectors: Create]	`E = 207000.`	Young’s modulus $\left(\frac{N}{mm^2}\right)$.
	`Nu = 0.3`	
	`Rho = 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.

[Collectors: Create]	`[create]`	Material property is created.
[Collectors: Create]	`Collector type: ▲` `[comps]`	Click on the input collector switch, ▲. Define a component.
[Collectors: Create]	`[material =]`	
[Choose one material]	`[STEEL]`	The material is STEEL.
[Collectors: Create]	`[name =] beam1`	This is the name of the first (default) collector.
[Collectors: Create]	`[create]` `[return]`	The collector was created. Now the collector `beam1` is the current component.

Step 3: Define The Nodes of The Beam

Create nodes and elements directly without using geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU-pg 1]	`[create nodes]`	
[Create Nodes]	◆ `type in`	

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 0`
	`[z =] 0`
	`[create]`
		Accept (0,0,0) for first point.

[Create Nodes: Type In]	`[x =] 40`
	`[y =] 0`
	`[z =] 0`
	`[create]`
		Point 2.

[Create Nodes: Type In]	`[x =] 80`
	`[y =] 0`
	`[z =] 0`
	`[create]`
		Point 3.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Create Nodes: Type In]	`[x =] 80`
	`[y =] 10`
	`[z =] 0`
	`[create]`
		Point 4.

[Create Nodes: Type In]	`[x =] 80`
	`[y =] 20`
	`[z =] 0`
	`[create]`
		Point 5.

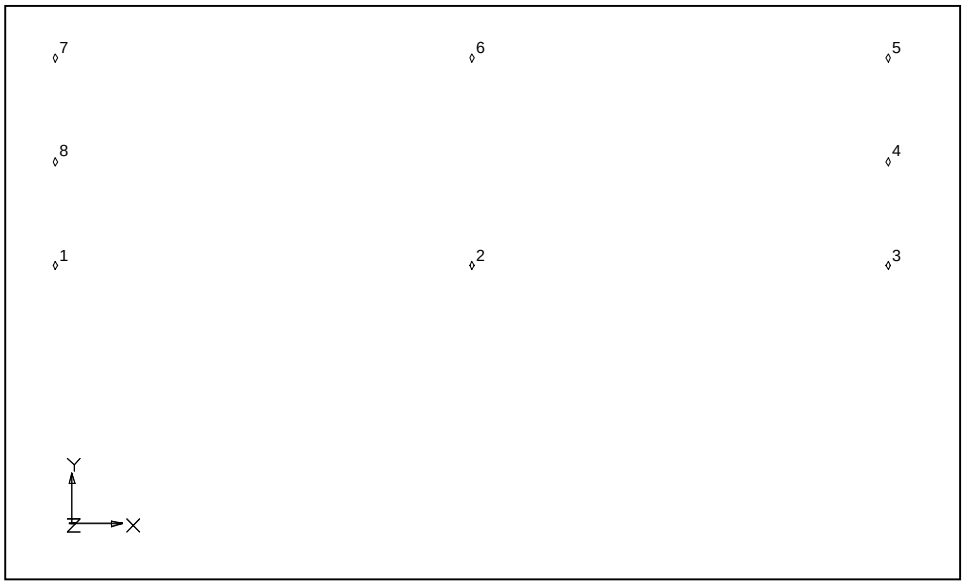
[Create Nodes: Type In]	`[x =] 40`
	`[y =] 20`
	`[z =] 0`
	`[create]`
		Point 6.

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 20`
	`[z =] 0`
	`[create]`
		Point 7.

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 10`
	`[z =] 0`
	`[create]`
		Point 8.

[Permanent MENU]	`[f]`	The four points should appear in the [DISPLAY] window.
[Create Nodes: Type In]	`[return]`	

Nodes Defining The Beam



Step 4: Define A GROUP

For this simple problem, it is not necessary to take the additional steps to define a GROUP in HYPERMESH. A default group has already been defined (beam1). In HYPERMESH, these are known as “collectors” or “components”. More complex models consisting of multiple regions require the use of multiple components. This will be examined in the following chapter.

Step 5: Create A Boundary Element Mesh

The boundary element mesh is now created. It consists of boundary elements and nodes. Only one boundary element will be created on each edge of the beam.

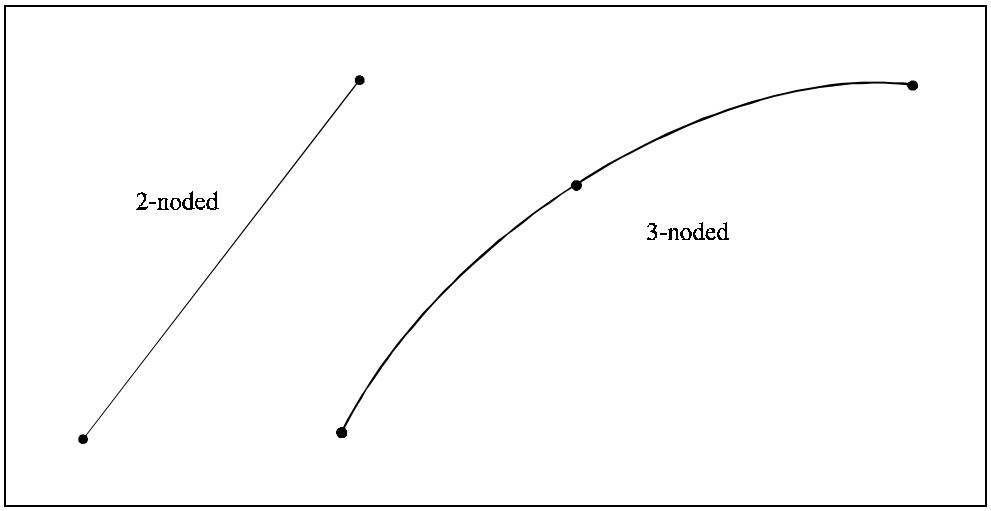
Normally, each boundary element will contain three nodes (two end nodes and one mid node), hence it is called a *quadratic* element.

Notice For Users of HYPERMESH

Although HYPERMESH does not support boundary elements directly, boundary elements are simulated using beam elements in 2-D. These beam elements do not require properties, such as area, orientation, etc., since they are not true beams. The translator step (PREBEST) later converts them into GPBEST boundary elements. HYPERMESH 2.1 supports quadratic beam finite elements. However, HYPERMESH 2.0 only supports linear

(2-noded) beam elements. The remainder of this chapter will assume the use of HYPERMESH 2.1.

2-D Boundary Elements



As illustrated in this figure, it is preferable to use 3-noded beam elements in GPBEST. HYPERMESH 2.1 users can use these. However, HYPERMESH 2.0 users must use linear (2-noded) beam elements.

Equivalencing and renumbering of nodes and elements are not necessary if a mesh is created in BESTVIEW. BESTVIEW automatically equivalences and renumbers entities. However, the user must include this extra step when building a model in HYPERMESH.

Note that an interior mesh is not needed; a mesh is only required on the boundary of the region. This is a tremendous modeling advantage inherent in the boundary element method.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU – pg 1]	◆ `pg 3`	Go to Page 3 of the Menus. This contains the bar element creation commands.
[MENU - pg 3]	`[bars]`	
[Bars]	◆ `bar3`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Bars: Create]	`(click on the leftmost ▲`	Click on the input collector switch, ▲. This allows you to choose a vector for specifying beam orientation. (This orientation is not used by GPBEST, but it is necessary to input it in HYPERMESH.)
[Bars: Create]	`[components]`	

[Bars: Create]	`[x comp =] 0`
	`[y comp =] 0`
	`[z comp =] 1`
		The [node A] button is now highlighted.

[DISPLAY]	`(click on Node 1)`	At this point, the [node B] button is now highlighted.
[DISPLAY]	`(click on Node 3)`	
[DISPLAY]	`(click on Node 2)`	Element 1 has been created. A label BAR3 is placed at the center of the element. The [node A] button is now highlighted.
[DISPLAY]	`(click on Node 3)`	At this point, the [node B] button is now highlighted.
[DISPLAY]	`(click on Node 5)`	
[DISPLAY]	`(click on Node 4)`	Element 2 has been created. A label BAR is placed at the center of the element. The [node A] button is now highlighted.
[DISPLAY]	`(click on Node 5)`	At this point, the [node B] button is now highlighted.
[DISPLAY]	`(click on Node 7)`	
[DISPLAY]	`(click on Node 6)`	Element 3 has been created. A label BAR is placed at the center of the element. The [node A] button is now highlighted.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[DISPLAY]	`(click on Node 7)`	At this point, the [node B] button is now highlighted.
[DISPLAY]	`(click on Node 1)`	
[DISPLAY]	`(click on Node 8)`	Element 4 has been created. A label BAR is placed at the center of the element. The [node A] button is now highlighted.
[Bars: Create]	`[return]`	
[MENU - pg 3]	`[edges]`	This is the menu that contains the equivalencing command.
[Edges]	`(click on [comps])`	
[Display]	◆ `beam1` `[select]`	Select component `beam1`.

[Edges]	`[preview equiv]`
	`[equivalence]`
	`[return]`
		Equivalence coincident nodes. In this example, there were no nodes to equivalence. In more complex models, some nodes will be removed.

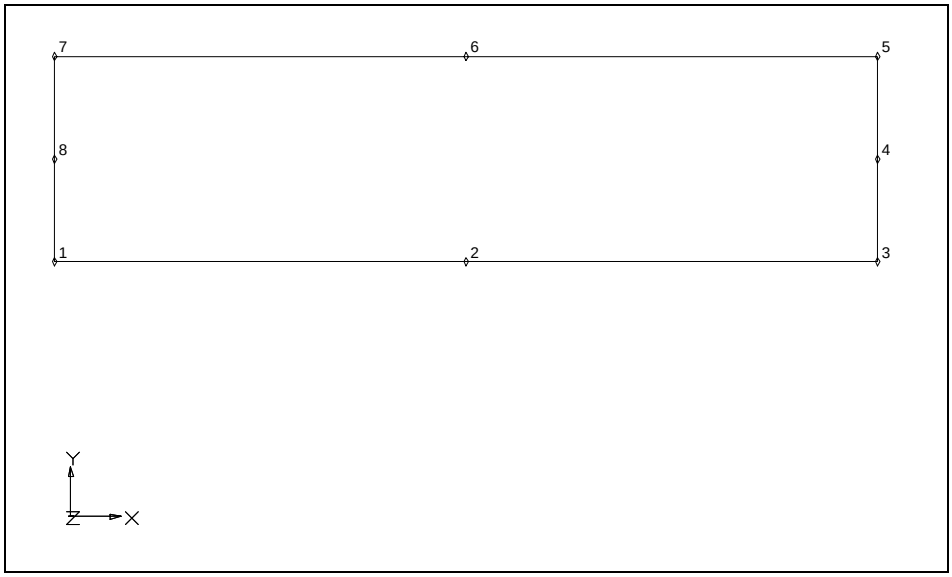
[MENU]	◆ `pg 4`	Go to Page 4 of the Menus. This contains the renumbering commands. Renumbering is not required. It is done just for user convenience.
[MENU - pg 4]	`[renumber]`	
[Renumber]	`(click on ▲)`	
[Renumber]	`[nodes]`	
[Renumber]	`(click on [nodes])`	Renumber nodes only.
[Renumber]	`(click on [all])`	This selects all nodes.
[Renumber]	`[return]`	

Step 6: Display Boundary Element Mesh

At this point, the model display in HYPERMESH can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Permanent MENU]	`[display]`	Proceed to erase the geometry items from the screen.
[Display]	`(click on ▲ to the left of “elems” or “geoms”)`	
[Display]	`[geoms]`	
[Display]	`[none]`	This turns off the display of geometry items (e.g., lines).
[Permanent Menu]	`[p]`	Only the BAR elements are left on the screen.
[Display]	`[return]`	
[Permanent Menu]	`[opts]`	
[Defines parameter…]	◆ `element handle` `[return]`	Click element handle OFF. The BAR3 labels disappear.
[MENU - pg 4]	`[numbers]`	
[Numbers]	`(click on ▲)`	
	`[nodes]`	Proceed to display node numbers.
[Numbers]	`(click on [nodes])`	
[Numbers]	`[all]`	Choose all nodes.
[Numbers]	`[on]`	Nodes are displayed, similar to the following figure.
[Numbers]	`[return]`	

Node Numbers



Step 7: Define The Boundary Conditions For The Model

> ***TIP****:* Note that GPBEST boundary conditions are not specified using the traditional FEM approach in HYPERMESH. While working in 2-D in HYPERMESH, the GPBEST user must use nodal displacements to specify both loads and boundary conditions. You must specify all nodes on an element to apply a boundary condition to the element in 2-D models.

Now the boundary conditions can be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X* and *Y* degrees of freedom on element 4.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual has an extensive description on specifying GPBEST boundary conditions in HYPERMESH. It would be best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, note that the boundary conditions and loads on 2-D models are specified with specific displacement set IDs. For example, displacement set 110 is used at the wall, since both *X* and *Y* displacements are zero there. Displacement set 101 is used at the tip of the beam, since the *X*-traction is non-zero there.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 4]	`[collectors]`	First, define the displacement cases applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate displacement set numbers.
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲ `[loadcols]`	Click on the input collector switch, ▲
[Collectors: Create]	`[name =] 110`	Displacement set 110=zero displacement components in *X*- and *Y*- directions at the left edge of the beam (WALL).
[Collectors: Create]	`[create]`	
[Collectors: Create]	`[return]`	At this point, Displacement set 110 has been created. In HYPERMESH terminology, it is a “Load Collector” named “110”.
[MENU - pg 4]	`[constraints]`	Proceed to apply the boundary condition.
[Constraints]	◆ `create`	
[Constraints: Create]	`(click on [nodes])`	
[Constraints: Create]	`[by id]`	
[DISPLAY]	`(click on the three nodes on the left edge of the beam - nodes 1, 7, 8)`	In 2-D models, it is necessary to select *all* nodes on a given element where you want to apply boundary conditions. If only one node is selected, it will be ignored by the PREBEST translator. At this point, nodes 1, 7, 8 have been selected.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Constraints: Create]	`[create]`	In [DISPLAY] window, nodes 1, 7, 8 are shown as constrained.
[Constraints: Create]	`[return]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 4]	`[collectors]`	Now work on the traction at the tip of the beam. First, create a constraint collector 101.
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲ `[loadcols]`	Click on the input collector switch, ▲
[Collectors: Create]	`[name =] 101`	Displacement set 101=non-zero traction.
[Collectors: Create]	`[create]`	
[Collectors: Create]	`[return]`	At this point, Displacement set 110 has been created. In HYPERMESH terminology, it is a “Load Collector” named “110”.
[MENU - pg 4]	`[constraints]`	Proceed to apply the traction boundary condition.
[Constraints]	◆ `create`	
[Constraints: Create]	`(click on [nodes])`	
[Constraints: Create]	`[by id]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[DISPLAY]	`(click on the three nodes on the right edge of the beam – 3, 4, 5)`	In 2-D, it is necessary to select *all* nodes on a given element where you want to apply boundary conditions. If only one node is selected, it will be ignored by the PREBEST translator. At this point, nodes 3, 4, 5 have been selected.
[Constraints: Create]	◆ `dof1 = 100.`	Only the dof1 value needs to be specified. Here, the traction magnitude is specified as 100 N/mm. Direction is specified by the Load set ID 101.
[Constraints: Create]	`[create]`	In [DISPLAY] window, nodes 3, 4, 5 are shown as constrained.
[Constraints: Create]	`[return]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 4]	◆ `pg 5`	
[MENU - pg 5]	`[export data]`	
[Export Data]	`template = /... /feoutput/hm/general` `filename = tut2d.hma`	This creates a HYPERMESH ASCII neutral file named `tut2d.hma`.
[Export Data]	`[write]`	
[Export Data]	`[return]`	
[MENU - pg 5]	`[files]`	
[Files]	`file = tut2d.hm`	
[Files]	`[save]`	
[Files]	`[yes]`	OK to overwrite.
[Files]	`[return]`	
[MENU - pg 5]	`[quit]`	Exit from HYPERMESH.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 8: Run GPBEST To Analyze The Boundary Element Model

At this point, the HYPERMESH (ASCII) neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`6 <CR>`	HYPERMESH.
[PREBEST]	`tut2d.hma <CR>`	HYPERMESH ASCII file name.
[PREBEST]	`Y <CR>`	Yes - this is the correct file.
[PREBEST]	`tut2d.dat<CR>`	Write to `tut2d.dat` file (default).
[PREBEST]	`2 <CR>`	2D - Plane Stress.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`1 <CR>`	Default thickness is 1 mm.
[PREBEST]	`2 <CR>`	Use quadratic variation for elements.
[PREBEST]	`N <CR>`	No body forces.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
		The PREBEST program ends, and the file `tut2d.dat` has been created. It can be run directly in GPBEST. Note that it is sometimes useful to examine the `hypbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining if the model was completely translated or not.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut2d.dat best-s > tut2d.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut2d.neu`**

Step 9: Post-Processing The Results

> ***TIP:*** Check your GPBEST results file before post-processing. It does not do much good to view the results graphically if fatal errors were encountered in the GPBEST run.

You can view the GPBEST results file (`tut2d.res`) and determine if GPBEST ran successfully using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, then proceed to view the results in HYPERMESH. For this step, it is best to start HYPERMESH with a new database.

Since HYPERMESH cannot read in the GPBEST neutral file directly, it is necessary to run the POSTBEST translator to create files that HYPERMESH will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`postbest`**`<CR>`	
[POSTBEST]	`6 <CR>`	HYPERMESH.
[POSTBEST]	`tut2d.neu <CR>`	Name of GPBEST neutral file.
[POSTBEST]	`N <CR>`	Do not renumber the nodes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

		Three files are created:
		`HYPER.GEOM`
		`PATRAN.dis.1`
		`PATRAN.nod.1`

> ***TIP:*** To find out what options are available when running the `hmpatran` command, just execute:
> `hmpatran -u`
> Depending on the problem size, etc., you may have to change the "size" or "upper" arguments.

At this point, an intermediate step must be carried out. This involves translating the patran results files into a HYPERMESH result file. HYPERMESH cannot read PATRAN result files directly.

The command used to translate the PATRAN results into a HYPERMESH result file is `hmpatran`. It comes with the HYPERMESH installation, and is described in Chapter 13 of the HYPERMESH Reference Manual. Note that the PATRAN results files must be named as they are created by the POSTBEST program. Do not rename them or delete them until `hmpatran` has been executed. After `hmpatran` has created a HYPERMESH result file, the PATRAN result files can be deleted.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`hmpatran PATRAN tut2dd.res`** `<CR>`	
		The HYPERMESH binary results file, `tut2dd.res`, has been created. It contains displacements, element results, and nodal results.
[UNIX]	**`rm PATRAN.*`**	Once the HYPERMESH result file has been successfully created, the PATRAN result files can be deleted.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`hm`**	Start HYPERMESH.
[MENU]	◆ `pg 5`	Go to Page 5 of the Menus.
[MENU - pg 5]	`[import data]`	Proceed to import model geometry.

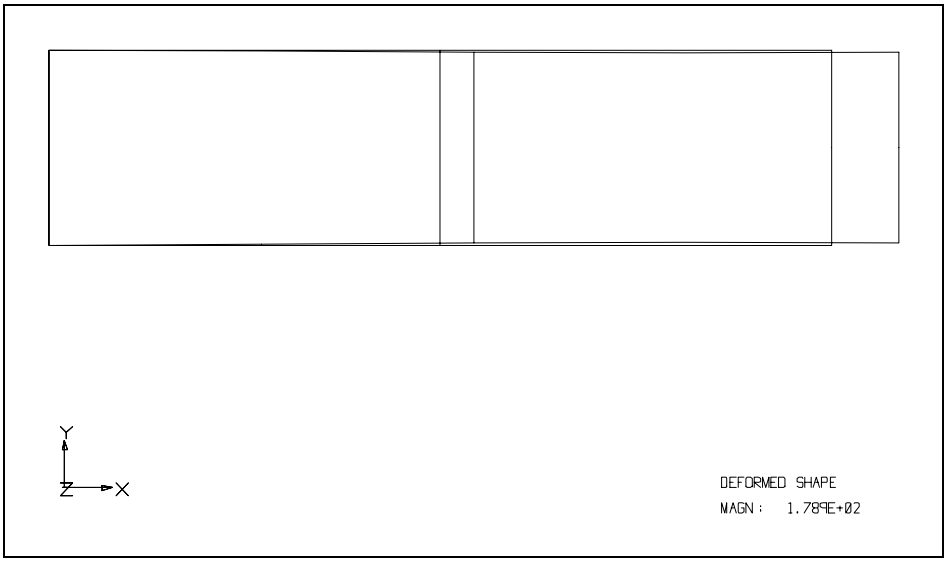
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Import Data]	`[translator = ] hm/feinput/hmascii`	
[Import Data]	`[filename = ] HYPER.GEOM`	
[Import Data]	`[import]`	
[Import Data]	`[return]`	
[Permanent MENU]	`[global]`	Proceed to import results.
[Global]	`[template file = ] hm/template/feoutput/hm /general`	
[Global]	`[results file = ] tut2dd.res`	
[Global]	`[return]`	

Step 10: Create A Deformed Shape Plot

Plot the deformed beam along with the original (undeformed) geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 5]	`[deformed]`	
[Deformed Shape]	`[data type = ]`	
[Select Item]	`(click on [Displacements])`	
[Deformed Shape]	`[deform]`	
[Deformed Shape]	`[return]`	

Deformed Geometry Plot



Step 11: Create A Color Fringe Stress Plot

The Von Mises stress plot is easily displayed by changing the Results Display. For this problem, the stress is uniform in the interior. Note however, that the interior stress is displayed even though there really is no mesh in the interior. This is because GPBEST automatically creates a temporary interior mesh for 2-D problems for interior post-processing purposes. The CONTOUR card in the **CASE data block triggers this step. The user does not have to create an interior mesh for post-processing purposes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 5]	`[contour]`	
[Contour]	`[data type = ]`	
[Select Item]	`(click on [Nodal Results Column 11])`	This is the column for Von Mises stress.
[Contour]	`[contour]`	
[Contour]	`[return]`	
[MENU - pg 5]	`[quit]`	Proceed to exit HYPERMESH
[Message]	`[Yes]`	Do not save this file.

A Note Regarding HYPERMESH 2.0

HYPERMESH 2.0 does not support 3-noded beam elements. If your GPBEST data file contains 3-noded boundary elements (in a 2-D analysis), the results will still display corectly. The reason for this is that GPBEST creates an internal mesh for post-processing purposes when the `CONTOUR` card is included in `**CASE`. These internal elements are brought into HYPERMESH properly. For post-processing purposes, the limitation of 2-noded beam elements does not present a problem if the `CONTOUR` card is used.

HYPERMESH 2.1 can import 3-noded beam elements directly.

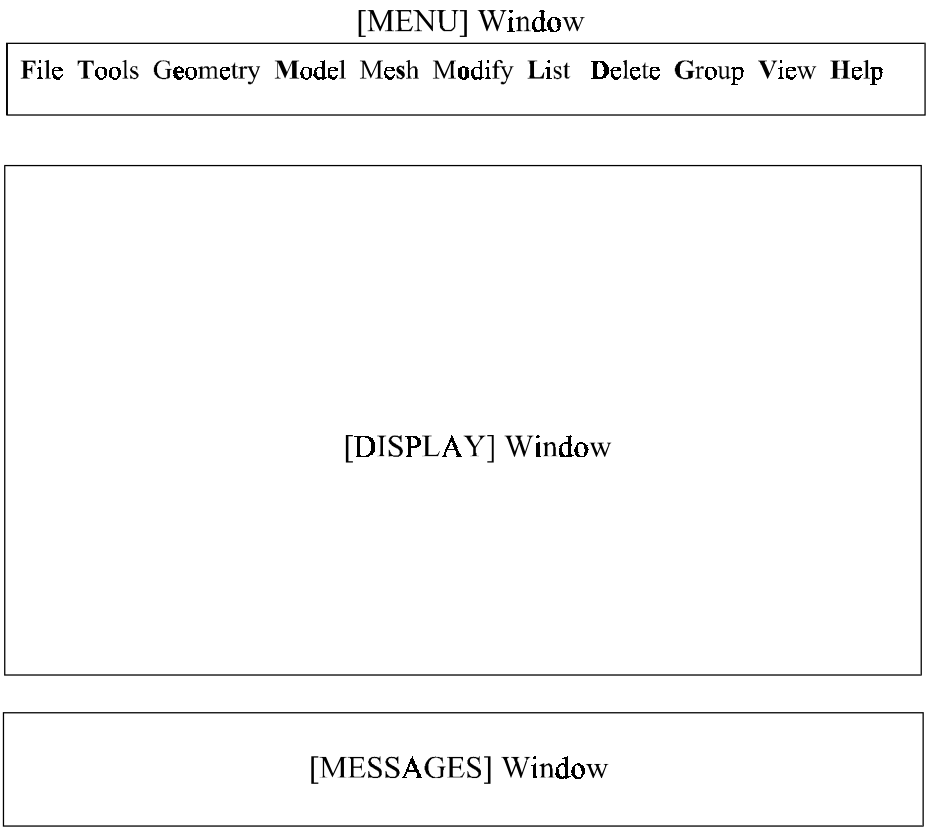
FEMAP Model

It is assumed that the FEMAP user is familiar with geometry creation, meshing, etc. in FEMAP. This tutorial is not intended to teach how to use FEMAP. It is intended to show an experienced FEMAP user how to interface with the GPBEST program. Some steps in the modeling process, such as boundary condition specification, require special attention. This tutorial will illustrate the special steps to follow to create a GPBEST boundary element model and view the results of a GPBEST analysis in FEMAP.

TIP: This tutorial covers FEMAP 5.0 or higher.

The names of the FEMAP windows are given in the following figure:

The FEMAP Window Names



Step 1: Start FEMAP

In this step, you will start the FEMAP program and name the database that will be used.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Windows]	**Femap**	Start FEMAP.
[MENU]	File	
	Save As:…	
[File Save As]	tut2e [SAVE]	This will name the database tut2e.mod.

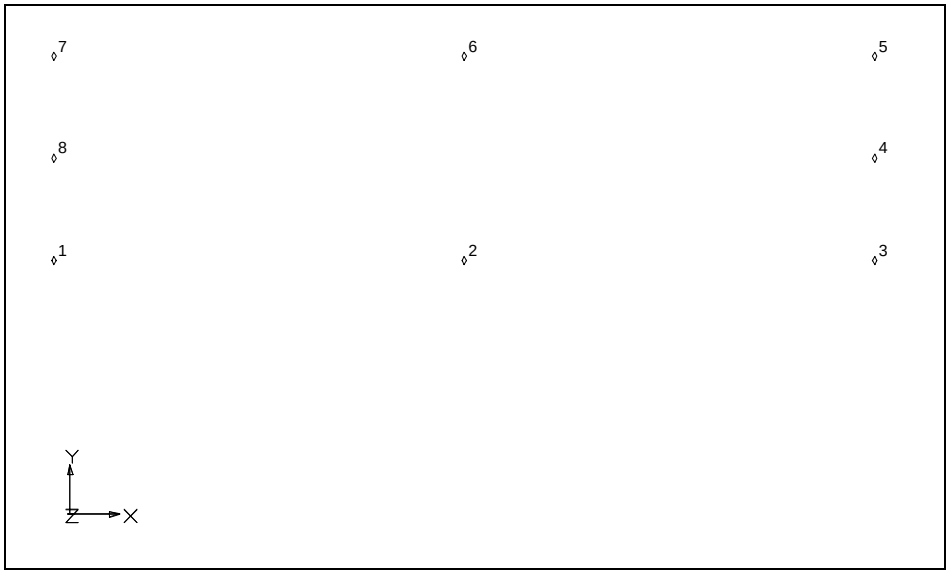
Step 2: Define The Nodes

Nodes and elements can be generated in FEMAP. However, for this simple problem, they will be defined manually

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	Model	
	Node	
[Point]	[Methods^]	
[Point]	Locate	
[Locate]	*x*=0	Point 1.
	y=0	
	z=0 [OK]	
[Locate]	*x*=40	Point 2.
	y=0	
	z=0 [OK]	
[Locate]	*x*=80	Point 3.
	y=0	
	z=0 [OK]	
[Locate]	*x*=80	Point 4.
	y=10	
	z=0 [OK]	
[Locate]	*x*=80	Point 5.
	y=20	
	z=0 [OK]	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Locate]	*x*=40 *y*=20 *z*=0 [OK]	Point 6.
[Locate]	*x*=0 *y*=20 *z*=0 [OK]	Point 7.
[Locate]	*x*=0 *y*=10 *z*=0 [OK] [Cancel]	Point 8.
[MENU]	View Options	
[View Options]	[Quick Options]	
[View Quick Options]	[Labels On] [Done]	
[View Options]	[OK]	
[MENU]	View AutoScale	The nodes should appear in the [DISPLAY] window.

Nodes



Step 3: Offset the Mid-Nodes

The mid-nodes are moved 1-unit in the Z-direction. The reason for this is explained in Step 6.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Modify`	
	`Move By`	
	`Node`	
[Entity Selection – Select Nodes]	`From [2]To [8] By [2]`	Nodes 2, 4, 6, 8.
	`[OK]`	
[Vector Locate]	`Base: (0,0,0)`	The nodes have been moved.
	`Tip: (0,0,1)`	
	`[OK]`	

Step 4: Define A GROUP

For this simple problem, it is not necessary to define a GROUP in FEMAP. More complex models consisting of multiple regions require the use of GROUPS. This will be examined in the following chapter.

Step 5: Define Material Properties

Proceed to define material properties for STEEL. Also, an element property must be defined in order to generate elements in Step 6.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Model`	
	`Material`	
[Define Isotropic Material]	`ID: 1`	
	`Title: STEEL`	
[Define Isotropic Material]	`Young's Modulus:207000.`	Young's modulus $\left(\frac{N}{mm^2}\right)$.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Define Isotropic Material]	`Poisson Ratio: 0.3`	
[Define Isotropic Material]	`Mass Density: 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.
[Define Isotropic Material]	`[OK]` `[Cancel]`	The material has been defined.
[MENU]	`Model` `Property`	Proceed to define a dummy property. FEMAP will not create elements without a property definition.
[Define Property]	`[Elem/Property Type…]`	
[Element/Property Type]	● `Beam` `[OK]`	

[Define Property – Beam Element Type]	`ID: 1`
	`Title: Dummy Beam`
	`Material: [1..STEEL]`	

[Define Property – Beam Element Type]	`[OK]` `[Cancel]`	Beam property has been defined.

Step 6: Create A Boundary Element Mesh

The boundary element mesh is now created. It consists of boundary elements and nodes. Only one boundary element will be created on each edge of the beam. Each boundary element will contain three nodes (two end nodes and one mid node), hence it is called a *quadratic* element.

Notice For Users of FEMAP

Although FEMAP does not support boundary elements directly, boundary elements are simulated using beam elements in 2-D. These beam elements do not require properties, such as area, orientation, etc., since they are not true beams. (However, a dummy property is required to *create* beam elements in FEMAP.) The translator step (PREBEST) later converts them into GPBEST

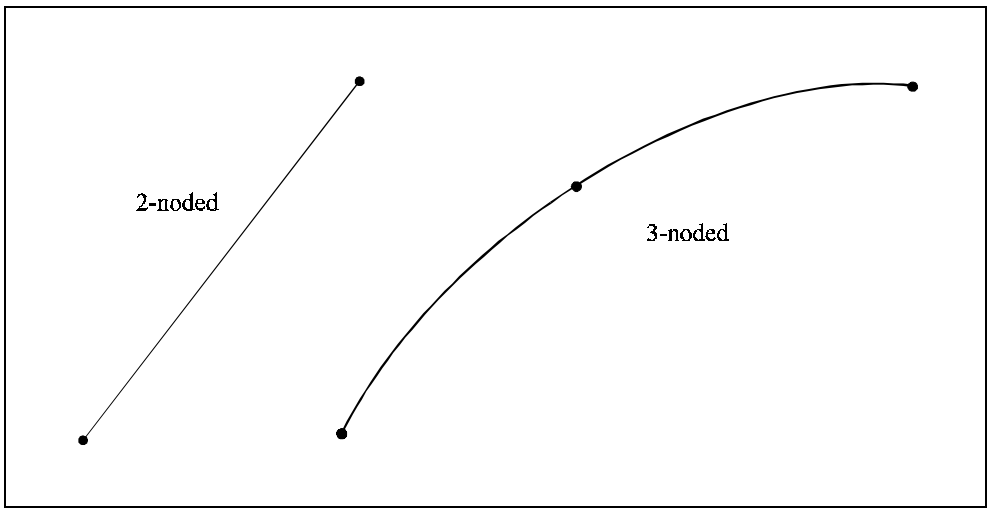
boundary elements. However, FEMAP only supports linear (2-noded) beam elements at this time.

The user is thus presented with two options: 1) Just use 2-noded (linear) beams to model boundary elements, or 2) Use the 2-noded beams but use the “Orientation node” as a mid-node. The latter approach is used in this tutorial. However, when specifying an orientation node, it cannot be colinear with the beam element. That is why, in **Step 3**, the mid-nodes were displaced in the Z-direction. After the elements have been created, they can be moved back into place. If the mid-node is not colinear with the element, i.e., if you want a curved element, then the mid-node is already offset and this additional step does not need to be taken.

When reading in a FEMAP neutral file containing beam elements from POSTBEST, the mid-nodes can be coincident. FEMAP only checks for a coincident orientation node when creating new elements.

This is merely a work-around for the fact that FEMAP does not support three-noded beam elements. In the future, it is hoped that FEMAP will support three-noded beam elements, and this work-around will not be necessary.

2-D Boundary Elements (mid-node specified as an “orientation node”)



Equivalencing and renumbering of nodes and elements are not necessary if a mesh is created in BESTVIEW. BESTVIEW automatically equivalences and renumbers entities. The user must include this extra step when building a model in FEMAP.

Note that an interior mesh is not needed; a mesh is only required on the boundary of the region. This is a tremendous modeling advantage inherent in the boundary element method.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Model` `Element`	Proceed to create the four beam elements.

[Define Beam Element]	`Property=1..Dummy Beam`
	`ID = 1`
	`Nodes 1 & 3`
	`Orientation Node = 2`
	`[OK]`
		Mid-node is handled through the “Orientation Node.”
		Also, nodes can also be picked with the mouse cursor.

[Define Beam Element]	`Property=1..Dummy Beam`
	`ID = 2`
	`Nodes 3 & 5`
	`Orientation Node = 4`
	`[OK]`	

[Define Beam Element]	`Property=1..Dummy Beam`
	`ID = 3`
	`Nodes 5 & 7`
	`Orientation Node = 6`
	`[OK]`	

[Define Beam Element]	`Property=1..Dummy Beam`
	`ID = 4`
	`Nodes 7 & 1`
	`Orientation Node = 8`
	`[OK] [Cancel]`	

[MENU]	`Modify`
	`Move By`
	`Node`
		Move mid-nodes back onto X-Y plane.

[Entity Selection – Select Nodes to Move]	`From [2] to [8] by [2]` `[OK]`	Nodes 2, 4, 6, 8.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Vector Locate]	`[Methods^] [Components]`
	`Base: (0,0,0)`
	`Tip: (0,0,-1)`
	`[OK]`	

[MENU]	`Tools`
	`Check`
	`Coincident Nodes`
		Equivalence coincident nodes.

[Entity Selection – Select Nodes to Check]	`[Select All] [OK]`	
[FEMAP]	`[No]`	Do not specify additional range of nodes to merge.

[Check/Merge Coincident]	`Maximum Distance: 0.005`
	`] Merge Coincident Entities`
	`[OK]`	

[MENU]	`Modify`
	`Renumber`
	`Node`
		Renumbering is not required. It is merely done for the user's convenience.

[Entity Selection – Select Nodes to Renumber]	`[Select All]` `[OK] [OK]`	

[MENU]	`Modify`
	`Renumber`
	`Element`	

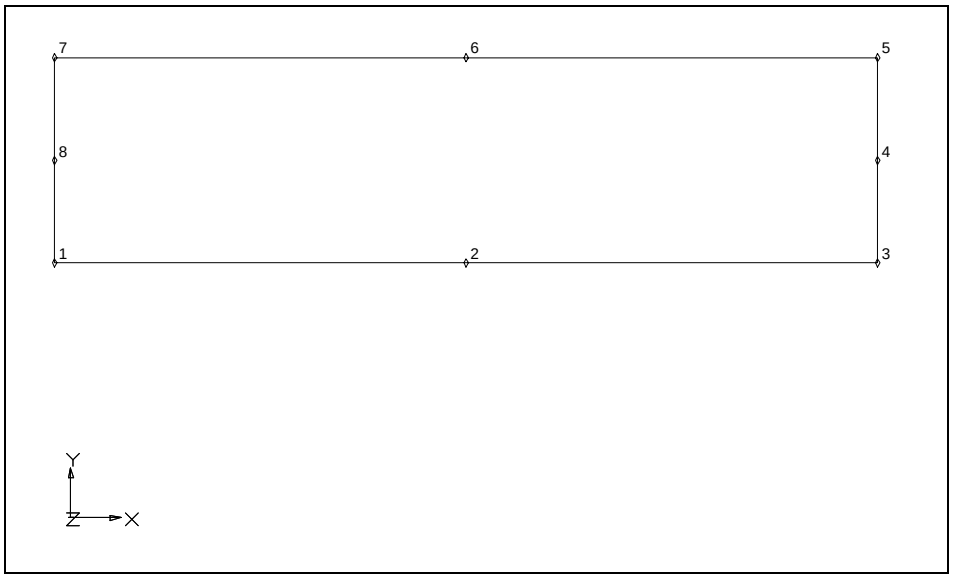
[Entity Selection – Select Elements to Renumber]	`[Select All]` `[OK] [OK]`	

Step 7: Display Boundary Element Mesh

At this point, the model display in FEMAP can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`View`	
	`Options`	
	`[Quick Options]`	
[View Quick Options]	`[All Entities Off]`	The points and lines no longer need to be displayed.
	`[Labels On]`	
[View Quick Options]	] `Node`	
	] `Element`	
	] `Load – Distributed Load`	
	] `Load – Pressure`	
[View Quick Options]	`[Done]`	The model, along with node number labels, appears in the [DISPLAY] window.
[View Options]	`[OK]`	

Nodes and Elements



Step 8: Define The Boundary Conditions For The Model

> ***TIP**:* Note that GPBEST boundary conditions are not specified using the traditional FEM approach in FEMAP. While working with 2-D models in FEMAP, the GPBEST user must use distributed element loads to specify both loads and boundary conditions.

Now the boundary conditions can be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X* and *Y* degrees of freedom on element 4.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual has an extensive description on specifying GPBEST boundary conditions in FEMAP. It would be best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, in 2-D, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 110 is used at the wall, since both *X* and *Y* displacements are zero there. Load set 101 is used at the tip of the beam, since the *X*-traction is non-zero there. Also, distributed element loads are used to specify all 2-D loads and boundary conditions.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Model`
	`Load`
	`Set`
		First, define the load case for the left edge (WALL) of the model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.

[Create or Activate Load Set]	`ID: 110`
	`Title: WALL`
	`[OK]`
		This creates Load case 110 AND makes it the current load set.

[MENU]	`Model`
	`Load`
	`Elemental`	

[Entity Selection – Enter Elements to Select]	`ID: 4` `[OK]`	You could also select element 4 by clicking on it.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Create Loads on Elements]	`[Distributed Load]`
	`Load-End A: 999.`
	`Load-End B: 999.`
	`[OK]`
		These are just dummy loads. In reality, Load set 110 is a constraint. FEMAP will not permit you to specify a zero-magnitude load.

[Distributed Load Direction]	● `Global X` `[OK]`	Always use Global-X, no matter what direction the load is oriented. Load direction is specified by the Load Set ID.
[Entity Selection – Enter Elements to Select]	`[Cancel]`	The “loads” should appear at the WALL.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Model`
	`Load`
	`Set`
		Now work on the TIP load.
		Load Set 101=non-zero traction.

[Create or Activate Load Set]	`ID: 101`
	`Title: TIP`
	`[OK]`
		This creates Load case 101 AND makes it the current load set.

[MENU]	`Model`
	`Load`
	`Elemental`	

[Entity Selection – Enter Elements to Select]	`ID: 2` `[OK]`	You could also select element 2 by clicking on it.

[Create Loads on Elements]	`[Distributed Load]`
	`Load-End A: 100.`
	`Load-End B: 100.`
	`[OK]`
		Only one value is needed inside the brackets. Here, the traction magnitude is specified as 100 N/mm. Direction is specified by the Load set ID 101.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Distributed Load Direction]	● `Global X` `[OK]`	Always use Global-X, no matter what direction the load is oriented. Load direction is specified by the Load Set ID.
[Entity Selection – Enter Elements to Select]	`[Cancel]`	The “loads” should appear at the TIP.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File` `Save`	

[MENU]	`File`
	`Export`
	`FEMAP Neutral`	

[Write Model To FEMAP Neutral]	`File Name: tut2e.fem` `[Write]`	This creates a FEMAP neutral file named `tut2e.fem`.
[Neutral File Write Options]	] `Write Analysis Model` ● `Full Model` `[OK]`	Other options may also be checked in addition to these.
[MENU]	`File` `Exit`	Save file before exiting.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User’s Manual.

Step 9: Run GPBEST To Analyze The Boundary Element Model

At this point, the FEMAP neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`7 <CR>`	FEMAP.
[PREBEST]	`tut2e.fem <CR>`	FEMAP neutral file name.
[PREBEST]	`tut2e.dat<CR>`	Write to `tut2e.dat` file (default).
[PREBEST]	`2 <CR>`	2D - Plane Stress.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`1 <CR>`	Default thickness is 1 mm.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`N <CR>`	No body forces.
		The PREBEST program ends, and the file `tut2e.dat` has been created. It can be run directly in GPBEST.
		Note that it is sometimes useful to examine the `fembest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining if the model was completely translated or not.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut2e.dat best-s > tut2e.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut2e.neu`**

Step 10: Post-Processing The Results

> ***TIP:*** Check your GPBEST results file before post-processing. It does not do much good to view the results graphically if fatal errors were encountered in the GPBEST run.

You can view the GPBEST results file (`tut2e.res`) and determine if GPBEST ran successfully using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, then proceed to view the results in FEMAP. For this step, it is best to start FEMAP with a new database.

Since FEMAP cannot read in the GPBEST neutral file directly, it is necessary to run the POSTBEST translator to create files that FEMAP will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**postbest**<CR>	
[POSTBEST]	7 <CR>	FEMAP.
[POSTBEST]	tut2e.neu <CR>	Name of GPBEST neutral file.
[POSTBEST]	N <CR>	Do not renumber the nodes.
		Three files are created: FEMAP.GEOM patran.dis.1 patran.nod.1

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Windows]	**femap**	Start up the FEMAP program. FEMAP starts up with a new, blank database.
[MENU]	File Import FEMAP Neutral	Import model.
[Read Model From FEMAP Neutral]	File name: FEMAP.GEOM [Open]	
[MENU]	File Import Analysis Results	Import results.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Import Results From]	● `Patran` `[OK]`	
[FEMAP]	`[YES]`	OK to Read Displacement Results File?
[Read Output From PATRAN Neutral]	`File name: patran.dis.1` `[Open]`	
[FEMAP]	`[YES]`	OK to Begin Reading File `patran.dis.1`?
[FEMAP]	`[NO]`	OK to Read Element Results File? (There are no element results for this problem.)
[FEMAP]	`[YES]`	OK to Read Nodal Results File?
[Read Output From PATRAN Neutral]	`File name: patran.nod.1` `[Open]`	
[FEMAP]	`[YES]`	OK to Begin Reading File `patran.nod.1` ?
[FEMAP]	`[NO]`	OK to Read Additional Files?
[MENU]	`View` `Options`	Turn off labels for post-processing.
[View Options]	`[Quick Options]`	
[View Quick Options]	`[Labels Off]` `[Done]`	
[View Options]	`[OK]`	

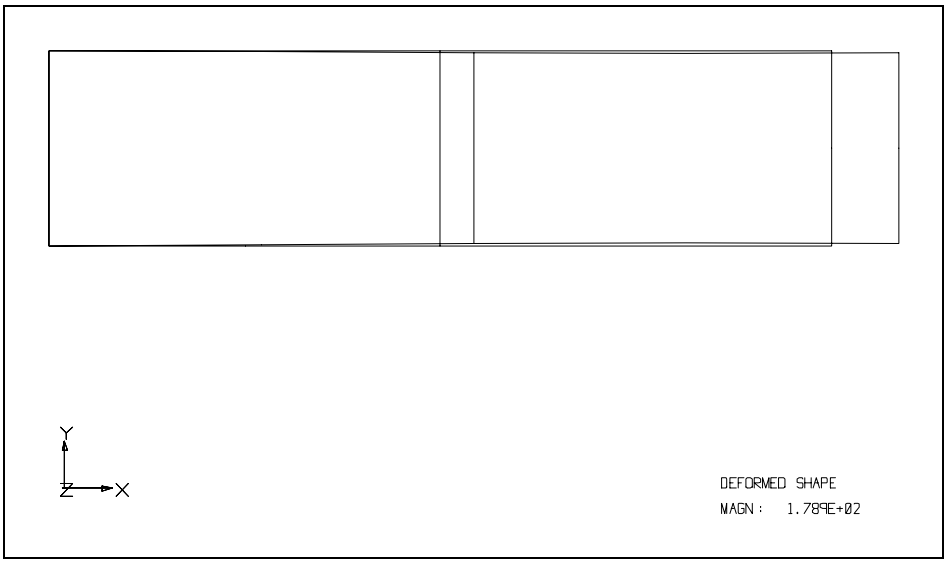
Step 11: Create A Deformed Shape Plot

Plot the deformed beam along with the original (undeformed) geometry.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`View` `Select`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[View Select]	● `Deform` `[Deformed and Contour Data]`	
[Select PostProcessing Data]	`[OK]`	
[View Select]	`[OK]`	

Deformed Geometry Plot



Step 12: Create A Color Fringe Stress Plot

The Von Mises stress plot is easily displayed by changing the Results Display. For this problem, the stress is uniform in the interior. Note however, that the interior stress is displayed even though there really is no mesh in the interior. This is because GPBEST automatically creates a temporary interior mesh for 2-D problems for interior post-processing purposes. The CONTOUR card in the **CASE data block triggers this step. The user does not have to create an interior mesh for post-processing purposes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`View`	
	`Select`	
[View Select]	● `Contour`	
	`[Deformed and Contour Data]`	
[Select PostProcessing Data]	`Contour: (select PATRAN Node Output 11)`	
	`[OK]`	
[View Select]	`[OK]`	
[MENU]	`File`	
	`Exit`	
[FEMAP]	`[No]`	Proceed to exit FEMAP without saving this model.

A Note Regarding FEMAP 5.0

FEMAP 5.0 does not support 3-noded beam elements. If your GPBEST data file contains 3-noded boundary elements (in a 2-D analysis), the results will still display corectly. The reason for this is that GPBEST creates an internal mesh for post-processing purposes when the `CONTOUR` card is included in `**CASE`. These internal elements are brought into FEMAP properly. For post-processing purposes, the limitation of 2-noded beam elements does not present a problem if the `CONTOUR` card is used.

See the note in **Step 6** for an explanation for the work-around for 3-noded beam elements.

CHAPTER 5 - TUTORIAL #3: 3-D PROBLEM

Introduction

The previous two chapters contained information on how to get started running GPBEST and BESTVIEW, along with how to build a simple 2-D boundary element model. This chapter parallels much of Chapter 4; except this chapter extends work in two areas: 1) a 3-D model is constructed, and 2) a two-GMR model is used.

Before starting this tutorial, it is important to understand exactly what GMRs are, when and why they are used, and how they interface with each other. The following section will give an overview of the GMR concept.

The GMR Concept

Overview

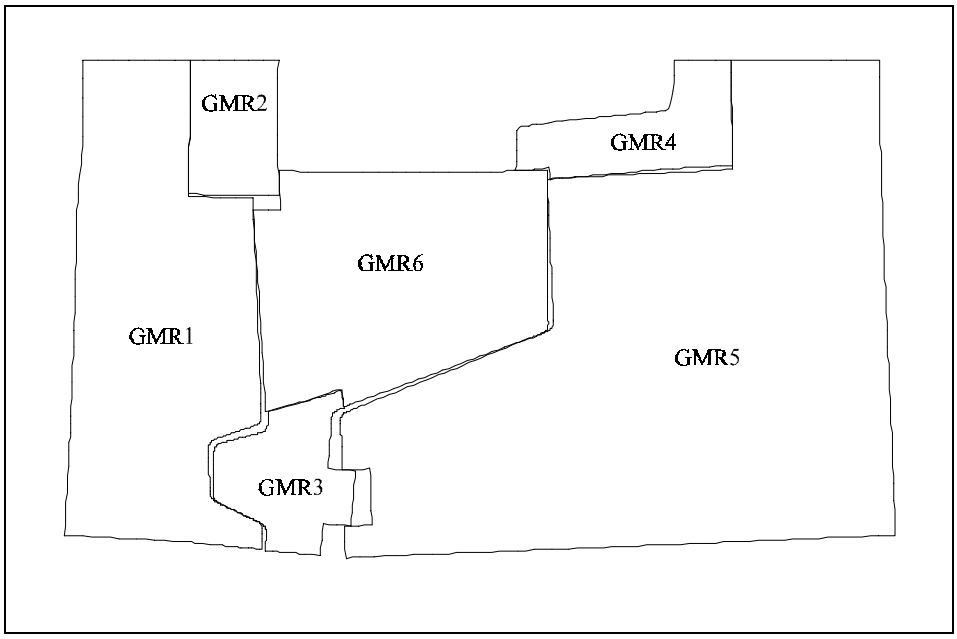
***TIP**:* See Section 5.3 of the GPBEST User's Manual for more detail about GMRs.

GMRs (Geometric Modeling Regions) are analogous to "substructures" in finite elements. However, their usage in BEM is not identical to FEM. Basically, their purpose is to allow the user to divide up a complete structure into component parts.

A single GMR is a complete boundary element model, complete with material properties, nodes, elements and boundary conditions. The 2-D beam in Chapter 4 is an example of a single-GMR model. However, for larger problems, it is more desirable to build multi-GMR models. Whenever multiple GMRs exist, there must be interface conditions specified between adjacent GMRs.

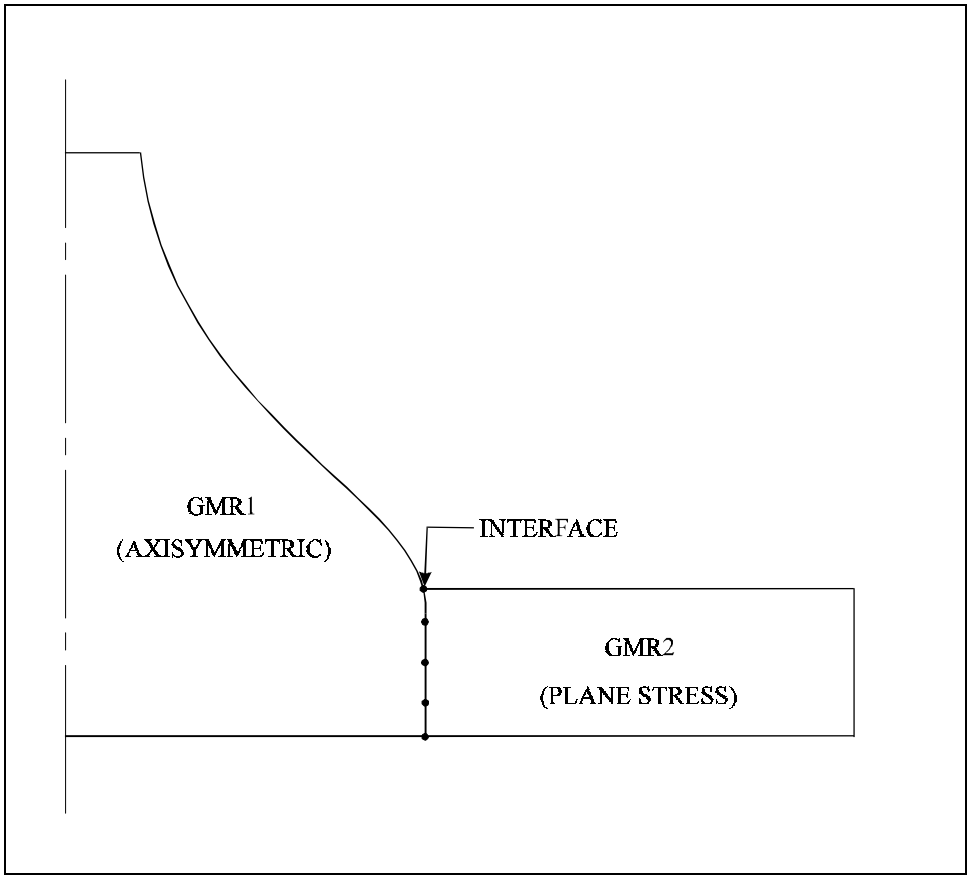
A six-GMR 2-D model is shown in the following figure. This problem involves a gap and contact analysis, where each GMR is free to move relative to adjacent GMRs. In order to conduct gap and contact analysis, a multi-GMR model must be used.

6-GMR 2-D Model



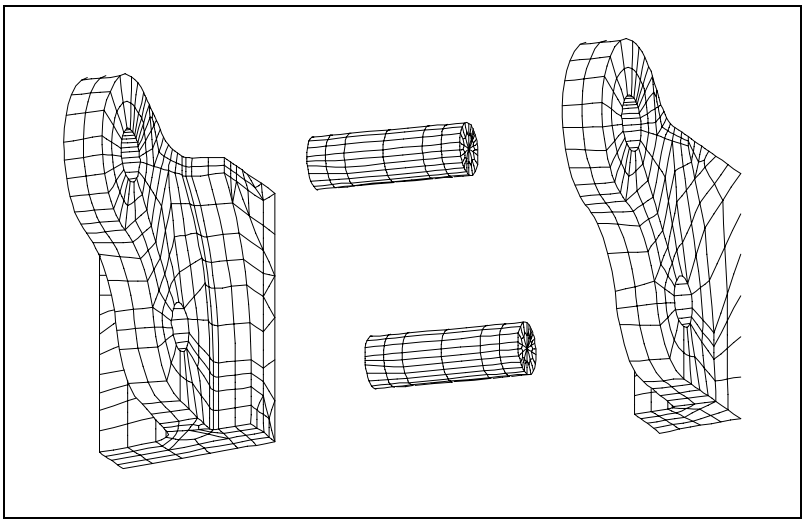
The following figure illustrates another type of multi-GMR model. In this problem, an axisymmetric region is in contact with a plane stress region. This is called a “mixed region problem.” The “interface” is located at the surface between the two regions.

2-GMR Mixed Model



A multi-GMR model of a 3-D component is illustrated in the following figure. This is a four-GMR model, where each portion of the bracket is modeled separately, and then combined to yield the complete model.

4-GMR 3-D Model Of A Bracket (Exploded View)



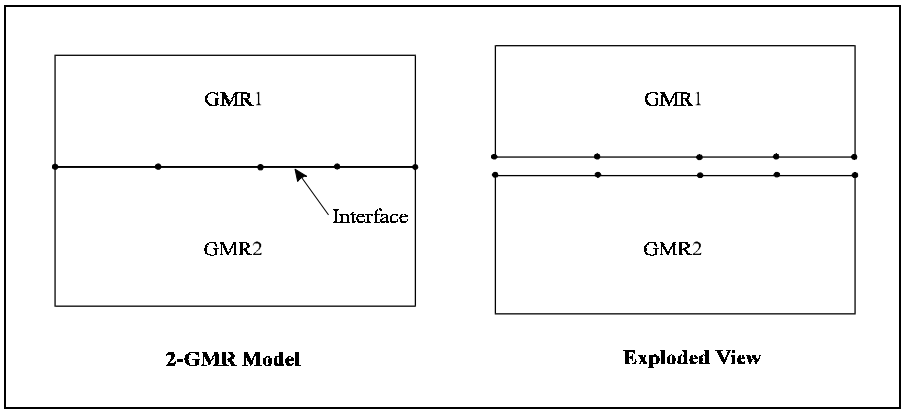
Characteristics Of GMRs

Section 5.3 of the GPBEST User's Manual lists all the requirements for GMR definitions. However, there are some basic characteristics of GMRs which can be listed here:

1) Each GMR is a self-contained region, complete with material properties, nodes, elements, surfaces, etc. This allows several persons to model various components of an overall structure. Only special attention must be paid to how the GMRs interface with each other.
2) Boundary conditions and loads are also specified on a GMR by GMR basis.
3) In linear analysis, each GMR must contain only one material property. If a component with several materials is modeled, each material must go into their own GMR.
4) Wherever two GMRs connect, the elements and nodes at the connecting surface constitute an interface. The behavior of the elements and nodes at the interface is specified by an "interface condition" (`**INTE`).
5) In the current release of GPBEST, the nodes and elements across the interface must line up.

The modeling details at the interface are illustrated in the following figure. It is apparent that each GMR consists of a complete boundary element model. However, at the interface, the nodes and elements overlap. The nodes from each GMR line up with each other at the interface. This is known as the "interface condition."

Two-GMR 2-D Model Illustrating The Interface Condition



Reasons For Using Multi-GMR Models

***TIP**:* See Section 5.3 of the GPBEST User's Manual for more detail about GMRs. The various reasons for using GMRs are explained in greater detail there.

Although it may be tempting to think of GMRs as being similar to substructures in FEM, the analogy is not perfectly valid. There are several reasons for using GMRs.

1) *Modeling Considerations.* As problems become quite large and complex, it is desirable to break up the large problem into various GMRs where each GMR represents a logical section of the component.

2) *GPBEST Limits.* As discussed at the end of Chapter 3, GPBEST has limits on the size of each GMR. Even if the entire problem could be modeled using a single GMR, it may run into a limit on the number of nodes (or number of elements) per GMR. This may necessitate dividing the model up into several GMRs.

3) *Efficiency.* A large problem that is broken down into several GMRs runs more efficiently that the corresponding one-GMR model. The integration and solution segments of GPBEST benefit greatly from a multi-GMR approach. The execution time and disk space requirements are both affected.

4) *Multiple Materials.* If a problem consists of multiple materials, then each material would logically go into its own GMR. For linear analysis, each GMR must contain only one material.

5) *2-D Problems With Regions Of Different Thickness.* 2-D planar problems can have regions of different thickness (in the *Z*-direction). Each GMR must have only one thickness, so multiple thicknesses must be handled with multiple GMRs.

6) *Interface Conditions.* A variety of interface conditions are permitted between GMRs. These will be enumerated in the next section.

Types Of Interfaces

> ***TIP**:* See Section 5.4 of the GPBEST User's Manual for more detail about interfaces. The various types of interfaces are given there. Also, many examples are included to help the user determine which type of interface condition is appropriate for their particular problem.

Wherever two GMRs meet, an interface condition must be specified. The interface determines how the two GMRs behave relative to each other. Single-GMR models do not have an interface condition.

The various types of interfaces available in GPBEST are listed here:

1) *Bonded Contact.* Continuity of all displacement components is imposed across the interface. The two GMRs behave as if they are "welded" together. If you are using multiple GMRs purely for efficiency considerations, then the interfaces must be bonded.
2) *Sliding Contact.* Continuity is required only for the component of displacement normal to the interface. The surfaces can slide relative to each other.
3) *Interference Fit.* This allows an imposed displacement between the GMRs at the interface.
4) *Nonlinear Contact.* This allows two GMRs to move relative to each other. If the interface is in compression, then the GMRs are considered to be in contact. If the interface is in tension, then the two GMR surfaces can pull apart, creating a gap between them.
5) *Spring.* A spring-type interface can be imposed between two GMRs.
6) *Cyclic Symmetry Contact.* This applies to the cyclic symmetry option in GPBEST. See the GPBEST User's Manual for more details.

GMR Considerations In Modeling Programs

Each modeling program requires GMRs to be handled somewhat differently. It is probably best to follow along in each tutorial and note how GMRs are defined. Typically, GMRs are defined through *groups*. Interfaces are defined at the time the GPBEST data file is being created, either in BESTVIEW or in the PREBEST translator.

***TIP**:* Do not equivalence overlapping nodes at interfaces. Duplicate nodes must remain as such in order to be translated properly into interfaces.

Usually, each GMR is modeled separately. For example, the group containing GMR1 could be stored away while GMR2 is being modeled, and so forth. After each GMR has been modeled and defined, they can all be merged together into a complete database. When this is carried out, there will be duplicate nodes and elements at the interfaces. The translator will be able to handle the overlapping entities at the interfaces.

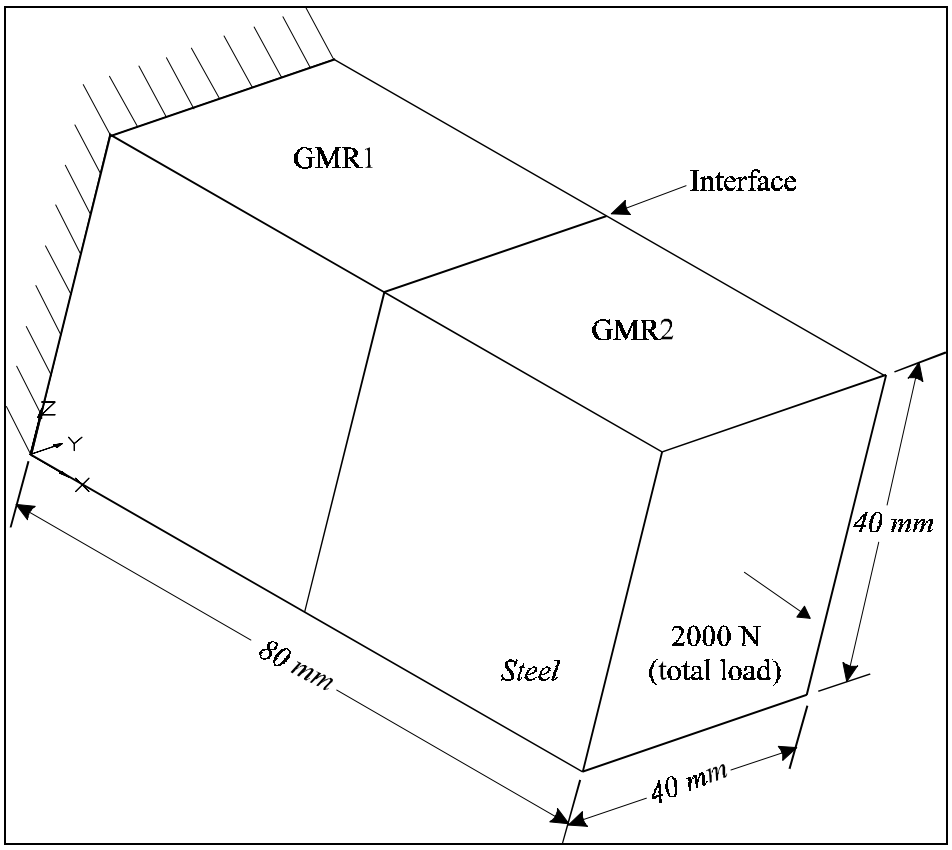
Problem Specification

It Is Estimated That It Will Take About One Hour To Go Through The Tutorial For Each Modeling Program In This Chapter

A 3-D cantilever beam is shown in the figure below. It is fixed at the wall and it is free to move at the free end (tip). A distributed static force of 2000 N (total) acts in the outward (*+X*) direction at the tip. The object is to determine the stress and deformation state of the beam under the given load.

Although this example is not a “large” model, it illustrates how multi-GMRs are handled. The problem is very similar to the beam in Chapter 4, but a 3-D surface model utilizing two GMRs is built. In particular, special modeling considerations at the interface are highlighted.

Beam Geometry



For the sake of checking the GPBEST results, it is useful to note the following beam and geometry calculations.

Analytical Solution

Maximum deflection at the tip of the beam (approximate):

$$\delta_{tip} = \frac{FL}{AE} = \frac{(2000N)(80mm)}{(40mm \times 40mm)(207000\,N/mm^2)} = 4.8309 \times 10^{-4}\,mm$$

Stress:

$$\sigma_{xx} = \frac{F}{A} = \frac{2000N}{40mm \times 40mm} = 1.25\,N/mm^2, \quad (everywhere)$$

Surface area:

$$A = (80 \times 40 \times 4) + (40 \times 40 \times 2) = 16000mm^2$$

Volume:

$$V = 80 \times 40 \times 40 = 128000mm^3$$

BESTVIEW Model

This BESTVIEW tutorial will parallel the 2-D problem in Chapter 4. However, where special differences arise due to the 3-D problem or multi-GMR nature, attention will be drawn to these.

Step 1: Start BESTVIEW

In this step, you will start the BESTVIEW program and name the database you will be using.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[ROOT]	**bestview**	Start BESTVIEW.
[PROMPT]	<ESC> **tut3a** [OK]	Name of new database.
[MENU]	SET-DIGITIZER	Turn DIGITIZER mode OFF.
[PROMPT]	[OFF]	

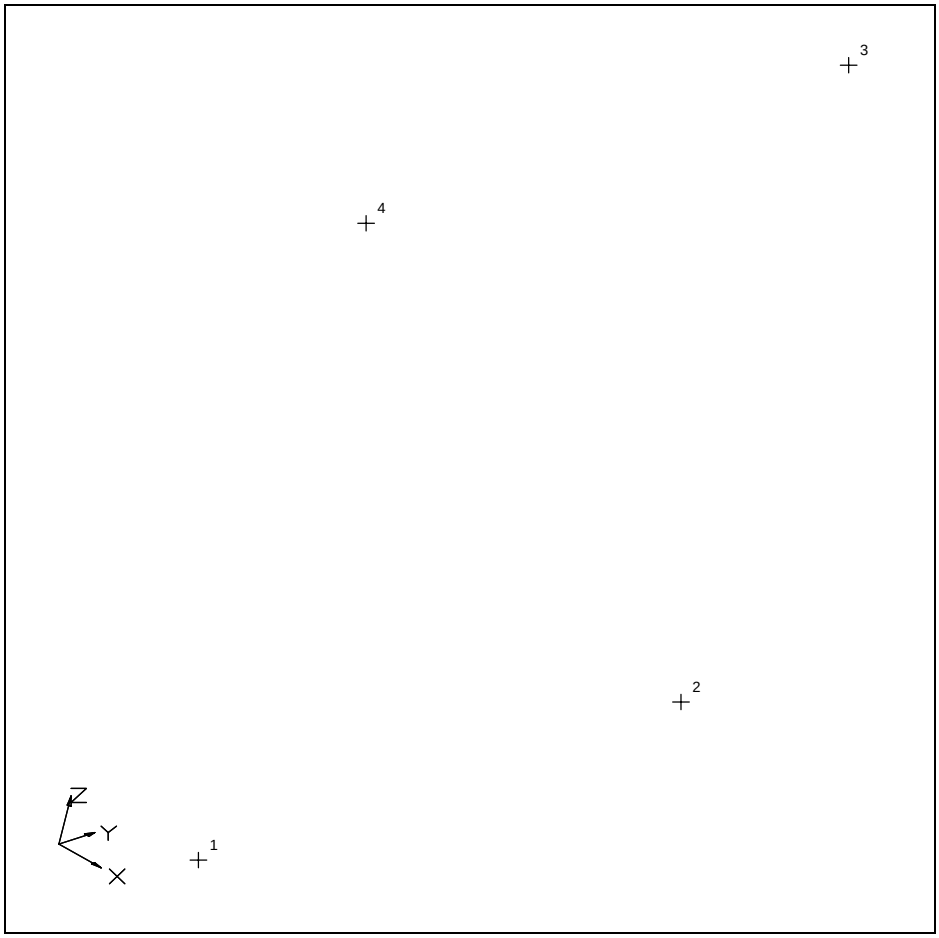
Step 2: Define Four Points

The method of creating this beam model will be to create the first GMR, then create the second GMR. To create the first GMR, we will define four corner points, define a surface from them, translate the surface to create a second surface, and define the remaining surfaces through the corner points.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	DEFINE-GEOMETRY	
[MENU]	POINT	
[MENU]	ENTER	
[PROMPT]	<ESC> 1 [OK]	First POINT number is 1.
[PROMPT]	0,0,0 <CR>	Point 1.
[PROMPT]	0,40,0 <CR>	Point 2.
[PROMPT]	0,40,40 <CR>	Point 3.
[PROMPT]	0,0,40 <CR>	Point 4. This defines the four corner points of the beam at the wall.
[PROMPT]	<ESC>	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`DEFI.POIN` `UP`	Move up one menu.
[MENU]	`VIEW-OPTIONS`	Look for a better viewing angle of the model.
[MENU]	`ROTATE`	
[PROMPT]	`-60,-30,-30 [OK]`	This is usually a good isometric viewing angle for 3-D models.
[PROMPT]	`[OK]`	
[MENU]	`CDRAW`	At this point, the [DISPLAY] window should appear as shown in the following figure.
[MENU]	`GLOB.VIEW` `UP`	Move up one menu.

Geometric Points Defining The Corners Of The End Section Of The Beam

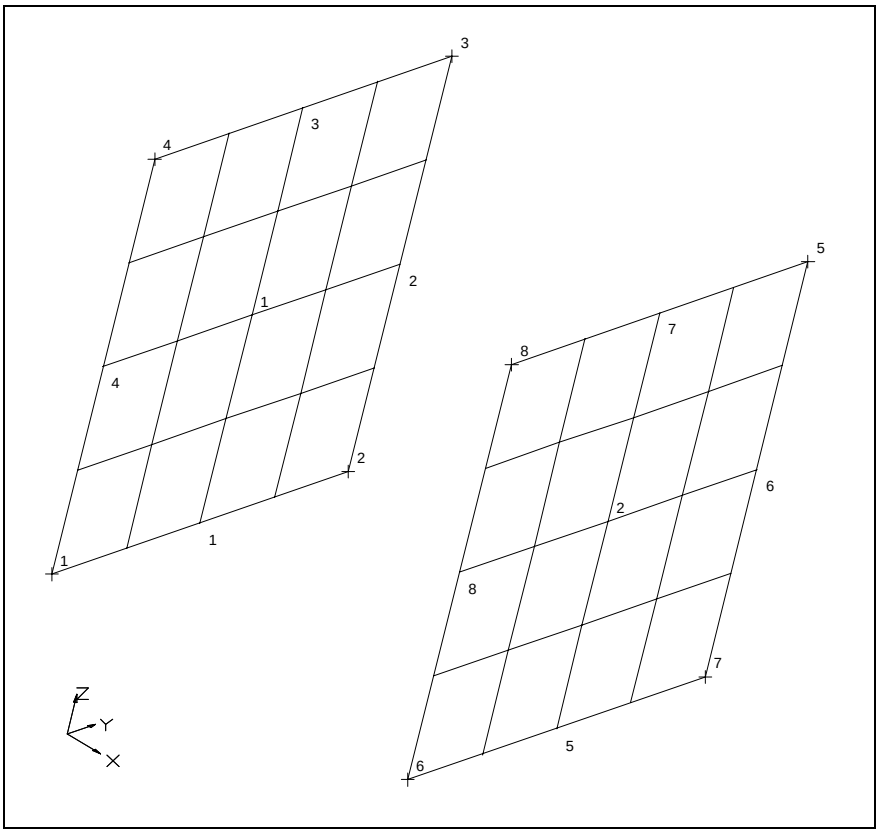


Step 3: Define The Geometric Surfaces

The four points defined in the previous step can be used to construct a geometric surface. From this surface, a second surface can be constructed through a translation. Once these two surfaces are defined along with their corner points, the remaining four surfaces of the cube for GMR1 can be defined via corner points.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`SURFACE`	You are still in the DEFINE-GEOMETRY menu.
[MENU]	`ENTER`	
[PROMPT]	`<ESC> 1 [OK]`	First SURFACE number is 1.
[PROMPT]	`1,2,3,4 <CR>`	Surface 1, at the “wall.” Surface 1 appears in [DISPLAY] window.
[PROMPT]	`<ESC>`	End input.
[MENU]	`TRANSLATE`	
[PROMPT]	`<ESC> 2 [OK]`	The next surface number is 2.
[PROMPT]	`<ESC> 1 [OK]`	Translate surface number 1 to create surface number 2.
[PROMPT]	`40,0,0 [OK] <ESC> <ESC>`	Translate 40 mm in *X*-direction.
[MENU]	`CDRAW`	The [DISPLAY] window should appear as in the following figure.

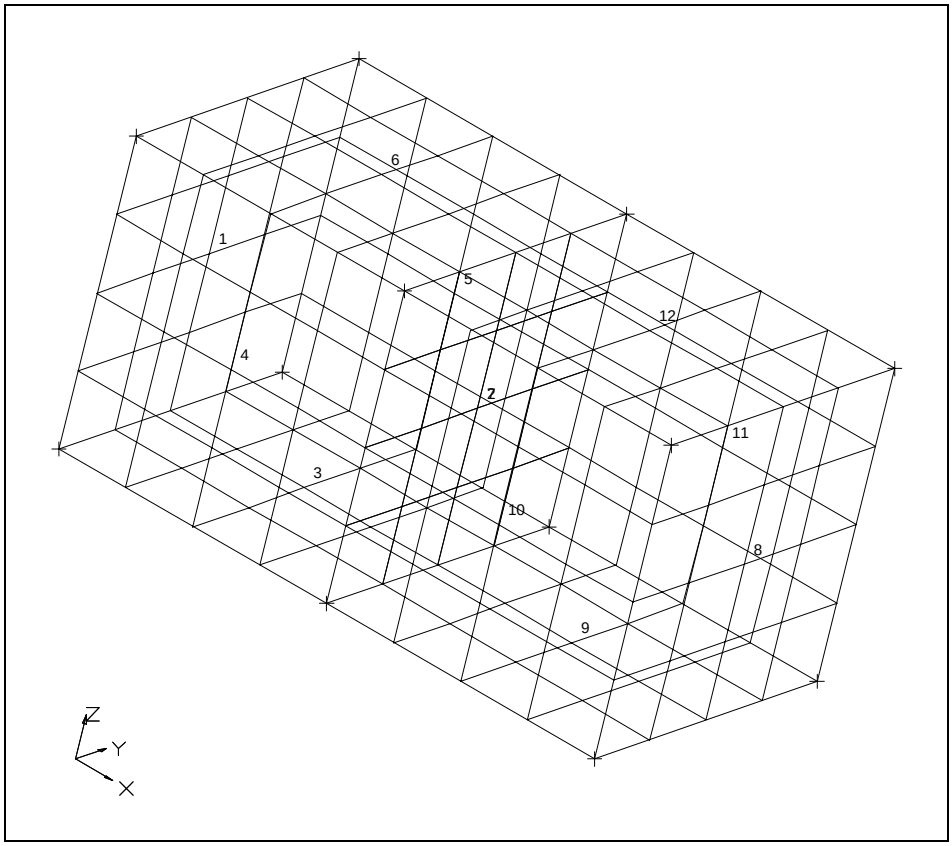
Geometric Points, Lines And Surfaces



INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	ENTER	Proceed to define four remaining surfaces using corner points.
[PROMPT]	<ESC> 3 [OK]	
[PROMPT]	1,6,7,2 [OK]	Surface 3.
[PROMPT]	1,6,8,4 [OK]	Surface 4.
[PROMPT]	2,7,5,3 [OK]	Surface 5.
[PROMPT]	4,8,5,3 [OK]	Surface 6.
[PROMPT]	<ESC>	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	TRANSLATE	Proceed to create six new surfaces by translating surfaces 1-6. This will define GMR2 and will complete the geometry modeling of the beam.
[PROMPT]	<ESC> 7 [OK]	The next surface number is 7.
[PROMPT]	<ESC> 1,6 [OK]	
[PROMPT]	40,0,0 [OK]	
[PROMPT]	<ESC> <ESC>	
[MENU]	CDRAW	Two cubes appear in the [DISPLAY] window.
[MENU]	DELETE	Surface 7 must be deleted since it is coincident with surface 2. BESTVIEW is able to handle the interface at surface 2.
[PROMPT]	<ESC> 7 [OK]	Select surface 7.
[PROMPT]	[YES] [YES]	
[PROMPT]	<ESC> <ESC>	
[MENU]	CDRAW	
[MENU]	DEFI.SURF UP	
[MENU]	DEFI UP	

Surfaces On The Faces Of The Two Cubes (Surface Numbers Are Shown)



Step 4: Define The Groups

In this model, we are using two groups, or GMRs, to model the beam. We must place the appropriate surfaces in each group. Group GMR1 will contain surfaces 1-6. GMR2 will contain surfaces 2,8-12. As mentioned previously, BESTVIEW does not require two separate surfaces at the interface.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`GROUPS`	
[MENU]	`ADD ENTITY`	
[PROMPT]	`GMR1 [OK]`	Name of GMR.
[MENU]	`SURFACE`	
[PROMPT]	`<ESC> 1,6 [OK]`	
[PROMPT]	`[YES]`	Save current changes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[PROMPT]	<ESC> PINK [OK]	Color of group. Color choices are displayed in the [ROOT] window.
[MENU]	ADD ENTITY	Proceed to define group GMR2.
[PROMPT]	GMR2 [OK]	Name of GMR.
[MENU]	SURFACE	
[PROMPT]	<ESC> 2 [OK]	
[PROMPT]	[YES]	Save current changes.
[PROMPT]	<ESC> CYAN [OK]	Color of group.
[MENU]	ADD ENTITY	
[PROMPT]	GMR2 [OK]	Name of GMR.
[MENU]	SURFACE	
[PROMPT]	<ESC> 8,12 [OK]	
[PROMPT]	[YES]	Save current changes.
[PROMPT]	[OK]	Keep CYAN as color of group GMR2.
[MENU]	GLOB.GROU UP	Move up one menu.
[MENU]	CDRAW	GMR1 is displayed in PINK, GMR2 is displayed in CYAN.

Step 5: Create A Boundary Element Mesh

The groups GMR1 and GMR2 created in the previous step are now used to generate the boundary element mesh. The boundary element mesh consists of boundary elements and nodes. Notice that elements are only placed on the surface of the beam. No interior mesh is required. This simplifies the meshing of complex 3-D models considerably.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	GENERATE MESH	
[MENU]	FAST-MESH	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[ROOT]	`0.005 <CR>`	A message appears in the [ROOT] window to “Give search radius:”
[ROOT]	`2 <CR>`	A message appears in the [ROOT] window to “Give mesh density for patches <n> ”. Specify two elements in each patch (surface) direction.
[MENU]	`GENE` UP	Move up one menu. The mesh has been created.

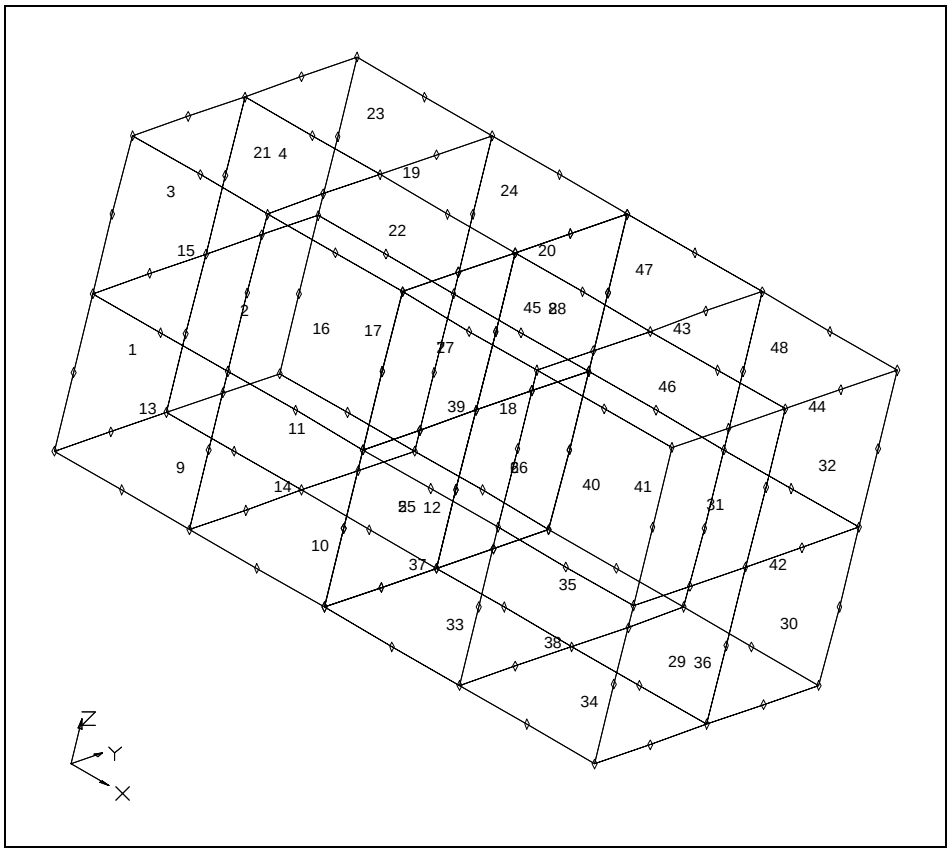
Step 6: Display Boundary Element Mesh

At this point, it would be helpful to display just the node and element entities.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`DISPLAY-OPTIONS`	
[MENU]	`ENTITY`	
[MENU]	`ALL`	
[PROMPT]	`[OFF]`	Turn everything OFF.
[MENU]	`MESH`	
[PROMPT]	`[ON]`	Turn only MESH items ON (NODES and ELEMENTS).
[MENU]	`GLOB.DISP.ENTI` UP	Move up one menu.
[MENU]	`LABEL`	
[MENU]	`MESH`	
[PROMPT]	`OFF`	
[MENU]	`ELEMENT`	
[MENU]	`ON`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[MENU]	`CDRAW`		See [DISPLAY] window in figure below. Only element numbers are displayed. Note that the elements at the `WALL` are 1-4. The elements at the `TIP` are 29-32.
[MENU]	`GLOB.DISP.LABE`	`UP`	Move up one menu.
[MENU]	`GLOB.DISP`	`UP`	Move up one menu.

Element Numbers



Step 7: Prepare The GPBEST Data File (PREPARE ANALYSIS)

The boundary element model has been defined in terms of nodes and elements. Proceed to prepare the GPBEST data file.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`PREPARE-ANALYSIS`	
[ROOT]	`<CR>`	Press <CR> to continue with "Prepare Analysis."
[MENU]	`CASE`	
[MENU]	`TITLE`	
[ROOT]	`3-D BEAM <CR>`	
[MENU]	`DIMENSION`	
[MENU]	`1. TDAN: 3-D ANALYSIS`	
[MENU]	`1. NONE`	No symmetry.
[MENU]	`PREP.CASE.DIME UP`	
[MENU]	`ANALYSIS TYPE`	
[MENU]	`ELASTIC`	
[MENU]	`PREP.CASE.ANAL UP`	
[MENU]	`PREP.CASE UP`	**CASE input has been completed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`MATERIAL`	Define material parameters.
[MENU]	`DEFINE`	
[ROOT]	`STEEL <CR>`	Name of the beam material.
[ROOT]	`I <CR>`	Isotropic.
[ROOT]	`0 <CR>`	Temperature T=0.
[ROOT]	`207000.0 <CR>`	Young's Modulus $\left(\frac{N}{mm^2}\right)$.
[MENU]	`0.3 <CR>`	Poisson's Ratio.
[ROOT]	`7.833E-9 <CR>`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen that are consistent with dimensions of the model which were mm.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	PREP.MATE UP	Move up one menu. **MATE input has been completed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	GMR	
[ROOT]	0 <CR>	Reference temperature T=0.
[MENU]	NEXT GMR	
[PROMPT]	[NO]	No reference points in GMR1.
[ROOT]	0 <CR>	Reference temperature T=0.
[MENU]	NEXT GMR	
[PROMPT]	[NO]	No reference points in GMR2.
[ROOT]	N <CR>	No GAP card. In the [DISPLAY] window, the interface is highlighted.
[ROOT]	1 <CR>	Use a BONDED interface. This acts as though the two GMRs are welded at the interface.

Step 8: Define The Boundary Conditions For The Model

Still in PREPARE DATA, the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	BCSET	
[ROOT]	WALL <CR>	Name of boundary condition set.
[ROOT]	GMR1	Boundary condition is located in GMR1. Note that this is *case-sensitive*.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[ROOT]	G <CR>	Global value.
[ROOT]	S <CR>	Selected elements.
[PROMPT]	[NO]	Do not use digitizer to select elements.
[PROMPT]	[NO]	Do not turn hidden on..
[PROMPT]	<ESC> A <CR>	Add entities.
[ROOT]	1-4 <CR>	Elements 1,2,3,4.
[ROOT]	. <CR>	Period (end input).
[ROOT]	C <CR>	In [DISPLAY] window, element 4 is highlighted. Confirm that this is correct.
[ROOT]	A <CR>	All nodes on elements 1-4.
[ROOT]	N <CR>	No time variation.
[ROOT]	D <CR>	Displacement.
[ROOT]	1 <CR>	Constrain *X*-direction.
[ROOT]	0 <CR>	Zero magnitude.
[ROOT]	C <CR>	Confirm.
[ROOT]	Y <CR>	Proceed to constrain *Y*-direction.
[ROOT]	G <CR>	Global value.
[ROOT]	N <CR>	No time variation.
[ROOT]	D <CR>	Displacement.
[ROOT]	2 <CR>	Constrain *Y*-direction.
[ROOT]	0 <CR>	Zero magnitude.
[ROOT]	C <CR>	Confirm.
[ROOT]	Y <CR>	Proceed to constrain *Z*-direction.
[ROOT]	G <CR>	Global value.
[ROOT]	N <CR>	No time variation.
[ROOT]	D <CR>	Displacement.
[ROOT]	3 <CR>	Constrain *Z*-direction.
[ROOT]	0 <CR>	Zero magnitude.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[ROOT]	`C <CR>`	Confirm.
[ROOT]	`N <CR>`	No more BCs in WALL BCSET.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`BCSET`	Proceed to define the distributed load (traction) at the right edge.
[ROOT]	`TIP <CR>`	Name of boundary condition set.
[ROOT]	`GMR2`	However the GMR names are listed in the [ROOT] window is how they should be typed here. In particular, pay attention to *case-sensitivity*.
[ROOT]	`G <CR>`	Global value.
[ROOT]	`S <CR>`	Selected elements.
[PROMPT]	`[NO]`	No digitizer.
[PROMPT]	`[NO]`	No hidden.
[PROMPT]	`<ESC> A <CR>`	Add entities.
[ROOT]	`29-32 <CR>`	Elements 29-32.
[ROOT]	`. <CR>`	Period.
[ROOT]	`C <CR>`	In [DISPLAY] window, elements 29-32 are highlighted.
[ROOT]	`A <CR>`	All nodes on elements.
[ROOT]	`N <CR>`	No time variation.
[ROOT]	`T <CR>`	Traction.
[ROOT]	`1 <CR>`	X-direction.

INPUT FOCUS	*COMMAND OR DATA ENTRY*		*COMMENTS*
[ROOT]	`1.25 <CR>`		Magnitude. Note that this is a force per unit area. The total force is 2000 N. The dimension of the tip is 40^2 mm^2. Therefore, 2000/(40x40) = 1.25 N/mm^2.
[ROOT]	`C <CR>`		Confirm.
[ROOT]	`N <CR>`		No more BCs.
[MENU]	`PREP`	`UP`	Move up one menu.
			**BCSET input has been completed.
[MENU]	`EXIT`		Proceed to exit BESTVIEW.
[PROMPT]	`[YES]`		Save this database.
[PROMPT]	`[YES]`		Exit BESTVIEW now.

Step 9: Run GPBEST To Analyze The Boundary Element Model

At this point, the GPBEST data file has been created by BESTVIEW. Its name is `tut3a.DAT`. Proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut3a.DAT best-s > tut3a.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut3a.neu`**

Step 10: Post-Processing The Results

For post-processing, BESTVIEW can read the GPBEST neutral file directly. Refer to the example of post-processing in BESTVIEW which was given in Chapter 4.

PATRAN Model

TIP: This tutorial covers MSC/PATRAN 7.5 or higher.

Step 1: Start PATRAN

In this step, you will start the PATRAN program and name the database you will be using.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`patran`**	Start PATRAN.
[MENU]	`File`	
[MENU]	`New...`	
[New Database]	`New Database Name: tut3b [OK]`	This will name the database `tut3b.db`.
[New Model Preferences]	`[OK]`	Accept defaults.

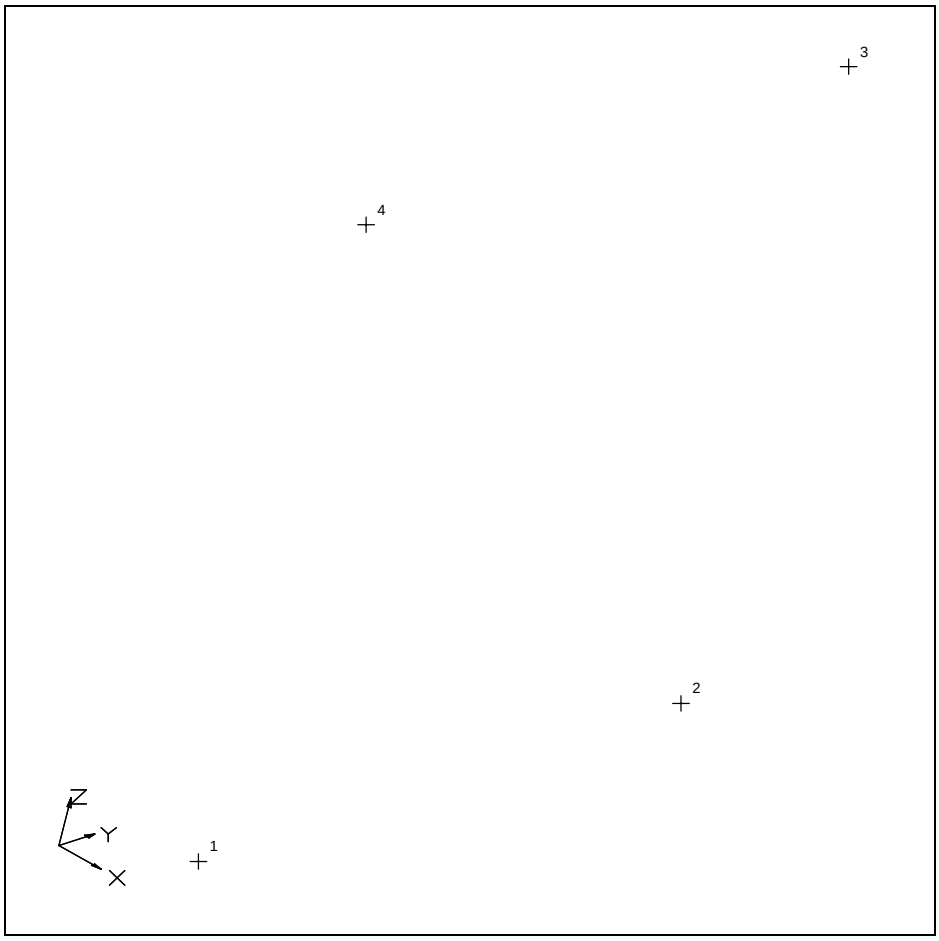
Step 2: Define Four Points

The method of creating this beam model will be to create the first GMR (i.e., group), then create the second GMR. To create the first GMR, we will define four corner points, define a surface from them, translate the surface to create a second surface, and define the remaining surfaces through the corner points. The first GMR will be meshed. The second GMR will be created from the first GMR using PATRAN groups.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Geometry`	Two windows are displayed.
[Geometry]	`Action: Create`	
	`Object: Point`	
	`Method: XYZ`	
[Geometry]	`[Apply]`	Accept (0,0,0) for first point.
[Geometry]	`Point Coordinates List: [0,40,0] [Apply]`	Point 2.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Geometry]	`Point Coordinates List:`	Point 3.
	`[0,40,40] [Apply]`	
[Geometry]	`Point Coordinates List:`	Point 4.
	`[0,0,40] [Apply]`	
		At this point, the four points should appear in the [DISPLAY] window.
[MENU]	`Viewing`	
[MENU]	`Angles...`	
[View Angles]	`(click on Screen Absolute)`	
[View Angles]	`Angles: -60 -30 -30 [Apply]`	
[Viewing]	`[Fit View]`	
[View Angles]	`[Cancel]`	
[MENU]	`Display`	
[MENU]	`Geometry`	
[Geometric Attributes]	`Point:  (check the box in front of “Label”)`	
	`[Apply]  [Cancel]`	

Geometric Points Defining The Corners Of The End Section Of The Beam



Step 3: Define The Geometric Surfaces

The four points defined in the previous step can be used to construct a geometric surface. From this surface, a second surface can be constructed through a translation. Once these two surfaces are defined along with their corner points, the remaining four surfaces of the cube can be defined via corner points.

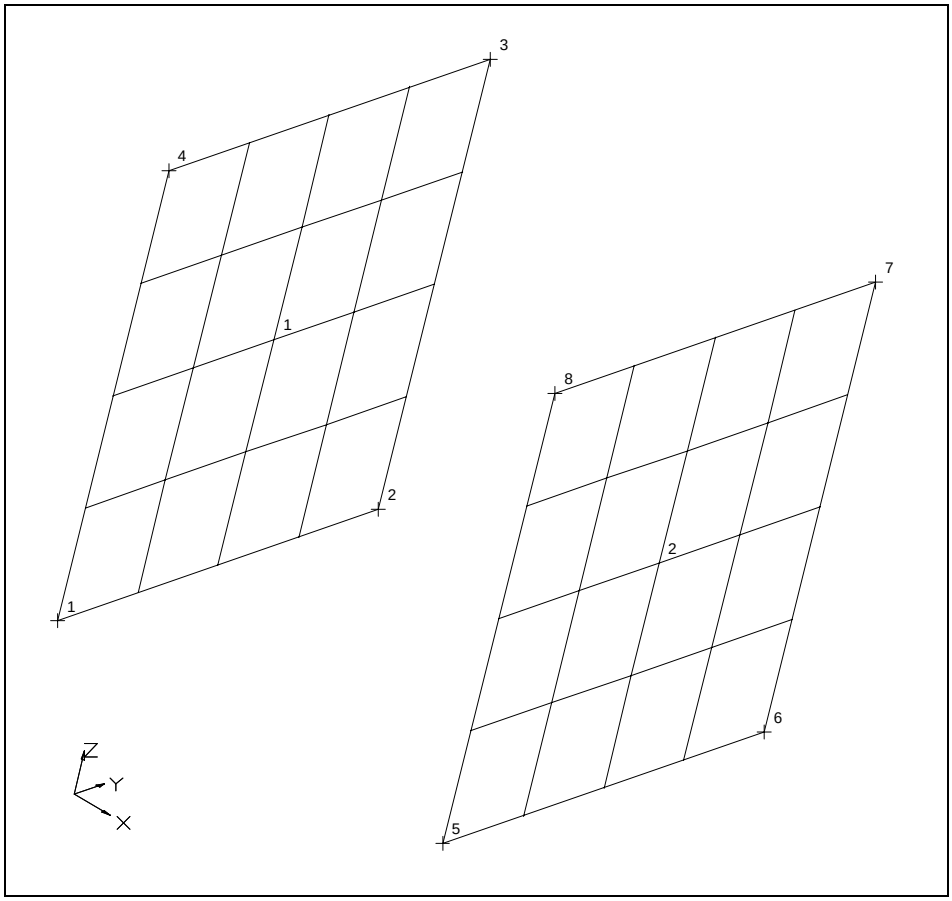
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Geometry]	`Action: Create`	
	`Object: Surface`	
	`Method: Vertex`	
[DISPLAY]	`(click on points 1,2,3,4)`	Surface 1 (i.e., patch 1) will be created.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Geometry]	`Action: Transform`
	`Object: Surface`
	`Method: Translate`
		Now translate Surface 1 to create Surface 2.

[Geometry]	`Translation Vector:` `<40 0 0>`	
[Geometry]	`Surface List: Surface 1`	
[Geometry]	`[Apply]`	Surface 2 is created.

[DISPLAY] Window After Creating Surface 2

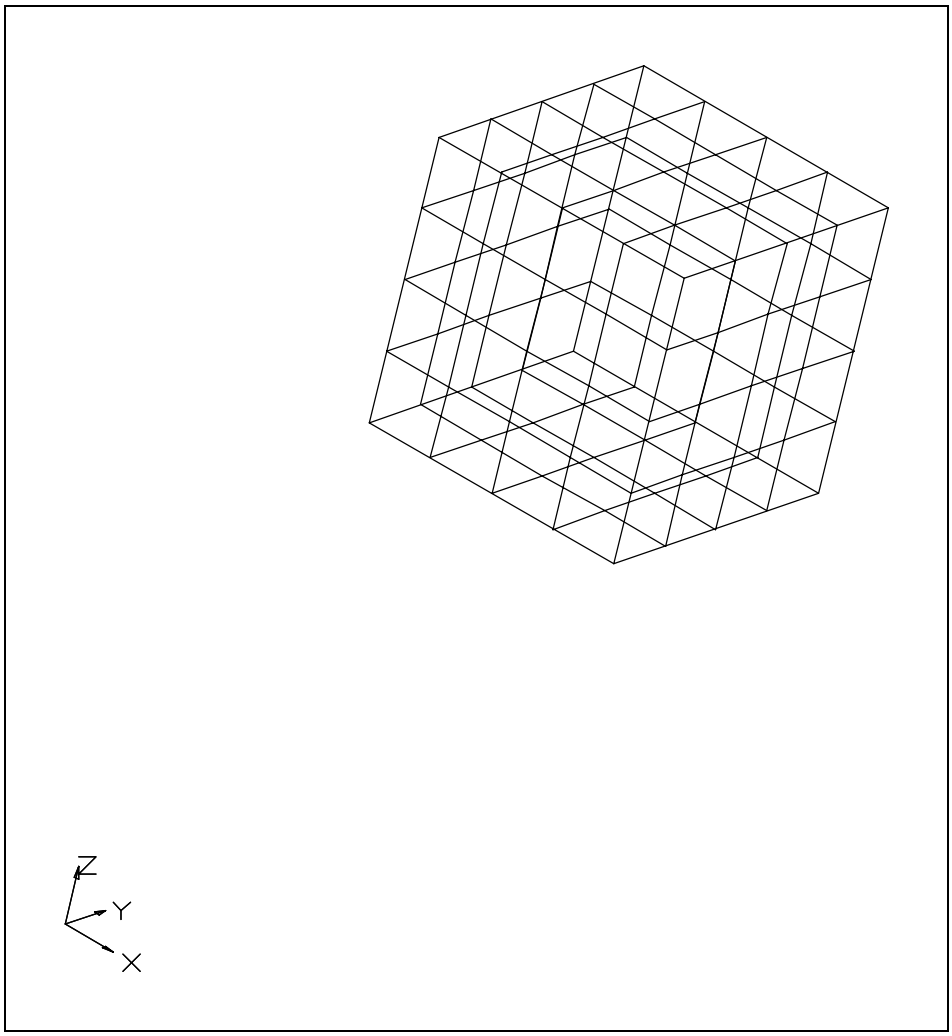


INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Geometry]	`Action: Create`
	`Object: Surface`
	`Method: Vertex`
		Proceed to create the other four surfaces by picking corner points.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[DISPLAY]	`Click on points 1,5,8,4.`	Surface 3.
[DISPLAY]	`Click on points 1,5,6,2.`	Surface 4.
[DISPLAY]	`Click on points 2,6,7,3.`	Surface 5.
[DISPLAY]	`Click on points 4,8,7,3.`	Surface 6.
		At this point, six surfaces have been created, one on each face of the cube.

Surfaces On The Six Faces Of The Cube



Step 4: Define Material Properties

Material properties are now defined. Start by defining the STEEL material property. Note that the material property only needs to be defined in PATRAN. It does not need to be associated with any elements. The PREBEST translator will extract the material properties from the neutral file.

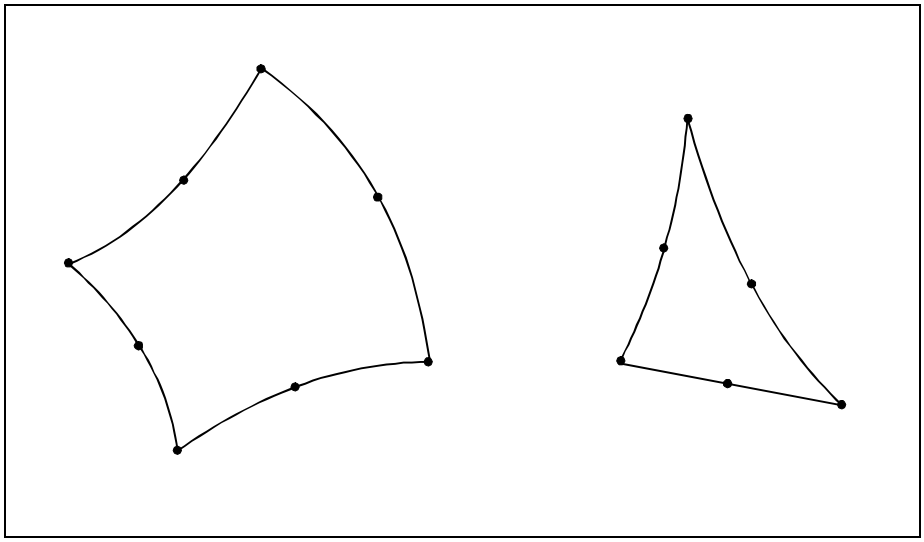
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Materials`	Define material parameters.
[Materials]	`Action: Create`	
	`Object: Isotropic`	
	`Method: Manual Input`	
[Materials]	`Material Name: Steel`	
[Materials]	`Input Properties...`	
[Input Options]	`Constitutive Model:`	
	`Linear Elastic`	
[Input Options]	`Elastic Modulus:207000.`	Young's modulus $\left(\frac{N}{mm^2}\right)$.
[Input Options]	`Poisson Ratio: 0.3`	
[Input Options]	`Density: 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.
[Input Options]	`[Apply]`	
[Input Options]	`[Cancel]`	Note that it is not necessary to assign "Property" information (Area, moment of inertia, beam orientation, etc.) to elements.

Step 5: Create A Boundary Element Mesh

The preceding surfaces can be used to generate a boundary element model. Although PATRAN does not support boundary elements directly, boundary elements are simulated using 3-D shell elements. These elements do not require properties, such as thickness, orientation, etc., since they are not true shells. The

translator step (PREBEST) later converts them into GPBEST boundary elements.

3-D Boundary Elements



As illustrated in this figure, make sure you use 6-noded triangular or 8-noded rectangular shell elements in PATRAN. Although 3-noded and 4-noded shell elements are more popular in finite element modeling, it is preferable to use quadratic elements in GPBEST.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Finite Elements`	
[Finite Elements]	`Action: Create`	
	`Object: Mesh`	
	`Type: Surface`	
[Finite Elements]	`Global Edge Length=20`	This will result in four surface elements on each cube face.
	`Element Topology: Quad8`	
[Finite Elements]	`Surface List:`	
	`Surface 1:6`	Mesh has been created.
	`[Apply]`	
[Finite Elements]	`Action: Equivalence`	Equivalence coincident nodes.
	`Object: All`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Finite Elements]	`Equivalencing Tolerance: .005`	
[Finite Elements]	`[Apply]`	
[Finite Elements]	`Action: Renumber`	This step is not really necessary. It is performed for the user's convenience.
[Finite Elements]	`Start ID: 1` `Node List: Node 1:#`	
[Finite Elements]	`[Apply]`	Nodes are renumbered from 1 to 74.

Step 6: Define The Groups

The preceding entities can all be put into one group called GMR1. This GMR can be copied to create GMR2. Notice that one result of creating the model this way is that a duplicate set of nodes and elements exist on the interface. However, each GMR contains a unique set of nodes and elements. The POSTBEST translator will detect the interface properly.

Note also that the nodal equivalencing was performed only on the nodes in group GMR1. If the entire model had been created first, we would have to be very selective about equivalencing nodes. The nodes must be equivalenced *on a group-by-group basis*. This is the only way to prevent the nodes on the interface from collapsing during the equivalence operation.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Group`	
[MENU]	`Create...`	
[Group]	`Action: Create`	
[Group]	`New Group Name: GMR1`	
[Group]	`Group Contents:` `Add All Entities`	Place everything in group GMR1. Alternatively, finite element entities (i.e., nodes and elements) could be placed exclusively in the group.
[Group]	`[Apply]`	

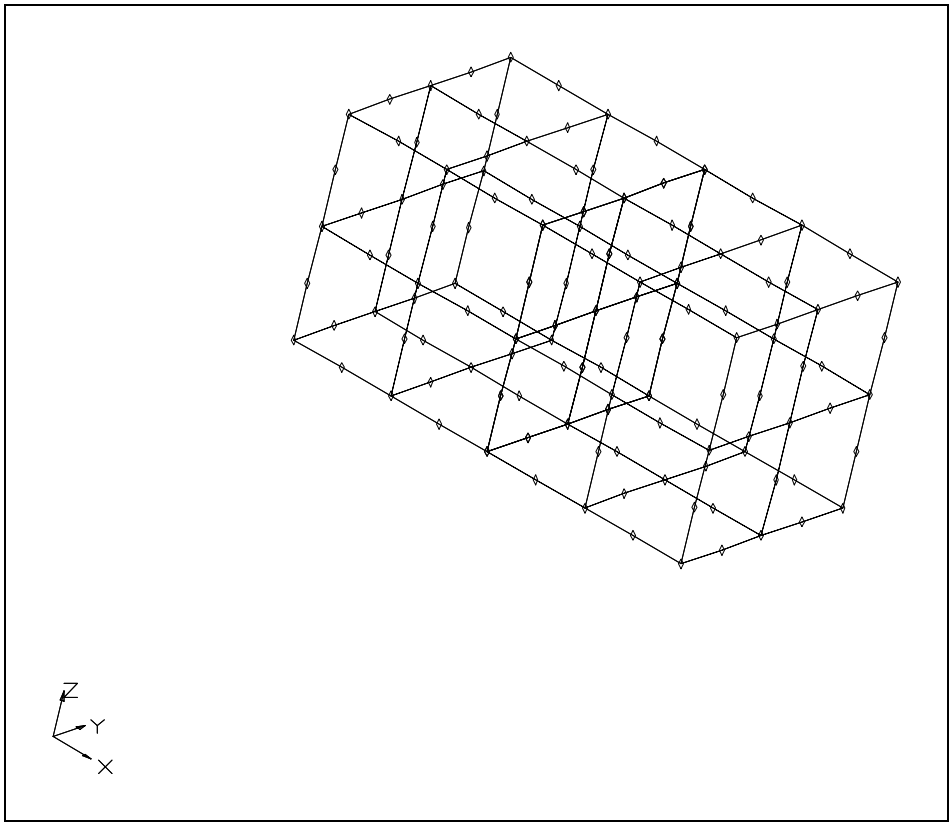
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Group]	`New Group Name: GMR2`	In this step, a second group (GMR2) is defined. It will be used to store entities created by a subsequent transformation. For the time being, it is empty.
[Group]	`Group Contents:` `Add Entity Selection`	
[Group]	`[Apply]`	
[Group]	`Action: Transform`	Proceed to create a second group GMR2, obtained by translating and copying group GMR1 40 mm in the *X*-direction.
[Group]	`Action: Transform` `Method: Translate`	
[Group]	`Current Group: GMR2`	
[Group]	`Change Target Group...`	
[Change Target Group]	`(click on GMR1)`	
[Change Target Group]	`[Cancel]`	
[Group]	`Translation Vector:` `< 40 0 0 >`	
[Group]	`[Apply]`	
[Message]	`[Yes For All]` `[Yes For All]`	Message indicating that there are duplicate points, elements, etc. This is acceptable. There will be duplicate points on the interface. The PREBEST translator will correctly handle this.

Step 7: Display Boundary Element Mesh

At this point, the model display in PATRAN can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Display`	
[MENU]	`Plot/Erase...`	
[Plot/Erase]	`[Erase All Geometry]`	The geometric entities no longer need to be displayed.
[Plot/Erase]	`[OK]`	
[MENU]	`Display`	
[MENU]	`Finite Elements…`	
[FEM Attributes]	`[Hide All Entity Labels]`	Turn off all labels.
[FEM Attributes]	`[Apply] [Cancel]`	The model, along with node number labels, appears in the [DISPLAY] window.

Nodes And Boundary Elements For The Entire Model



Step 8: Define The Boundary Conditions For The Model

Now the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X, Y* and *Z* degrees of freedom on the elements at the wall.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual contains an extensive description on specifying GPBEST boundary conditions in PATRAN. It is best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 113 is used at the wall since *X, Y* and *Z* displacements are zero there. Load set 101 is used at the tip of the beam since the *X*-traction is non-zero there.

Pay particular attention to the fact that while using PATRAN, in 2-D models the boundary conditions are specified on *nodes*, while in 3-D they are specified on *elements*. In 3-D models, such as the one here, *element pressure* is used to specify *all* boundary conditions for GPBEST. Whether the boundary condition is a traction or a displacement is determined by the load set ID.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Viewing`	It will help to change the view back to the *X-Y* plane. This will make selecting the boundary conditions much easier.
[MENU]	`Angles...`	
[View Angles]	`(select Screen Absolute)`	
[View Angles]	`0 0 0`	
[View Angles]	`[Apply] [Cancel]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Load Cases`	First, define the load cases applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.
[Load Cases]	`Action: Create`	
[Load Cases]	`Load Case Name: 101`	*X*-component of traction is non-zero at tip of beam.
[Load Cases]	`[Apply]`	
[Load Cases]	`Load Case Name: 113`	*X-, Y- and Z*-components of displacement are zero at the wall.
[Load Cases]	`[Apply]`	All load cases that are needed are listed in the [Existing Load Cases] section.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Loads/BCs`	

[Load/Boundary Cond]	`Action: Create`
	`Object: Pressure`
	`Type: Element Uniform`
	`Analysis type: Structural`
		Element pressure is used even though this is a displacement boundary condition. Load set ID=113 specifies that this is a displacement boundary condition.

[Load/Boundary Condition]	`Current Load Case: (Click on this box to select next load set.)`	Work on the WALL BC first.
[Change Current Load Case]	`(Select load case 113)` `[OK]`	113=zero displacement components in *X,Y,Z* directions.
[Load/Boundary Condition]	`New Set Name: WALL`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Load/Boundary Condition]	`Target Element Type: 2D`	
[Load/Boundary Condition]	`[Input Data...]`	
[Input Data]	`Top Surface Pressure: 0` `[OK]`	
[Input Data]	`[Select Appl Region...]`	
[Select Application Region]	`Geometry Filter: FEM`	Turn on FEM filter.
[DISPLAY]	`(Draw box around left edge of model)`	You should select *all* elements on the left edge of the model. Alternatively, enter “Element 1:4” after “Select 2D Elements or Edges”.
[Select Application Region]	`[Add]` `[OK]`	Elements 1-4 are selected.
[Load/Boundary Condition]	`[Apply]`	
[Load/Boundary Condition]	`Current Load Case: (Click on this box to select next load set.)`	Now work on the traction load at the `TIP` of the beam.
[Load/Boundary Condition]	`(Select load case 101)` `[OK]`	101=non-zero traction.
[Load/Boundary Condition]	`New Set Name: TIP`	
[Load/Boundary Condition]	`[Input Data...]`	

[Input Data]	`Top Surface Pressure:`
	`1.25`
	`[OK]`
		Here, the traction magnitude is specified as 1.25 N/mm^2. Direction is specified by using the Load Set ID 101.

[Input Data]	`[Select Appl Region...]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Select Application Region]	`Geometry Filter: FEM`	Turn on FEM filter.
[DISPLAY]	`(Draw box around right edge of the model)`	You should select *all* elements on the right edge of the model. Alternatively, enter “Element 29-32” after “Select 2D Elements or Edges”.
[Select Application Region]	`[Add]` `[OK]`	Elements 29-32 are selected.
[Load/Boundary Condition]	`[Apply]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File`	
[MENU]	`Export...`	
[Export]	`Format: Neutral` `Create New File: /users/john/work/tut3b.out`	This creates a PATRAN neutral file named `tut3b.out.1`.
[Export]	`[Apply]`	
[Warnings]	`[OK]`	Ignore warning message.
[MENU]	`File`	
[MENU]	`Quit`	Exit from PATRAN. It is sometimes advantageous to examine the PATRAN “Journal File” (`tut3b.db.jou`). This file lists the commands and messages given during a PATRAN session.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 9: Run GPBEST To Analyze The Boundary Element Model

At this point, the PATRAN neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`3 <CR>`	PATRAN.
[PREBEST]	`tut3b.out.1 <CR>`	PATRAN neutral file name.
[PREBEST]	`Y <CR>`	"Yes." This is the correct file.
[PREBEST]	`tut3b.dat <CR>`	Write to `tut3b.dat` file (default).
[PREBEST]	`1 <CR>`	3-D model.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`3 <CR>`	High level of quality on element checks.
[PREBEST]	`1 <CR>`	Since there are two GMRs in this model, PREBEST is asking whether you want to have the same functional variation in both GMRs.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`1 <CR>`	Use a bonded interface. This acts as though the two GMRs were welded at their interface.
[PREBEST]	`N <CR>`	No body forces.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
		The PREBEST program ends, but the file `tut3b.dat` has been created. It can be run directly in GPBEST. Note that it is sometimes useful to examine the `patbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining whether or not the model was completely translated.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**< tut3b.dat best-s > tut3b.res <CR>**
[UNIX]	**mv NEUTRAL.OUT tut3b.neu**

Step 10: Post-Processing The Results

You can view the GPBEST results file (`tut3b.res`) and determine if GPBEST ran successfully by using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, you can proceed to view the results in PATRAN. For this step, it is best to start PATRAN with a new database.

Since PATRAN cannot read in the GPBEST neutral file directly, the POSTBEST translator must be run in order to create files that PATRAN will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**postbest**<CR>	
[POSTBEST]	3 <CR>	PATRAN.
[POSTBEST]	tut3b.neu <CR>	Name of GPBEST neutral file.
[POSTBEST]	N <CR>	Do not renumber the nodes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

		Three files are created:
		`PATRAN.GEOM`
		`PATRAN.DIS.1`
		`PATRAN.NOD.1`

To post-process the model in PATRAN, read in the PATRAN neutral file (`PATRAN.GEOM`) and the PATRAN results files (`PATRAN.NOD.1, etc.`) as demonstrated in Tutorial 2 (Chapter 4).

I-DEAS Model

A Note About The Units Used In I-DEAS

Please see the note about units in the I-DEAS tutorial in Chapter 4. We will keep track of our own units and ignore the units given in I-DEAS. SI units will be chosen merely to turn off unit conversions in I-DEAS.

TIP: This tutorial covers I-DEAS Master Series 6 or higher.

Step 1: Start I-DEAS

In this step, you will start the I-DEAS program and name the database you will be using. It is also good to set some view settings before proceeding.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`ideas`**	Start I-DEAS.
[I-DEAS Start]	`Project: john`	
	`Model File Name: tut3c.mf1`	
	`Application: Simulation`	
	`Task: Master Modeler`	
[I-DEAS Start]	`[OK] [OK]`	Create model database.
[MENU]	`[Options]`	
[MENU]	`[Units]` `Meter (newton)`	This turns off unit conversion in I-DEAS. See note at the beginning of the I-DEAS section of Chapter 4.
[MENU]	`[Options]` `[View Settings...]`	

[View Settings]	`View Angle: -60,-30,-30`
	`Viewport Diameter: 120`
	`[OK]`
		This is a useful isometric viewing angle.

Step 2: Define The Geometry Of The Beam

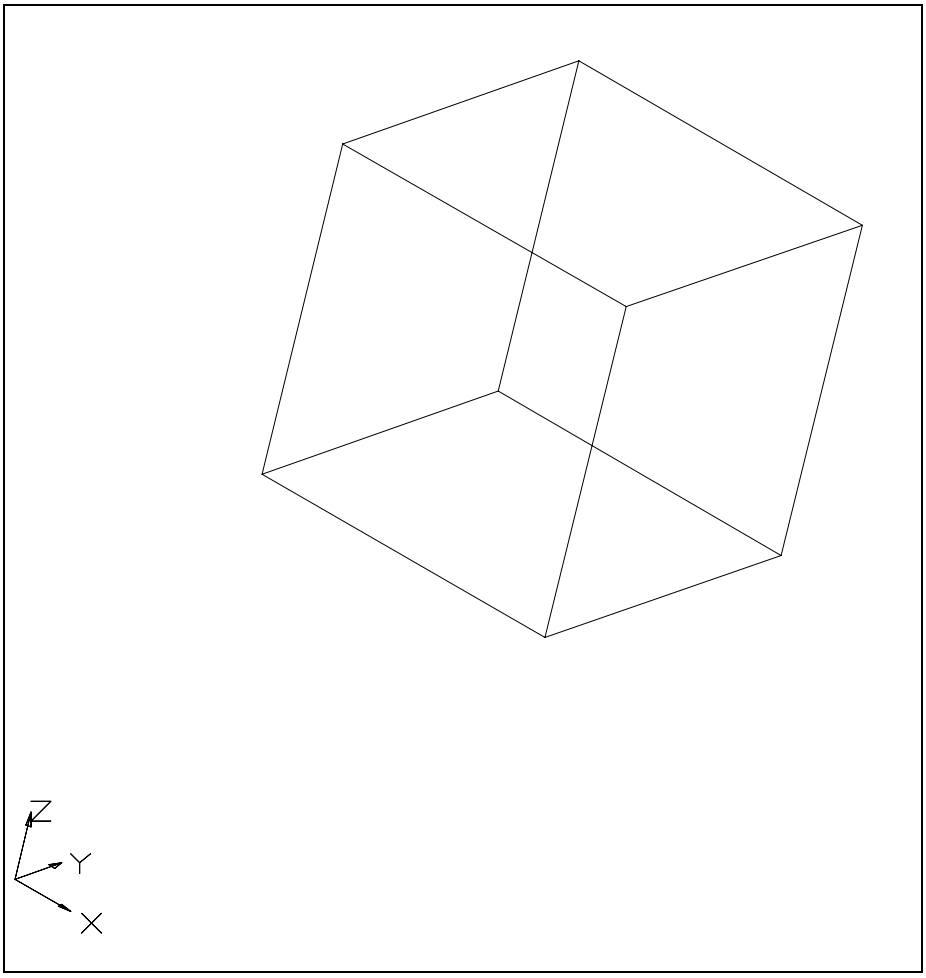
I-DEAS allows the user to create very detailed geometry in 2-D and 3-D. In this case, the beam will be modeled by a “block” of the appropriate dimensions. This is merely geometry and has nothing to do with nodes and elements.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Parts...`	

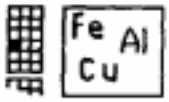
[Part Catalog]	`[Block]`
	`Length: 40`
	`Height: 40`
	`Depth: 40`
		The block is 40mm x 40mm x 40mm.

[Part Catalog]	`[OK]`	A block defining the beam geometry has been created. This defines the cube geometry for GMR1.
[Icons]	`Name` `Parts...`	
[Name]	`(move cursor until it is near the block face and it appears highlighted; click mouse)`	
[Name]	`Name: Box1 [OK]`	

The Named Part Box1



Step 3: Define Material Properties

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Task]	`[Master Modeler]` `change to [Meshing]`	Switch to FEM meshing mode.
[Icons]	 `Materials...`	It is convenient to define material properties before meshing.
[Materials]	`Material Label: 1` `Name: Steel1`	
[Materials]	`[Quick Create]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Quick Create]	`[Modulus of Elasticity]`
	`207000`
	`[Modify Value]`
		Young's Modulus $\left(\frac{N}{mm^2}\right)$.
		Ignore the units given by I-DEAS, since we are keeping track of our own units.

[Quick Create]	`[Poisson ratio]`
	`0.3`
	`[Modify Value]`	

[Quick Create]	`[Mass Density]`
	`7.833E-9`
	`[Modify Value]`
		Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$.
		Notice that units are chosen consistent with dimensions of the model, which are mm.

[Quick Create]	`[OK]`	
[Materials]	`[OK]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Create FE Model...`	Proceed to generate the mesh.

[FE Model Create]	`Part: Box1`
	`Material: Steel1`
	`FE Model Name: Fem1`	

[FE Model Create]	`[OK]`	

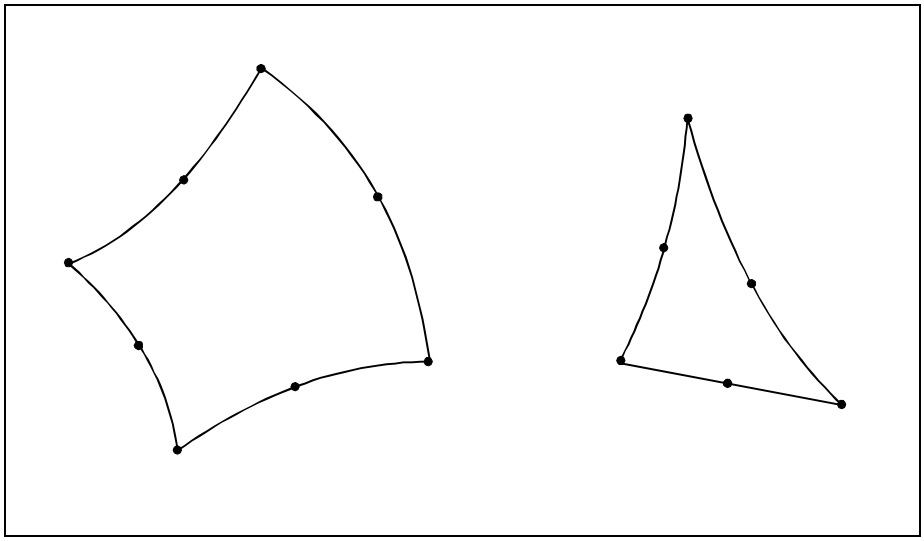
Step 4: Create A Boundary Element Mesh

The boundary element mesh is now created. It consists of boundary elements and nodes. Four boundary elements will be created on each face of the block. Each boundary element will

contain eight nodes (four corner nodes and four mid nodes). Hence, it is called a *quadratic* element.

Although I-DEAS does not support boundary elements directly, boundary elements are simulated using 3-D shell elements. These elements do not require properties, such as thickness, orientation, etc., since they are not true shells. The translator step (PREBEST) later converts them into GPBEST boundary elements.

3-D Boundary Elements



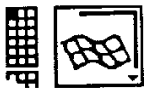
As illustrated in this figure, make sure you use 6-noded triangular or 8-noded rectangular shell elements in I-DEAS. Although 3-noded and 4-noded shell elements are more popular in finite element modeling, it is preferable to use quadratic elements in GPBEST.

Equivalencing and renumbering of nodes and elements are not necessary if you create a mesh in I-DEAS. I-DEAS automatically equivalences and renumbers entities.

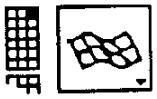
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Create FE Model...`	Proceed to generate the mesh.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

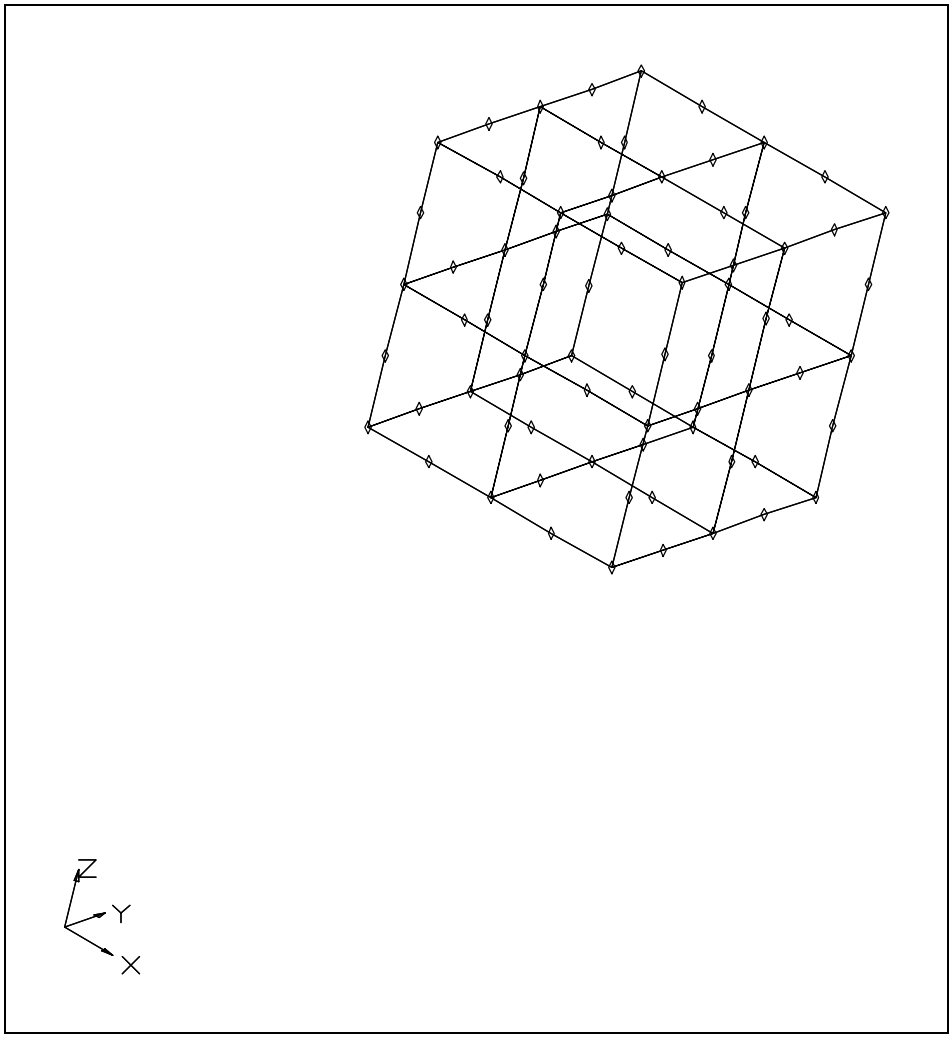
[FE Model Create]	`Part: Box1`
	`Material: Steel1`
	`FE Model Name: Fem1`	

[FE Model Create]	`[OK]`	
[Icons]	 `Define Shell Mesh...`	
[DISPLAY]	`(Hold down Right Mouse button and select [All])`	This selects all six faces of the block.
[DISPLAY]	`(click Middle Mouse button to confirm)`	

[Define Mesh]	`Mesh Type: Free`
	`Element Family: Thin Shell`
	`Element Type: 8 noded`
	`Element Length: 20`
		This will result in four elements per face.

[Define Mesh]	`[Apply] [OK]`	
[Icons]	 `Shell Mesh...`	
[DISPLAY]	`(Hold down Right Mouse button and select [All])`	This selects all six faces of the block.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[DISPLAY]	`[YES]`	Keep this mesh. The model should appear as in the following figure. (You may have to tell IDEAS to display nodes and elements.)

Nodes And Boundary Elements For GMR1



Step 5: Define The Groups

At this point, a surface mesh has been generated on the faces of the cube. The following steps illustrate how GMRs are defined through the Permanent Group feature in I-DEAS.

Steps To Follow To Create GMR2 From GMR1

1. GMR1 is defined to be the cube we already meshed.
2. A copy of these entities are translated 40mm in the *X*-direction, resulting in the second cube.
3. GMR3 is a temporary group defined to consist of ALL entities; i.e., both cubes.
4. GMR2 is defined to be (GMR3 - GMR1). In other words, GMR2 contains all entities that are not in GMR1.

5. Having served its purpose, GMR3 is deleted. What remains is GMR1 (the left cube) and GMR2 (the right cube).

An Important Note about modeling GMRs in I-DEAS

NOTE: The following procedure merely illustrates one way to create GMR2 from GMR1. However, in actual modeling practice, many other methods can be used to create GMR2. Experienced users may choose a different approach.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Group...`	Define a Permanent Group which will be recognized by the PREBEST translator as GMR1.
[Group]	`Create Group...`	
[DISPLAY]	`(Hold down Right Mouse button and select [All])`	
[DISPLAY]	`(select [All of Above])`	This selects all nodes and elements.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Prompt]	`GMR1 <CR>`	Permanent Group name.

At this point, group GMR1 contains all nodes and elements that have been defined on the part ‘Box1.’ Proceed to create what will eventually be called GMR2 by translating (and copying) GMR1 40*mm* in the *X*-direction.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Prompt]	`/element <CR>`	
[Prompt]	`copy_and_orient <CR>`	
[DISPLAY]	`(Hold down Right Mouse button and select [All])`	
[DISPLAY]	`(click Middle Mouse button to confirm)`	All elements are selected.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Prompt]	`<CR>`	New elements will start with 25 and an increment of 1.
[Prompt]	`<CR>`	New node numbers will start with 75 and an increment of 1.
[DISPLAY]	`(select [Translate])`	
[Prompt]	`40,0,0 <CR>`	
[DISPLAY]	`[YES]`	Keep the elements and nodes.

At this point, the new GMR has been created. However, the new elements and nodes must be placed in a new group. Proceed by creating GMR3, which contains all nodes and elements in the model.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Group...`	
[Group]	`Create Group...`	
[DISPLAY]	`(Hold down Right Mouse button and select [All])`	
[DISPLAY]	`(select [All of Above])`	This selects all nodes and elements.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Prompt]	`GMR3 <CR>`	Permanent Group name.

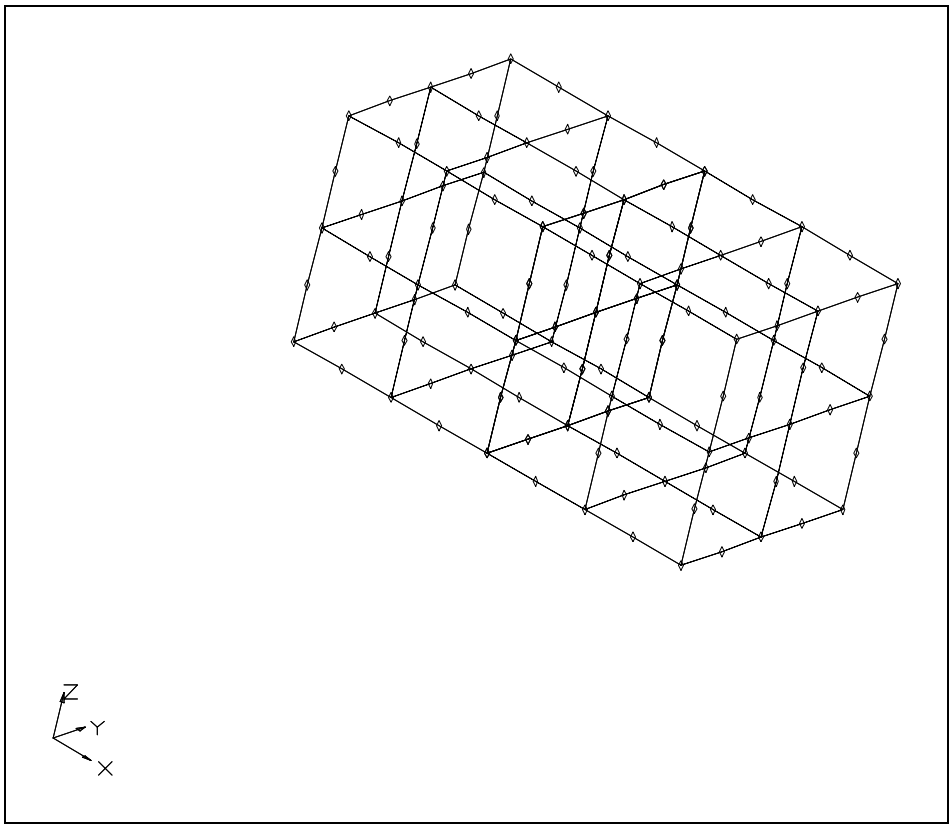
GMR3 now contains all nodes and elements. GMR1 contains nodes and elements in the left half of the model. Now proceed to define GMR2, which contains only the nodes and elements in the right half of the model.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Group]	Create Group...	
[DISPLAY]	(Hold down Right Mouse button and select [Use Group])	
[DISPLAY]	(select [Boolean Groups]	
[Prompt]	GMR3 <CR>	What is actually happening is that we are using the “Boolean Difference” operation: GMR2 = GMR3 - GMR1.
[DISPLAY]	(select [Difference])	
[Prompt]	GMR1 <CR>	
[DISPLAY]	Done	
[DISPLAY]	(click Middle Mouse button to confirm)	Done selecting nodes and elements.
[Prompt]	GMR2 <CR>	
[Prompt]	/group <CR>	Group GMR3 has served its purpose. It is no longer needed.
[Prompt]	delete <CR>	
[Prompt]	GMR3 <CR>	
[DISPLAY]	[YES]	GMR3 is deleted. GMR1 and GMR2 remain.
[Group]	[Close]	

Step 6: Display Boundary Element Mesh

At this point, you can manipulate the model display in I-DEAS. It is helpful to check the model, connectivity, etc. The following figure illustrates the nodes and boundary elements in the two GMRs:

Nodes And Boundary Elements For The Entire Model



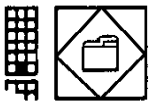
Step 7: Define The Boundary Conditions For The Model

Now the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X, Y* and *Z* degrees of freedom on the elements at the wall.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual contains an extensive description on specifying GPBEST boundary conditions in I-DEAS. It is best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 113 is used at the wall since *X*, Y, and *Z* displacements are zero there. Load set 101 is used at the tip of the beam since the *X*-traction is non-zero there. Pressures are used to model both loads and boundary conditions.

Proceed to define the fixed boundary condition at the `WALL` (i.e., the face at *X=0*).

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Task]	`[Meshing]` `change to [Boundary Conditions]`	First, define the load cases applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.
[Icons]	 `Sets ...`	The proper Load Set number must be defined..

[Set Management]	`Type: Load`
	`Set: 113`
	`[Create]`
	`[Make Current]`
	`[Dismiss]`
		Load set 113=zero displacement components in *X,Y,Z* directions.

The proper Load Set number has been defined. Proceed to apply the fixed boundary condition to the `WALL`.

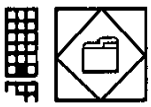
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	`Pressure...`	`WALL` boundary condition.
[DISPLAY]	`(click Right Mouse button and hold it down while selecting [Filter...])`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Selection Filter]	`click on Element_Free_Face`
	`click right arrow`
	`click on Surface...`
	`click left arrow`
	`[OK]`
		The end result of these steps is that “Surface...” is now Non-Pickable, while “Element_Free_Face” is Pickable.

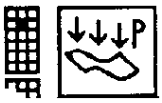
[DISPLAY]	`(Hold down [Shift] key and select all four elements on the WALL face of the block)`	The `WALL` face is at the left side of the model.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Pressure on Element Face]	`Pressure Amplitude: 999` `[OK]`	This is a dummy X-force. It should be non-zero.

The fixed boundary condition has been defined. Proceed to apply the traction to the TIP (i.e., the face of the model at *X=80mm*). First, the proper Load Set number must be defined.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Sets ...`	The proper Load Set number must be defined. Refer to the table in Section 9.5.1.2.3 of the BESTVIEW manual for a listing of the appropriate load set numbers.

[Set Management]	`Type: Load`
	`Set: 101`
	`[Create]`
	`[Make Current]`
	`[Dismiss]`
		Load set 101=non-zero *X*-force.
		NOTE: Before typing in the Load Set number, be sure to erase whatever label appears on the “`Set`” line.

The proper Load Set number has been defined. Proceed to apply the traction load to the `TIP`.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Icons]	 `Pressure...`	`TIP` boundary condition.
[DISPLAY]	`(click Right Mouse button and hold it down while selecting [Filter])`	

[Selection Filter]	`click on Surface...`
	`click left arrow`
	`[OK]`
		The end result of these steps is that “Surface...” is now Non-Pickable, while “Element_Free_Face” is Pickable.

[DISPLAY]	`(Hold down [Shift] key and select all four elements on the TIP face of the block)`	The `TIP` face is at the right side of the model.
[DISPLAY]	`(click Middle Mouse button to confirm)`	
[Pressure on Element Face]	`Pressure Amplitude: 1.25` `[OK]`	Here, the traction magnitude is specified as 1.25 N/mm^2. Direction is specified by using the Set ID 101.
[DISPLAY]		Even though the pressure arrows are pointing inward (-*X*-direction), this does not concern us. We specified a + magnitude.

The boundary conditions have been defined. Save your model and exit I-DEAS.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File`	Note that you must save your database before you Export the model.
[MENU]	`Save`	
[MENU]	`File`	
[MENU]	`Export...`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Export Selections]	`I-DEAS Simulation Universal File`	
[Export Selections]	`[OK]`	
[Export Universal File]	`File Name: tut3c`	This creates an I-DEAS neutral file named `tut3c.unv`.
[Export Universal File]	`(select Fem1)`	
[Message]	`[OK]`	Create new Universal File.
[Export Universal File]	`[OK]`	Watch for messages in the [List] window. They can help determine what was written to the Universal File.
[MENU]	`File`	
[MENU]	`Exit`	Exit from I-DEAS.
[Message]	`[YES]`	Save before exiting I-DEAS.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 8: Run GPBEST To Analyze The Boundary Element Model

At this point, the I-DEAS universal file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

If GPBEST does not execute properly, go back to Chapter 2 and review the GPBEST installation procedure.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`5 <CR>`	I-DEAS MS (R3-R6).
[PREBEST]	`tut3c.unv <CR>`	I-DEAS universal file name.
[PREBEST]	`tut3c.dat <CR>`	Write to `tut3c.dat` file (default).

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[PREBEST]	`1 <CR>`	3D.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`N <CR>`	Do not scale data. It is important that “No” be chosen to retain the original dimensions.
[PREBEST]	`1 <CR>`	Element Severity Check: 1=LOW.
[PREBEST]	`1 <CR>`	Same functional variation over the whole model.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`1 <CR>`	Bonded interface.
[PREBEST]	`N <CR>`	No body forces.
		The PREBEST program ends, and the file `tut3c.dat` has been created. It can be run directly in GPBEST.
		Note that it is sometimes helpful to examine the `ideasbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining if the model was completely translated or not.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut3c.dat best-s > tut3c.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut3c.neu <CR>`**

Step 9: Post-Processing The Results

The GPBEST results file (`tut3c.res`) can be viewed to determine if GPBEST ran successfully by using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, then proceed to view the results in I-DEAS. For this step, it is best to start I-DEAS with a new database.

Since I-DEAS cannot read in the GPBEST neutral file directly, you must run the POSTBEST translator to create files that I-DEAS will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`postbest`**`<CR>`	
[POSTBEST]	`5 <CR>`	I-DEAS MS (R3-R6).
[POSTBEST]	`tut3c.neu <CR>`	Name of GPBEST neutral file.
[POSTBEST]	`N <CR>`	Do not renumber the nodes.
[POSTBEST]	`N <CR>`	Do not scale the data.
	`cp IDEAS.unv tut3cc.unv` `<CR>`	This universal file is created: `IDEAS.unv` Note that it may be helpful to check the `bestideas.log` file for any error messages.

To post-process the model in I-DEAS, read the I-DEAS Universal File (`tut3cc.unv`) into I-DEAS, as was demonstrated in Tutorial 2 (Chapter 4).

HYPERMESH Model

Step 1: Start HYPERMESH

TIP: This tutorial covers HYPERMESH 2.1 or higher.

In this step, you will start the HYPERMESH program and name the database you will be using. It is probably easiest to define material properties right away.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**hm**	Start HYPERMESH.
[MENU]	◆ `pg 1`	Go to Page 1 of the Menus.
[MENU-pg 1]	`[files]`	
[Files]	`File= tut3d.hm`	This will name the database `tut3d.hm`.
	`[save]`	
	`[return]`	
[Permanent Menu]	`[global]`	
[Global]	`Template file: …/feoutput/hm/general`	
[Permanent Menu]	`[return]`	

Step 2: Define Material Properties

[New Model Preferences]	`[collectors]`	
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲	Click on the input collector switch, ▲. Proceed to define the material properties.
	`[mats]`	
[Collectors: Create]	`[name =] STEEL`	This is the name of the material. Overwrite the defailt material name, `part1`.
[Collectors: Create]	`E = 207000.`	Young's modulus $\left(\frac{N}{mm^2}\right)$.
	`Nu = 0.3`	

	`Rho = 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.
[Collectors: Create]	`[create]`	Material property is created.
[Collectors: Create]	`Collector type: ▲` `[comps]`	Hypermesh requires a collector to be defined. Its concept is similar to “groups” in BESTVIEW and GMRs in GPBEST. Click on the input collector switch, ▲. Define a component.
[Collectors: Create]	`[material =]`	
[Choose one material]	`[STEEL]`	The material is STEEL.
[Collectors: Create]	`[name =] GMR1`	This is the name of the first collector.
[Collectors: Create]	`[create]` `[return]`	The collector was created. Now the collector `GMR1` is the current component.

Step 3: Define Eight Points

The method of creating this beam model will be to create the first GMR (i.e., group), then create the second GMR. To create the first GMR, we will define eight corner points, and define the surfaces through the corner points. The first GMR will be meshed. The second GMR will be created from the first GMR using HYPERMESH components.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU-pg 1]	`[create nodes]`	
[Create Nodes]	◆ `type in`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 0`
	`[z =] 0`
	`[create]`
		Accept (0,0,0) for first point.

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 40`
	`[z =] 0`
	`[create]`
		Point 2.

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 40`
	`[z =] 40`
	`[create]`
		Point 3.

[Create Nodes: Type In]	`[x =] 0`
	`[y =] 0`
	`[z =] 40`
	`[create]`
		Point 4.

[Create Nodes: Type In]	`[x =] 40`
	`[y =] 0`
	`[z =] 0`
	`[create]`
		Point 5.

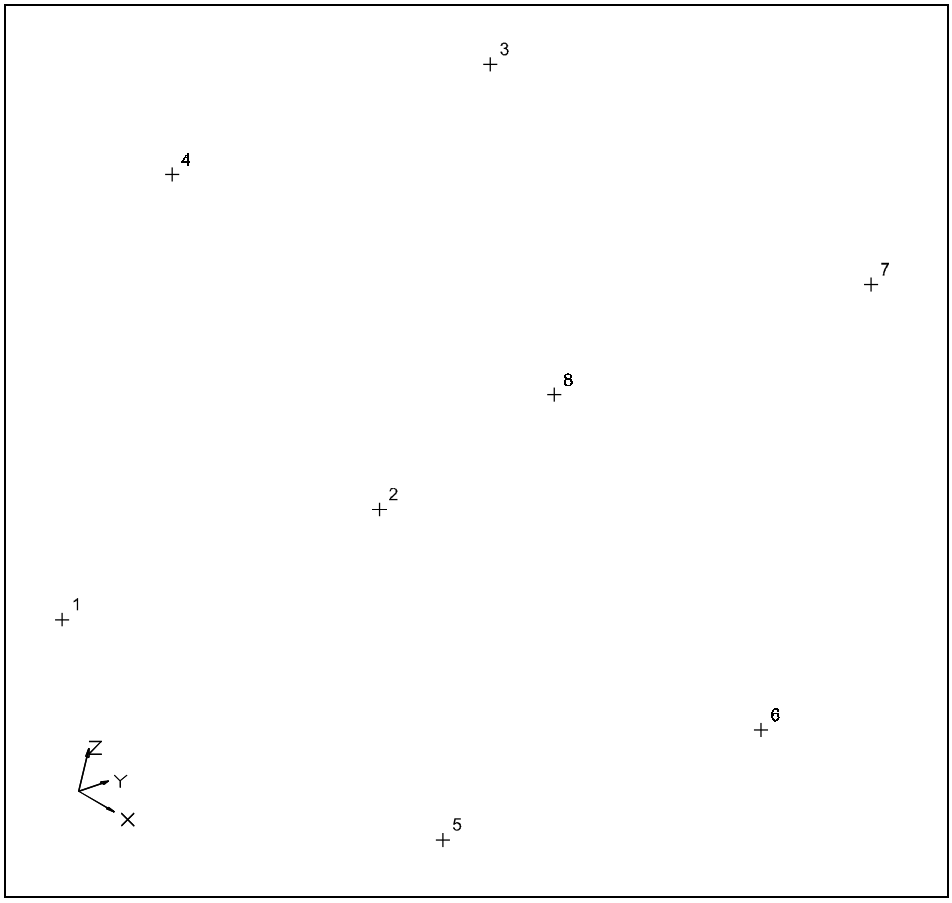
[Create Nodes: Type In]	`[x =] 40`
	`[y =] 40`
	`[z =] 0`
	`[create]`
		Point 6.

[Create Nodes: Type In]	`[x =] 40`
	`[y =] 40`
	`[z =] 40`
	`[create]`
		Point 7.

[Create Nodes: Type In]	`[x =] 40`
	`[y =] 0`
	`[z =] 40`
	`[create]`
		Point 8.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Permanent MENU]	`[f]`	The eight points should appear in the [DISPLAY] window.
[Create Nodes: Type In]	`[return]`	
[Permanent MENU]	`[t]`	View model at an isometric viewing angle.
[True View]	`[thetax =] -60`	
	`[thetay =] 0`	
	`[thetaz =] -60`	
	`[set angles]`	
	`[return]`	
[Permanent MENU]	`[view]`	
	`[save1]`	From now on, the isometric viewing angle can be recalled by selecting [view], then [restore1].
[MENU]	◆ `pg 4`	Proceed to turn node labels on.
[MENU - pg 4]	`[numbers]`	
[Numbers]	`(click on ▲)`	Click on the input collector switch, ▲.
	`[nodes]`	
[Numbers]	`(click on [nodes])`	
[Numbers]	`[all]`	
[Numbers]	`[on]`	
[Numbers]	`[return]`	

Geometric Points Defining The Corners Of The Left Half Of The Beam



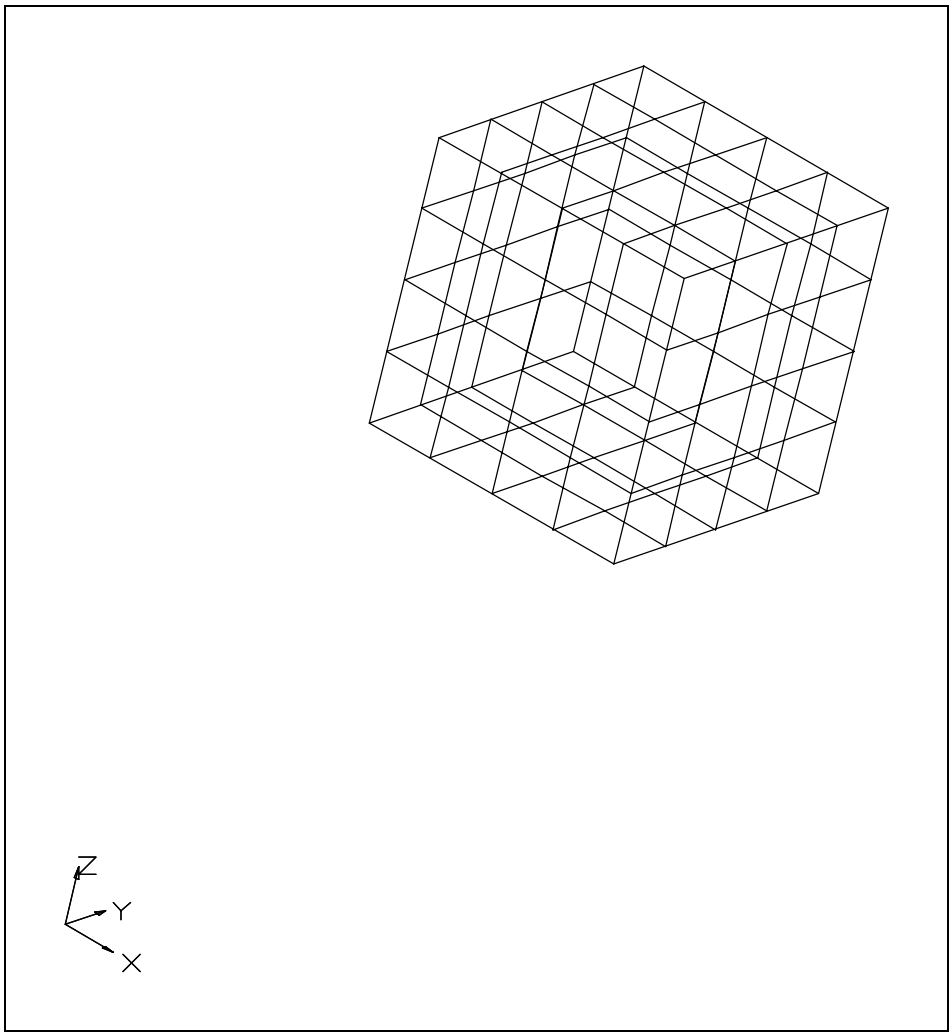
Step 4: Define The Geometric Surfaces

The eight points defined in the previous step can be used to construct the six geometric surfaces.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	◆ `pg 2`	
[MENU - pg 2]	`[spline]`	Define spline surfaces via 4 corner points each.
[Spline Surface/Mesh]	`(click on ▲ in the middle of the screen)` `[surface only]`	Click on the input collector switch, ▲. Create only surfaces, not the mesh also.
[Spline Surface/Mesh]	`(click on ▲ at left edge of screen)` `[nodes]`	Use the corner points to define surfaces.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Spline Surface/Mesh]	`(click on [nodes])`	
[Spline Surface/Mesh]	`[by id]`	
[DISPLAY]	`(select four nodes to define first surface.)`	Surface 1. Nodes 1,2,3,4.
[Spline Surface/Mesh]	`[create]`	
[DISPLAY]	`(select four nodes to define second surface.)`	Surface 2. Nodes 5,6,7,8.
[Spline Surface/Mesh]	`[create]`	
[DISPLAY]	`(select four nodes to define third surface.)`	Surface 3. Nodes 1,5,8,4.
	`[create]`	
[DISPLAY]	`(select four nodes to define fourth surface.)`	Surface 4. Nodes 1,5,6,2.
[Spline Surface/Mesh]	`[create]`	
[DISPLAY]	`(select four nodes to define fifth surface.)`	Surface 5. Nodes 2,6,7,3.
[Spline Surface/Mesh]	`[create]`	
[DISPLAY]	`(select four nodes to define sixth surface.)`	Surface 6. Nodes 4,8,7,3.
[Spline Surface/Mesh]	`[create]`	
[Spline Surface/Mesh]	`[return]`	At this point, six surfaces have been created, one on each face of the cube.

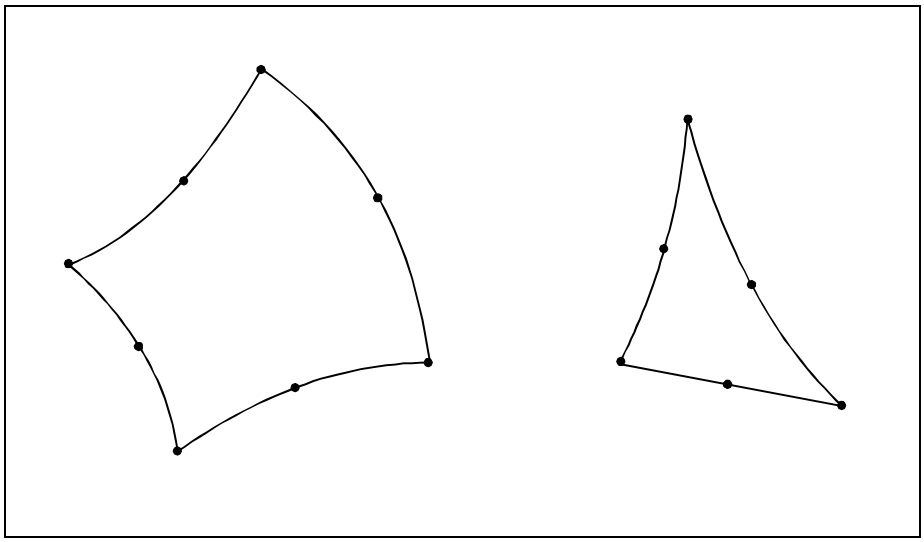
Surfaces On The Six Faces Of The Cube



Step 5: Create A Boundary Element Mesh

The preceding surfaces can be used to generate a boundary element model. Although HYPERMESH does not support boundary elements directly, boundary elements are simulated using 3-D shell elements. These elements do not require properties, such as thickness, orientation, etc., since they are not true shells. The translator step (PREBEST) later converts them into GPBEST boundary elements.

3-D Boundary Elements



As illustrated in this figure, make sure you use 6-noded triangular or 8-noded rectangular shell elements in HYPERMESH. Although 3-noded and 4-noded shell elements are more popular in finite element modeling, it is preferable to use quadratic elements in GPBEST.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Permanent MENU]	`[global]`	
[Global]	`[element order: ]` `(click on ◆ [second])`	Selection changes from [first] to [second]. This will generate quadratic shell elements.
[Global]	`[element size =] 20.`	This will result in four elements per face.
[Global]	`[return]`	
[MENU]	`pg 2`	
[MENU - pg 2]	`[automesh]`	
[Automesh]	`(click on ◆ [multiple surfs])`	
[Automesh]	`(click on [surfs])` `[all]`	All surfaces are to be meshed.
[Automesh]	`(click on ◆ [quads])`	Do not use trias. Use quads.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Automesh]	`[mesh]`	
[Automesh]	`[return]`	
[MENU - pg 3]	`[edges]`	This is the menu that contains the equivalencing command.
[Edges]	`(click on [comps])`	
[Select collectors]	◆ `GMR1`	Select component `GMR1`.
	`[select]`	
[Edges]	`[preview equiv]`	Equivalence coincident nodes.
	`[equivalence]`	
	`[return]`	
[MENU]	◆ `pg 4`	Go to Page 4 of the Menus. This contains the renumbering commands.
[MENU - pg 4]	`[renumber]`	
[Renumber]	`(click on ▲)`	
	`[nodes]`	Renumber nodes only.
[Renumber]	`(click on [nodes])`	
	`(click on [all])`	This selects all nodes.
[Renumber]	`[renumber]`	Renumber nodes. This step is not really necessary. It is performed for the user's convenience. Now nodes are numbered from 1 to 74.
[Renumber]	`[return]`	

Step 6: Define The Groups

The preceding entities can all be put into one group called GMR1. This GMR can be copied to create GMR2. Notice that one result of creating the model this way is that a duplicate set of nodes and elements exist on the interface. However, each GMR contains a unique set of nodes and elements. The POSTBEST translator will handle the interface properly.

Note also that the nodal equivalencing was performed only on the nodes in group GMR1. If the entire model had been created first, we would have to be very selective about equivalencing nodes. The nodes must be equivalenced *on a group-by-group basis*. This is the only way to prevent the nodes on the interface from collapsing during the equivalence operation.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	◆ `pg 1`	Go to Page 1 of the Menus.
[New Model Preferences]	`[collectors]`	Define the second collector, GMR2.
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲	Click on the input collector switch, ▲. Define a component.
	`[comps]`	
[Collectors: Create]	`[material =]`	
[Choose one material]	`[STEEL]`	The material is STEEL.
[Collectors: Create]	`[name =] GMR2`	This is the name of the second collector.
[Collectors: Create]	`[create]`	The collector was created. Now the collector `GMR2` is the current component.
	`[return]`	
[MENU]	◆ `pg 3`	Go to Page 3 of the Menus.
[MENU - pg 3]	`[translate]`	
[Translate]	`(click on` ▲ `)`	Translate along x-axis.
	`[x-axis]`	
[Translate]	`(click on` ◆ `[magnitude =) 40.`	Translate 40*mm* along x-axis..
[Translate]	`(click on` ▲ `)`	Translate and copy elements.
	`[elems]`	
[Translate]	`(click on [elems])`	All elements.
	`[all]`	
[Translate]	`(click on [elems] again)`	
[Translate]	`[duplicate]`	Translate and copy elements.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Translate]	`(click on [current comp])`	New elements will be put into the current component, `GMR2`.
[Translate]	`[translate +]`	
[Permanent MENU]	`[f]`	Replot to screen with fill option.
[Translate]	`[return]`	

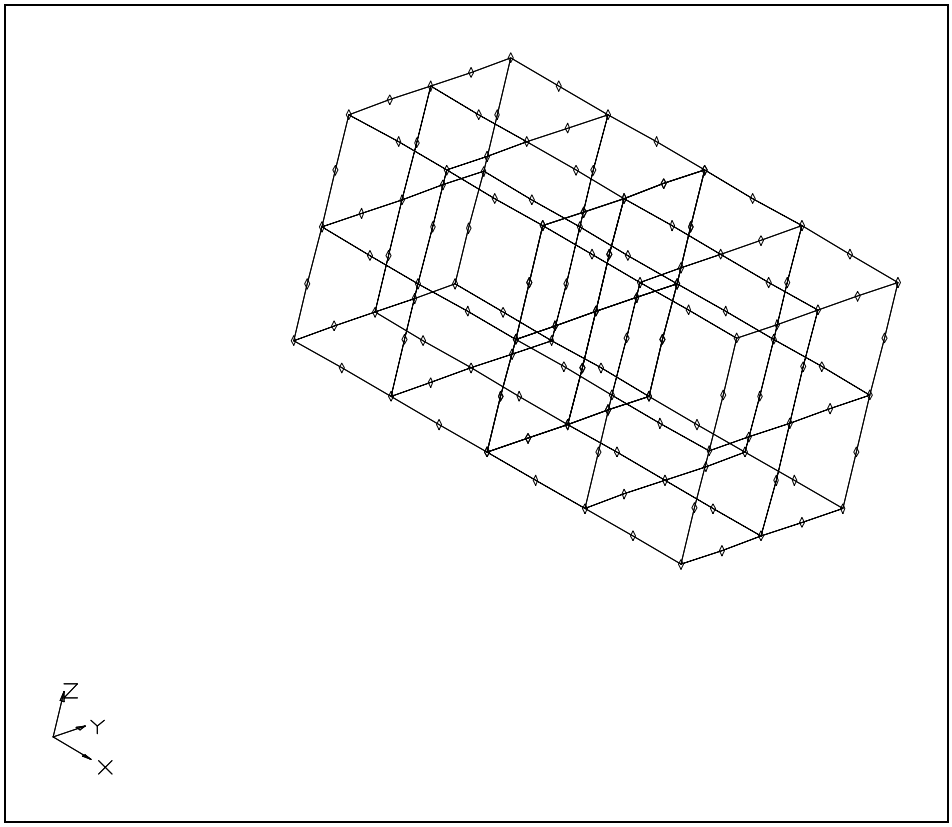
Step 7: Display Boundary Element Mesh

At this point, the model display in HYPERMESH can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Permanent MENU]	`[display]`	Proceed to erase the geometry items from the screen.
[Display]	`(click on ▲ to the left of “elems” or “geoms”)`	
[Display]	`[geoms]`	
[Display]	`[none]`	This turns off the display of geometry items (e.g., lines).
[Permanent Menu]	`[p]`	Only the elements are left on the screen.
[Display]	`[return]`	
[MENU]	`◆ pg 4`	Go to Page 4 of the Menus.
[MENU - pg 4]	`[numbers]`	
[Numbers]	`(click on ▲)`	
[Numbers]	`[nodes]`	Proceed to turn off display of node numbers (if they are showing at this point).
[Numbers]	`(click on [nodes])`	
[Numbers]	`[all]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Numbers]	`[all off]`	Choose all nodes. Nodes and elements are displayed, similar to the following figure, without any entity numbers.
[Numbers]	`[return]`	

Nodes And Boundary Elements For The Entire Model



Step 8: Define The Boundary Conditions For The Model

Now the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X, Y* and *Z* degrees of freedom on the elements at the wall.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual contains an extensive description on specifying GPBEST boundary conditions in HYPERMESH. It is best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 113 is used at the wall since *X, Y* and *Z* displacements are zero there. Load set 101 is used at the tip of the beam since the *X*-traction is non-zero there.

Pay particular attention to the fact that, while in HYPERMESH, 2-D models have the boundary conditions specified on *nodes*, while 3-D models have boundary conditions specified on *elements*. In 3-D models, such as the one here, *element pressure* is used to specify *all* boundary conditions for GPBEST (constraints and tractions). Whether the boundary condition is a traction or a displacement is determined by the load set ID.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Permanent MENU]	`[view]`	It will help to change the view back to the *X-Z* plane. This will make selecting the boundary conditions much easier.
[View]	`[left]`	Move cursor out of the way.
[MENU - pg 4]	`[collectors]`	First, define the load cases applicable for this model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲ `[loadcols]`	Click on the input collector switch, ▲
[Collectors: Create]	`[name =] 113`	Load set 113=zero displacement components in *X*-, *Y*- and *Z*- directions at the left edge of the beam (WALL).
[Collectors: Create]	`[create]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Collectors: Create]	`[return]`	At this point, Load set 113 has been created. In HYPERMESH terminology, it is a “Load Collector” named “113”.
[MENU - pg 4]	`[pressures]`	Proceed to apply the fixed boundary condition at the wall..
[Pressures]	◆ `create`	
[Pressures : Create]	`(click on [elems])`	
	`[by window]`	
[DISPLAY]	`(click mouse to draw a selection box around the four elements at the left edge of the model. You need to click the mouse 4X.)`	
[Processing elements]	`[select entities]`	At this point, elements 25-28 have been selected.
[Pressures Create]	`(click on ▲ [vectors])`	
[Pressures Create]	`(click on [magnitude =]) 999`	This is a dummy magnitude. PREBEST will ignore it, but HYPERMESH requires non-zero pressures.
[Pressures Create]	`(click on ▲ [x-axis])`	Direction is specified by using the Load Set ID 101.
[Pressures Create]	`[create]`	In [DISPLAY] window elements 25-28 are shown with pressures.
[Pressures: Create]	`[return]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 4]	`[collectors]`	Now work on the traction load at the `TIP` of the beam.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Collectors]	◆ `create`	
[Collectors: Create]	`Collector type:` ▲ `[loadcols]`	Click on the input collector switch, ▲
[Collectors: Create]	`[name =] 101`	Load set 101=non-zero traction.
[Collectors: Create]	`[create]`	
[Collectors: Create]	`[return]`	At this point, Load set 101 has been created. In HYPERMESH terminology, it is a “Load Collector” named “101”.
[MENU - pg 4]	`[pressures]`	Proceed to apply the traction boundary condition at the `TIP`.
[Pressures]	◆ `create`	
[Pressures : Create]	`(click on [elems])`	
[Pressures: Create]	`[by window]`	
[DISPLAY]	`(click mouse to draw a selection box around the four elements at the right edge of the model. You need to click the mouse 4X.)`	
[Processing elements]	`[select entities]`	At this point, elements 53-56 have been selected.
[Pressures Create]	`(click on` ▲ `[vectors])`	
[Pressures Create]	`(click on [magnitude =]) 1.25`	Here, the traction magnitude is specified as 1.25 N/mm^2. Direction is specified by using the Load Set ID 101.
[Pressures Create]	`(click on` ▲ `[x-axis])`	The PREBEST translator will ignore this. Direction is specified by using the Load Set ID 101.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Pressures Create]	`[create]`	In [DISPLAY] window, elements 53-56 are shown with pressures..
[Pressures: Create]	`[return]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU - pg 4]	◆ `pg 5`	
[MENU - pg 5]	`[export data]`	
[Export Data]	`template = /... /feoutput/hm/general` `filename = tut3d.hma`	NOTE: Do not use a NASTRAN template by accident. This creates a HYPERMESH ASCII file file named `tut3d.hma.`
[Export Data]	`[write]`	
[Export Data]	`[return]`	
[MENU - pg 5]	`[files]`	
[Files]	`file = tut3d.hm`	
[Files]	`[save]`	
[Files]	`[yes]`	OK to overwrite.
[Files]	`[return]`	
[MENU - pg 5]	`[quit]`	Exit from HYPERMESH.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 9: Run GPBEST To Analyze The Boundary Element Model

At this point, the HYPERMESH (ASCII) neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`6 <CR>`	HYPERMESH
[PREBEST]	`Tut3d.hma <CR>`	HYPERMESH (ASCII) neutral file name.
[PREBEST]	`Y <CR>`	“Yes.” This is the correct file.
[PREBEST]	`tut3d.dat <CR>`	Write to `tut3d.dat` file (default).
[PREBEST]	`1 <CR>`	3-D model.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`3 <CR>`	High level of quality on element checks.
[PREBEST]	`1 <CR>`	Since there are two GMRs in this model, PREBEST is asking whether you want to have the same functional variation in both GMRs.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`1 <CR>`	Use a bonded interface. This acts as though the two GMRs were welded at their interface.
[PREBEST]	`N <CR>`	No body forces.
		The PREBEST program ends, but the file `tut3d.dat` has been created. It can be run directly in GPBEST.
		Note that it is sometimes useful to examine the `hypbest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining whether or not the model was completely translated.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**`< tut3d.dat best-s > tut3d.res <CR>`**
[UNIX]	**`mv NEUTRAL.OUT tut3d.neu`**

Step 10: Post-Processing The Results

You can view the GPBEST results file (`tut3d.res`) and determine if GPBEST ran successfully by using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, you can proceed to view the results in HYPERMESH. For this step, it is best to start HYPERMESH with a new database.

Since HYPERMESH cannot read in the GPBEST neutral file directly, the POSTBEST translator must be run in order to create files that HYPERMESH will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`postbest`**`<CR>`	
[POSTBEST]	`6 <CR>`	HYPERMESH
[POSTBEST]	`tut3d.neu <CR>`	Name of GPBEST neutral file.

[POSTBEST]	`N <CR>`	Do not renumber the nodes.
		Three files are created:
		`HYPER.GEOM`
		`PATRAN.dis.1`
		`PATRAN.nod.1`

To post-process the model in HYPERMESH, read in the HYPERMESH ASCII neutral file (`HYPER.GEOM`) and the HYPERMESH results file (`tut3dd.res`) as demonstrated in Tutorial 2 (Chapter 4). The intermediate `hmpatran` step must be performed in order to convert PATRAN results into HYPERMESH result file format.

FEMAP Model

TIP: This tutorial covers FEMAP 5.0 or higher.

Step 1: Start FEMAP

In this step, you will start the FEMAP program and name the database you will be using.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Windows]	**`Femap`**	Start FEMAP.
[MENU]	`File`	
	`Save As:…`	
[File Save As]	`tut3e [SAVE]`	This will name the database `tut3e.mod`.

Step 2: Create a Volume Defining a Cube

The method of creating this beam model will be to create the first GMR (i.e., group), then create the second GMR. To create the first GMR, we will define a geometric cube. In the process of creating a volume, FEMAP also creates the six surfaces, which can be used for meshing. The first GMR will be meshed. The second GMR will be created from the first GMR using FEMAP groups.

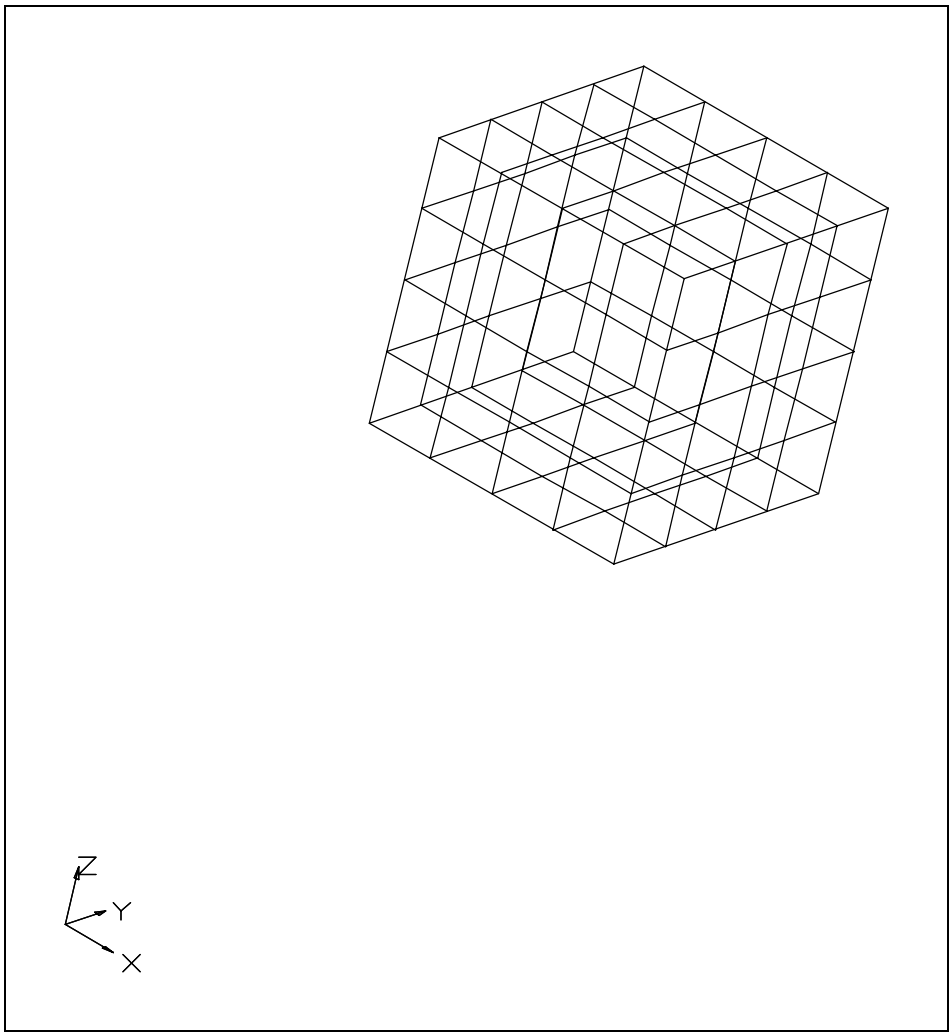
INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Geometry`	
	`Volume`	
	`Corners`	
[Volume]	`[Methods^]`	
[Volume]	`Locate`	
[Locate]	*x*=0	Point 1.
	y=0	
	z=0 [OK]	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Locate]	*x*=40	Point 2.
	y=0	
	z=0 [OK]	
[Locate]	*x*=40	Point 3.
	y=40	
	z=0 [OK]	
[Locate]	*x*=0	Point 4.
	y=40	
	z=0 [OK]	
[Locate]	*x*=0	Point 5.
	y=0	
	z=40 [OK]	
[Locate]	*x*=40	Point 6.
	y=0	
	z=40 [OK]	
[Locate]	*x*=40	Point 7.
	y=40	
	z=40 [OK]	
[Locate]	*x*=0	Point 8.
	y=40	
	z=40 [OK] [Cancel]	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	View	
	Rotate	
[View Rotate]	● Model Axes	
	X=-60	
	Y=0	
	Z=-60	
	[OK]	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`View` `AutoScale`	The cube should appear in the [DISPLAY] window. In addition to the volume, FEMAP also created six surfaces.

Surfaces On The Six Faces Of The Cube



Step 3: Define Material Properties

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Model` `Material`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Define Isotropic Material]	`ID: 1` `Title: STEEL`	
[Define Isotropic Material]	`Young's Modulus:207000.`	Young's modulus $\left(\frac{N}{mm^2}\right)$.
[Define Isotropic Material]	`Poisson Ratio: 0.3`	
[Define Isotropic Material]	`Mass Density: 7.833E-9`	Mass density of steel $\left(\frac{N\,s^2}{mm^4}\right)$. Notice that units are chosen consistent with dimensions of the model, which are mm.
[Define Isotropic Material]	`[OK]  [Cancel]`	
[MENU]	`Model` `Property`	Proceed to define a dummy element property. FEMAP will not create a beam without an element property definition.
[Define Property]	`[Elem/Property Type…]`	
[Element/Property Type]	● `Plate`] `Parabolic Elements` `[OK]`	

[Define Property – Plate Element Type]	`ID: 1`
	`Title: Dummy Plate`
	`Material: [1..STEEL]`	

[Define Property – Plate Element Type]	`[OK]  [Cancel]`	

Step 4: Create A Boundary Element Mesh

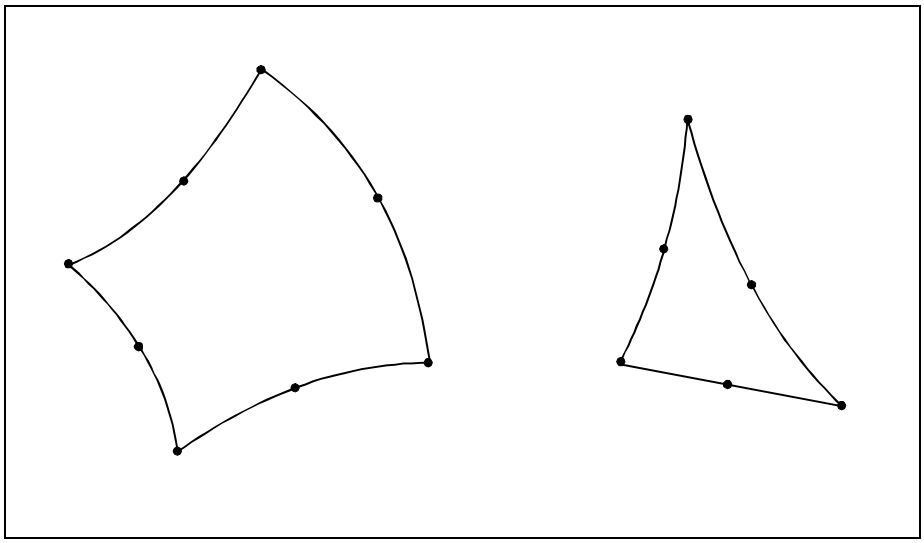
The preceding surfaces can be used to generate a boundary element model. Although FEMAP does not support boundary

elements directly, boundary elements are simulated using 3-D shell elements (or plates). These elements do not require properties, such as thickness, orientation, etc., since they are not true shells. The translator step (PREBEST) later converts them into GPBEST boundary elements.

Notice For Users of FEMAP

Although plate properties are not required to define the boundary element model, FEMAP requires properties and materials to be defined before the mesh can be generated. The FEMAP user must define dummy plate element properties.

3-D Boundary Elements



As illustrated in this figure, make sure you use 6-noded triangular or 8-noded rectangular shell elements in FEMAP. Although 3-noded and 4-noded shell elements are more popular in finite element modeling, it is preferable to use quadratic elements in GPBEST.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Mesh`
	`Mesh Control`
	`Default Size`
		Proceed to create the boundary elements.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[Default Mesh Size]	`Size: 20`
	`Min Elem: 1`
	`[OK]`	

[MENU]	`Mesh`
	`Geometry`
	`Surface`	

[Entity Selection – Select Surfaces to Mesh]	`[Select All]` `[OK]`	

[Geometry Mesh Options]	`Property: 1.Dummy Plate`
	`Generate: ● Elements & Nodes`
	`Element Shape: ● Quad`
	`[OK]`	

[MENU]	`Delete`
	`Model`
	`Node`
		Some extra nodes are sometimes generated. Delete any unconnected nodes.

[Entity Selection – Select Nodes]	`[Select All]` `[OK]`	
[FEMAP]	`[Yes]`	FEMAP will not allow you to delete nodes if they are attached to elements. Thus, this procedure deletes only unconnected nodes.

[MENU]	`Tools`
	`Check`
	`Coincident Nodes`
		Equivalence coincident nodes.

[Entity Selection – Select Nodes to Check]	`[Select All] [OK]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[FEMAP]	`[No]`	Do not specify additional range of nodes to merge.

[Check/Merge Coincident]	`Maximum Distance: 0.005`
	`] Merge Coincident Entities`
	`[OK]`	

[MENU]	`Modify`
	`Renumber`
	`Node`
		Renumbering is not required. It is merely done for the user's convenience.

[Entity Selection – Select Nodes to Renumber]	`[Select All]` `[OK] [OK]`	

Step 5: Define The Groups

The preceding entities can all be put into one group called GMR1. This GMR can be copied to create GMR2. Notice that one result of creating the model this way is that a duplicate set of nodes and elements exist on the interface. However, each GMR contains a unique set of nodes and elements. The PREBEST translator will handle the interface properly.

Note also that the nodal equivalencing was performed only on the nodes in group GMR1. If the entire model had been created first, we would have to be very selective about equivalencing nodes. The nodes must be equivalenced *on a group-by-group basis*. This is the only way to prevent the nodes on the interface from collapsing during the equivalence operation.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`Group` `Set`	

[Create or Activate Group]	`ID: 1`
	`Title: GMR1`
	`[OK]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Group`
	`Node`
	`ID`
		Place all nodes in group GMR1.

[Entity Selection – Select Nodes]	`[Select All]` `[OK]`	

[MENU]	`Group`
	`Element`
	`ID`
		Place all elements in group GMR1.

[Entity Selection – Select Elements]	`[Select All]` `[OK]`	
[MENU]	`Group` `Set`	

[Create or Activate Group]	`ID: 2`
	`Title: GMR2`
	`[OK]`	

[MENU]	`Group`
	`Operations`
	`Automatic Add…`
		When new entities are subsequently created (via the COPY command), they will be automatically placed in group GMR2.

[Automatically Add to Group]	● `Active` `[OK]`	

[MENU]	`Mesh`
	`Copy`
	`Element`
		Copying the elements will also copy the nodes connected to them.

[Entity Selection – Select Elements to Copy]	`[Select All]` `[OK]`	
[Generation Options]	`[OK]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Vector]	`[Methods^]` `Global Axes`	Proceed to create a second group (GMR2), obtained by copying group GMR1 and translating 40 mm in the *X*-direction.

[Vector Global Axis – Select Vector to Copy Along]	`Base: X=0 Y=0 Z=0`
	● `Positive`
	● `X Axis`
	`Length: 40.0`
	`[OK]`	

[MENU]	`View` `Autoscale`	GMR2 has been created.

Step 6: Display Boundary Element Mesh

At this point, the model display in FEMAP can be manipulated. It is a good idea to check the model, connectivity, etc.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Group`
	`Operations`
	`Automatic Add…`	

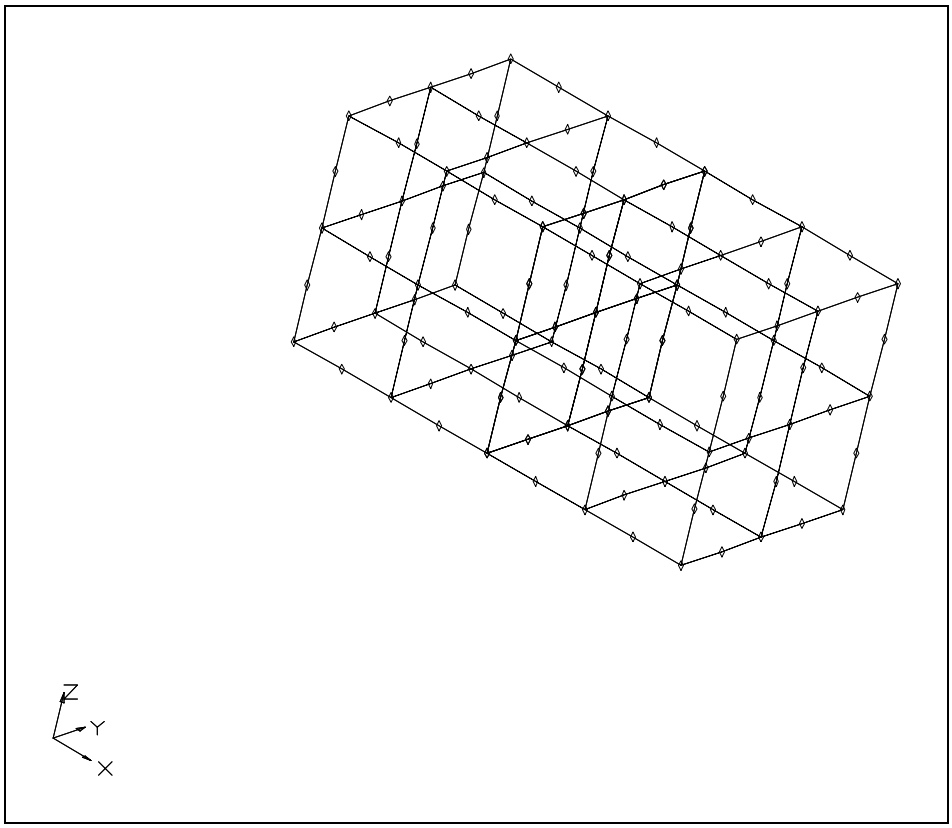
[Automatically Add to Group]	● `None` `[OK]`	Turn this option OFF.

[MENU]	`View`
	`Options`
	`[Quick Options]`	

[View Quick Options]	`[All Entities Off]` `[Labels Off]`	The geometric entities no longer need to be displayed.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[View Quick Options]	] `Node`	
	] `Element`	
	] `Load - Pressure`	
[View Quick Options]	`[Done]`	
[View Options]	`[OK]`	The model, appears in the [DISPLAY] window.
[MENU]	`Group`	
	`Set`	
[Create or Activate Group]	`[Reset]`	Neither group is now “current.”

Nodes And Boundary Elements For The Entire Model



Step 7: Define The Boundary Conditions For The Model

Now the boundary conditions must be specified. There are two boundary conditions for this model. First, the wall is completely fixed. This requires constraining the *X, Y* and *Z* degrees of freedom on the elements at the wall.

Also, there is a distributed load on the right edge of the beam. The total load there is 2000 N. However, it is specified as a traction (distributed load) in GPBEST.

Chapter 9 of the BESTVIEW manual contains an extensive description on specifying GPBEST boundary conditions in FEMAP. It is best to refer to it while performing the following steps. *GPBEST boundary conditions are not specified in the same way as finite element boundary conditions.* In particular, note that the boundary conditions and loads are specified with specific load set IDs. For example, load set 113 is used at the wall since *X, Y* and *Z* displacements are zero there. Load set 101 is used at the tip of the beam since the *X*-traction is non-zero there.

In 3-D models, such as the one here, *element pressure* is used in FEMAP to specify *all* boundary conditions for GPBEST. Whether the boundary condition is a traction or a displacement is determined by the Load set ID.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`View` `Rotate`	It will help to change the view back to the *X-Z* plane. This will make selecting the boundary conditions much easier.
[View Rotate]	`[ZX Front]` `[OK]`	

[MENU]	`Model`
	`Load`
	`Set`
		First, define the load case for the left edge (WALL) of the model. Refer to the table in Chapter 9 of the BESTVIEW manual for a listing of the appropriate load set numbers.

[Create or Activate Load Set]	`ID: 113`
	`Title: WALL`
	`[OK]`
		This creates Load case 113 AND makes it the current load set.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Model`
	`Load`
	`Elemental`	

[Entity Selection – Enter Elements to Select]	`(Hold down the <shift> key and draw a box around the left edge of the model)` `[OK]`	This should select the four elements (21-24) that make up the WALL.

[Create Loads on Elements]	`[Pressure]`
	`Pressure: 999.`
	`[OK]`
		These are just dummy loads. In reality, Load set 113 is a constraint.

[Face Selection for Elemental Loads]	`Face = 1` `[OK]`	This selects face 1. You should always select “face 1”.
[Entity Selection – Enter Elements to Select]	`[Cancel]`	If you want to display the distributed loads, hit <CTRL><Q> to bring up the Quick Options Window, then check “Load – Pressure”.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

[MENU]	`Model`
	`Load`
	`Set`
		Now work on the traction load at the `TIP` of the beam.

[Create or Activate Load Set]	`ID: 101`
	`Title: TIP`
	`[OK]`
		This creates Load case 101 AND makes it the current load set.
		Load Set 101 is non-zero traction.

[MENU]	`Model`
	`Load`
	`Elemental`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[Entity Selection – Enter Elements to Select]	`(Hold down the <shift> key and draw a box around the right edge of the model)` `[OK]`	This should select the four elements (37-40) that make up the TIP.

[Create Loads on Elements]	`[Pressure]`
	`Pressure: 1.25`
	`[OK]`
		Here, the traction magnitude is specified as 1.25 N/mm^2. Direction is specified by using the Load Set ID 101.

[Face Selection for Elemental Loads]	`Face = 1` `[OK]`	This selects face 1. You should always select “face 1”.
[Entity Selection – Enter Elements to Select]	`[Cancel]`	

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[MENU]	`File` `Save`	

[MENU]	`File`
	`Export`
	`FEMAP Neutral . . .`	

[Write Model To FEMAP Neutral]	`File Name: tut3e.fem` `[Write]`	This creates a FEMAP neutral file named `tut3e.fem`.
[Neutral File Write Options]	] `Write Analysis Model` ● `Full Model` `[OK]`	Other options may be checked in addition to these.

[Warnings]	`File`
	`Exit`
	`[Yes]`
		Save file before exiting.

TIP: Many of the default responses to PREBEST questions can be accepted by merely hitting the <CR>. For more information on PREBEST, refer to Chapter 9 in the BESTVIEW User's Manual.

Step 8: Run GPBEST To Analyze The Boundary Element Model

At this point, the FEMAP neutral file has been created. However, it must be translated into GPBEST format using PREBEST.

After the GPBEST data file is created, proceed to run the analysis. It should only take a few seconds to complete. After the program completes running, rename the neutral file.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**`prebest`**	
[PREBEST]	`7 <CR>`	FEMAP.
[PREBEST]	`tut3e.fem <CR>`	FEMAP neutral file name.
[PREBEST]	`tut3e.dat <CR>`	Write to `tut3e.dat` file (default).
[PREBEST]	`1 <CR>`	3-D model.
[PREBEST]	`1 <CR>`	Elastic analysis.
[PREBEST]	`N <CR>`	No exterior region.
[PREBEST]	`N <CR>`	No symmetry.
[PREBEST]	`3 <CR>`	High level of quality on element checks.
[PREBEST]	`1 <CR>`	Since there are two GMRs in this model, PREBEST is asking whether you want to have the same functional variation in both GMRs.
[PREBEST]	`2 <CR>`	Quadratic elements.
[PREBEST]	`1 <CR>`	Use a bonded interface. This acts as though the two GMRs were welded at their interface.
[PREBEST]	`N <CR>`	No body forces.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
		The PREBEST program ends, but the file `tut3e.dat` has been created. It can be run directly in GPBEST. Note that it is sometimes useful to examine the `fembest.log` file. It contains error messages, area and volume calculations, and problem statistics. It can be helpful in determining whether or not the model was completely translated.

INPUT FOCUS	*COMMAND OR DATA ENTRY*
[UNIX]	**< tut3e.dat best-s > tut3e.res <CR>**
[UNIX]	**mv NEUTRAL.OUT tut3e.neu**

Step 9: Post-Processing The Results

You can view the GPBEST results file (`tut3e.res`) and determine if GPBEST ran successfully by using some of the suggestions given in Chapter 3 of this tutorial. If it did run successfully, you can proceed to view the results in FEMAP. For this step, it is best to start FEMAP with a new database.

Since FEMAP cannot read in the GPBEST neutral file directly, the POSTBEST translator must be run in order to create files that FEMAP will recognize.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*
[UNIX]	**postbest**<CR>	
[POSTBEST]	7 <CR>	FEMAP.
[POSTBEST]	tut3e.neu <CR>	Name of GPBEST neutral file.
[POSTBEST]	N <CR>	Do not renumber the nodes.

INPUT FOCUS	*COMMAND OR DATA ENTRY*	*COMMENTS*

		Three files are created:
		`FEMAP.GEOM`
		`patran.dis.1`
		`patran.els.1`
		`patran.nod.1`

To post-process the model in FEMAP, read in the FEMAP neutral file (`FEMAP.GEOM`) and the FEMAP results files (`patran.nod.1`, `etc.`), which are in PATRAN ASCII format, as demonstrated in Tutorial 2 (Chapter 4).

CHAPTER 6 - ADDITIONAL HELPFUL INFORMATION

Introduction

At this point, you should be able to build basic boundary element models, run them in GPBEST, and view the results in a graphics program. However, there are still some important things that the new user needs to know in order to effectively interact with the GPBEST system. These miscellaneous topics are the subject of this chapter.

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, etc.) 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 Chapter 3, 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. By doing this, the new user will become familiar with the GPBEST input file and Section 5 of the GPBEST User's Manual. Only after this has been accomplished should a translation of modeling geometry from a CAD system to GPBEST be undertaken.

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:

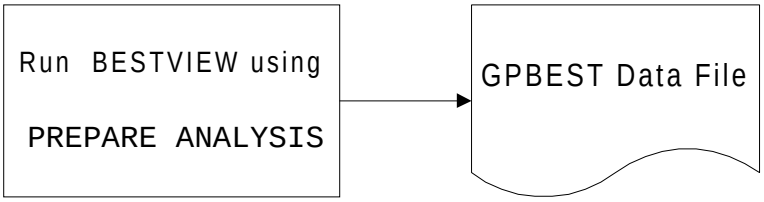
1. To display boundary element models and results of GPBEST analyses.
2. To interface with other CAD programs.
3. To apply boundary conditions and loads to boundary element models, whether they have been created in BESTVIEW or not.

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. An important reason to use BESTVIEW for applying loads and boundary conditions is the fact that most options for GPBEST are supported.

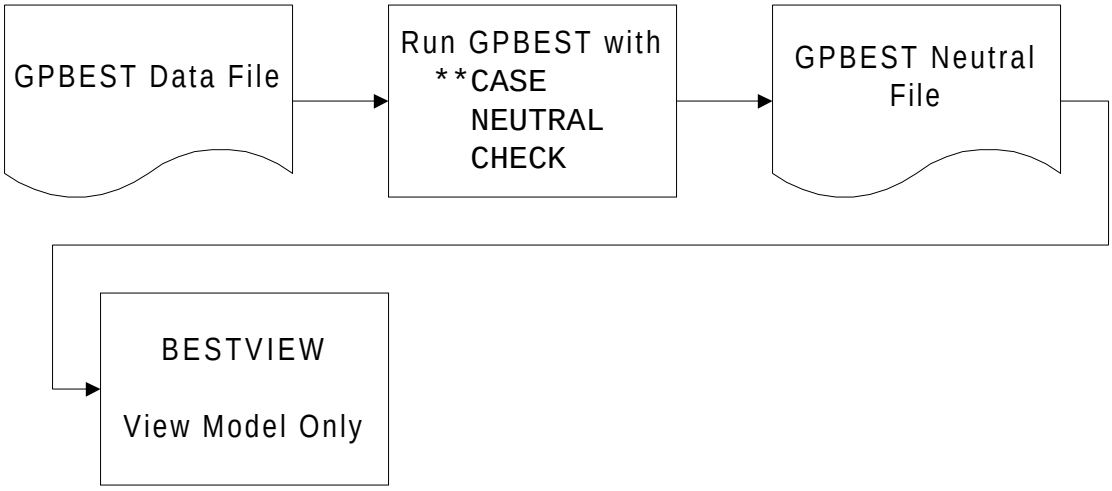
An important aspect of BESTVIEW is that it is designed to work along with other CAD programs while not necessarily replacing them. Thus, a typical GPBEST user could use PATRAN, for example, to model the part geometry and build the boundary element mesh, then transfer the boundary element model into BESTVIEW 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 be viewed either in BESTVIEW or PATRAN.

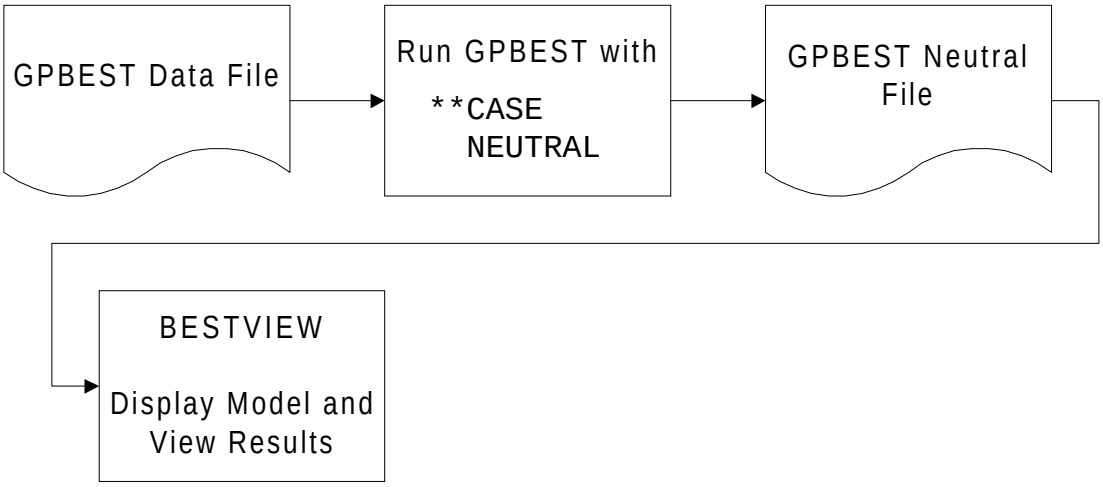
If BESTVIEW is used to create a boundary element model, then it can write out a GPBEST data file using the "Prepare Analysis" option:



If you have a GPBEST data file and would like to view the model in BESTVIEW, then you must run a quick GPBEST "CHECK" 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 (Volume II). New users of BESTVIEW can effectively start with the tutorials in this manual and with Section 1.6, “Quick Start Section,” and Chapter 8, “A Tutorial on BESTVIEW,” both in the BESTVIEW User’s Manual (Volume II).

PATRAN

TIP: For further information on PATRAN, consult Chapter 9 in the BESTVIEW User's Manual (Volume II).

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.

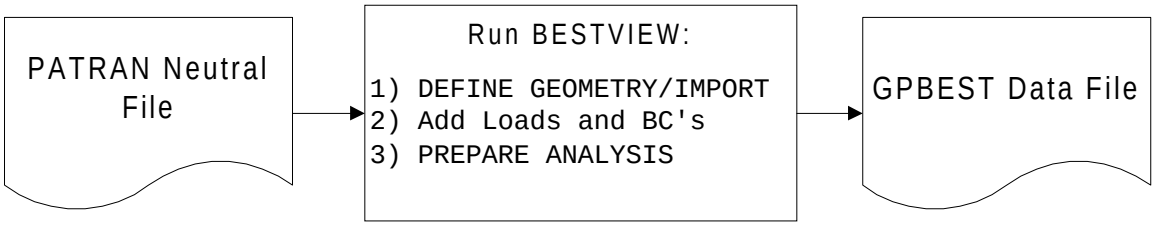
Support for GPBEST is provided through two PATRAN translators:

1. PREBEST - to go from PATRAN to GPBEST
2. POSTBEST - to go from GPBEST to PATRAN

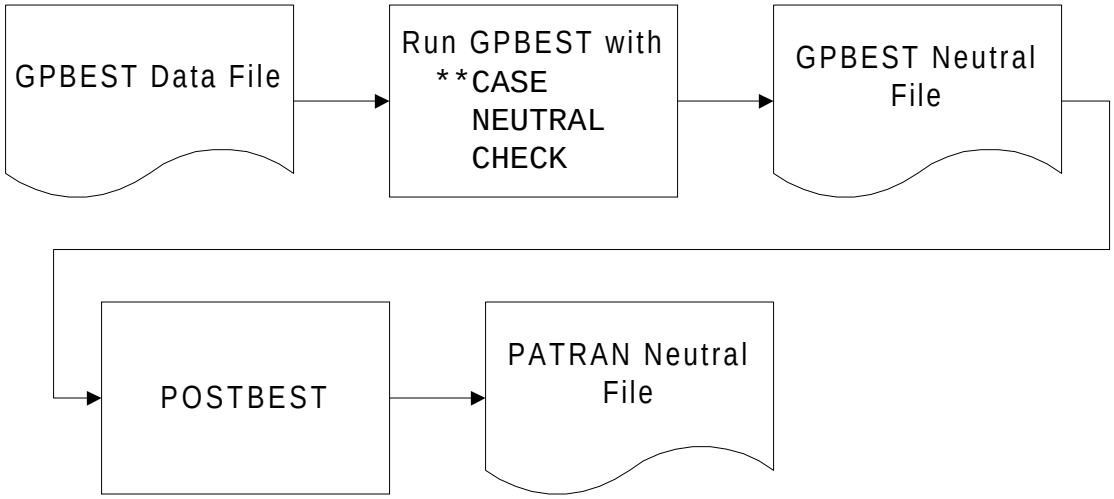
PATRAN can be used to create part geometry, generate a boundary element surface mesh, apply boundary conditions, and view results of a GPBEST analysis.

If a user creates the part geometry and 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:

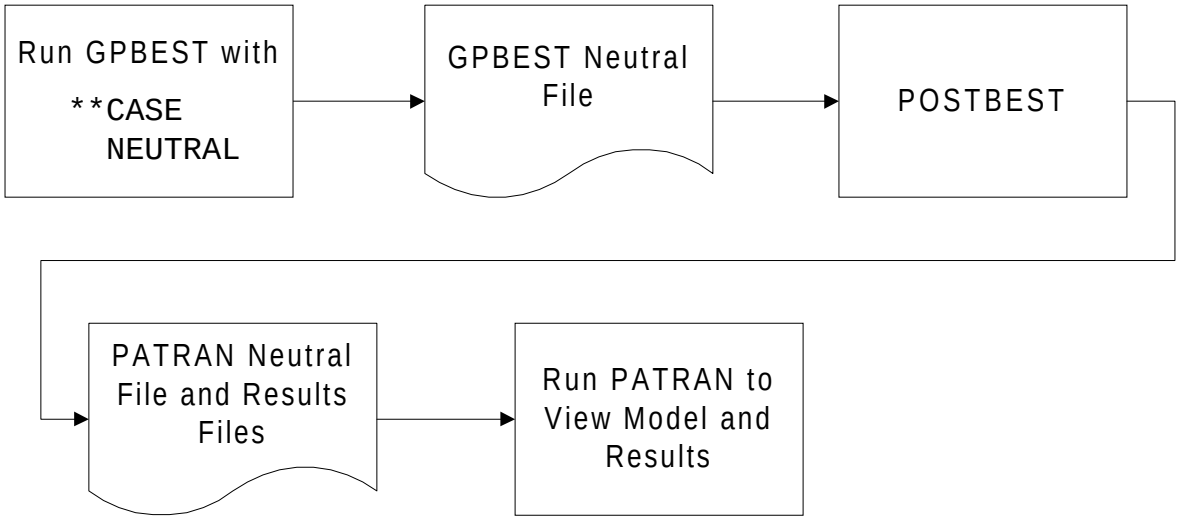
If PATRAN is used only to generate the boundary element model but omits loads and boundary conditions, then BESTVIEW can be used to add these and write out the GPBEST input file:



If the user has a GPBEST data file and wants to view only 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 can be translated by POSTBEST 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.

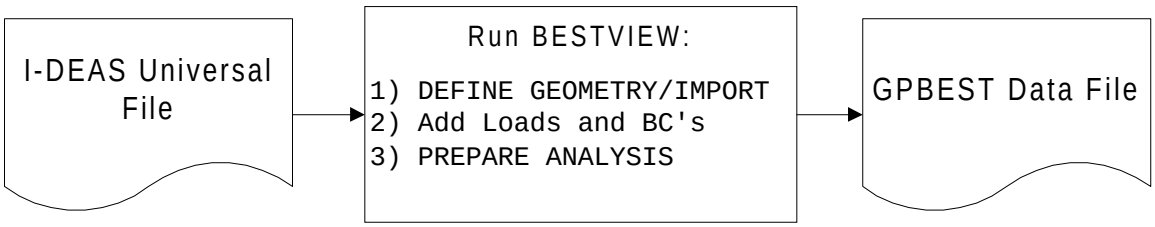
I-DEAS

> ***TIP:*** For further information on I-DEAS, consult Chapter 9 in the BESTVIEW User's Manual (Volume II).

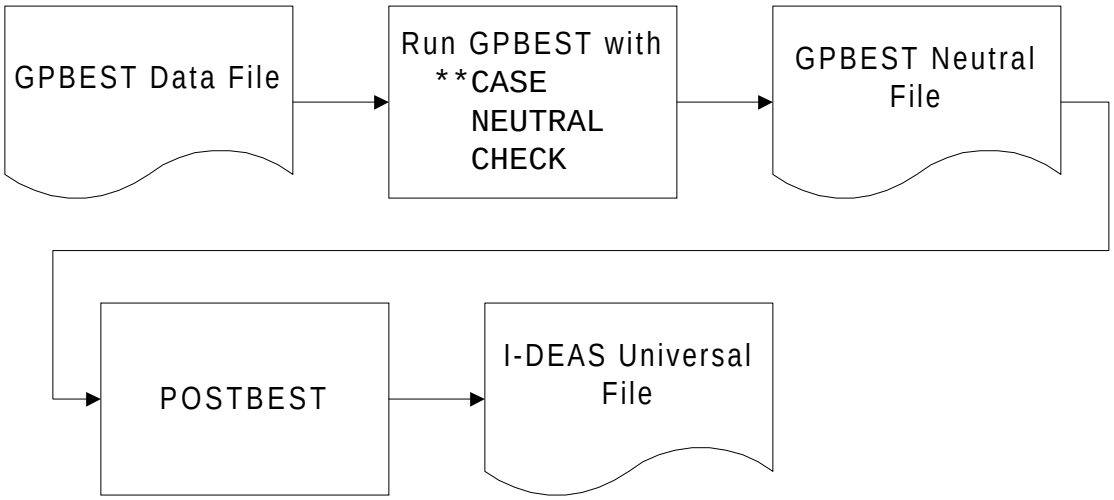
Much of what was stated in the previous section regarding PATRAN applies to I-DEAS. Both PATRAN and I-DEAS are fully supported in PREBEST and POSTBEST.

If a user creates the part geometry and boundary element mesh and applies loads and boundary conditions in I-DEAS, then the GPBEST input file is translated directly by PREBEST:

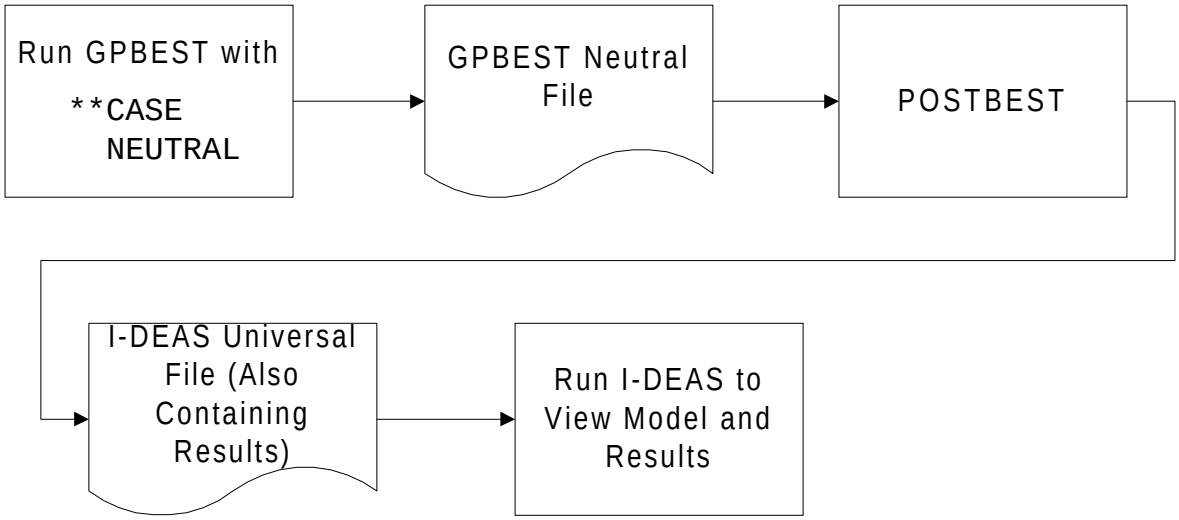
If I-DEAS is used only to generate the boundary element model but loads and boundary conditions are left out, then BESTVIEW can be used to add loads and boundary conditions and write out the GPBEST input file:



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, then 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.

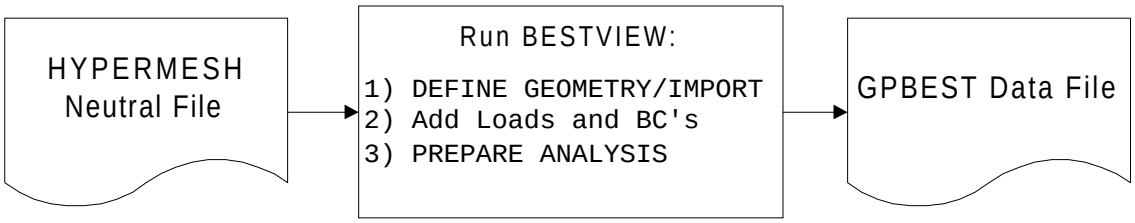
HYPERMESH

> ***TIP:*** For further information on HYPERMESH, consult Chapter 9 in the BESTVIEW User's Manual (Volume II).

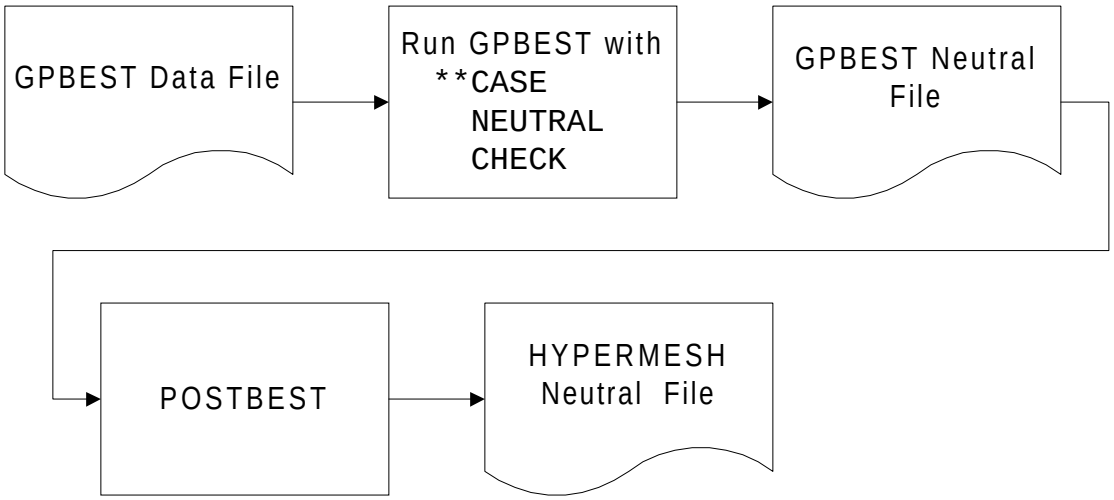
Much of what was stated in the previous section regarding PATRAN applies to HYPERMESH. The HYPERMESH program is fully supported by the PREBEST and POSTBEST translators.

If a user creates the part geometry and boundary element mesh and applies loads and boundary conditions in HYPERMESH, then the GPBEST input file is translated directly by PREBEST:

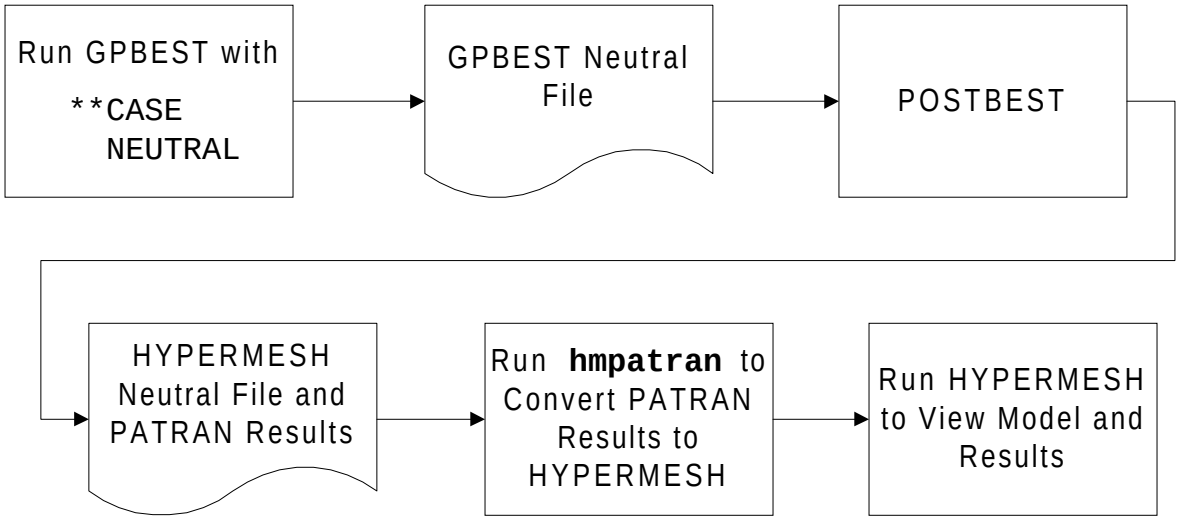
If HYPERMESH is used only to generate the boundary element model but loads and boundary conditions are left out, then BESTVIEW can be used to add loads and boundary conditions and write out the GPBEST input file:



If the user has a GPBEST data file and wants to view the model in HYPERMESH, POSTBEST can be used to generate a HYPERMESH neutral file:



Finally, if the user already has a GPBEST data file and wants to view the model and results in HYPERMESH, then POSTBEST can be used to create a HYPERMESH neutral file and a HYPERMESH results file:



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.

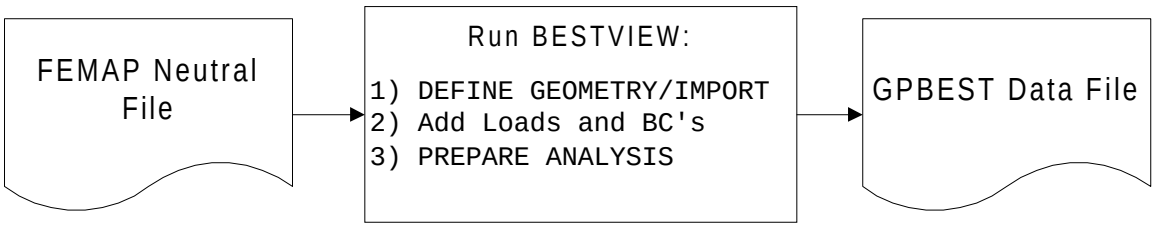
FEMAP

TIP: For further information on FEMAP, consult Chapter 9 in the BESTVIEW User's Manual (Volume II).

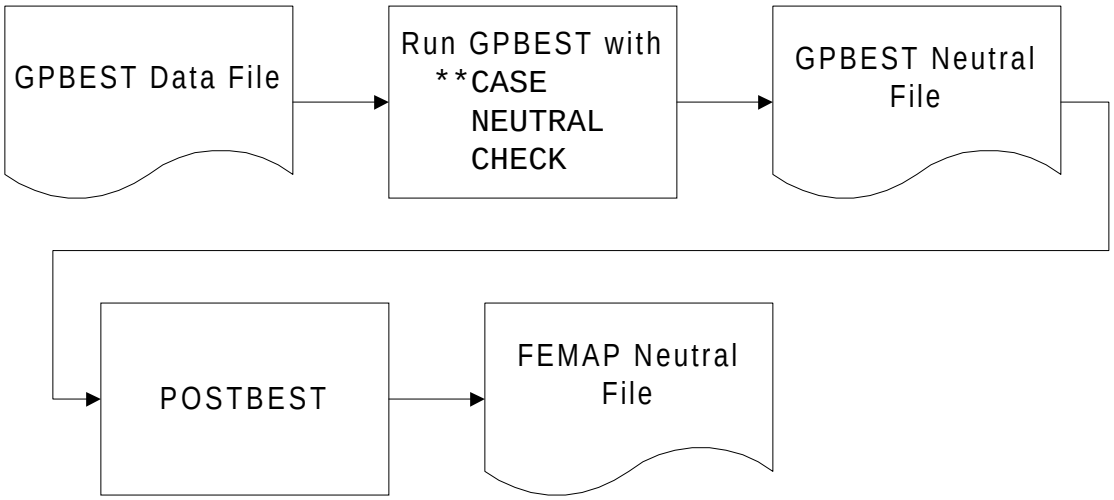
Much of what was stated in the previous section regarding PATRAN applies to FEMAP. Both PATRAN and FEMAP are fully supported in PREBEST and POSTBEST.

If a user creates the part geometry and boundary element mesh and applies loads and boundary conditions in FEMAP, then the GPBEST input file is translated directly by PREBEST:

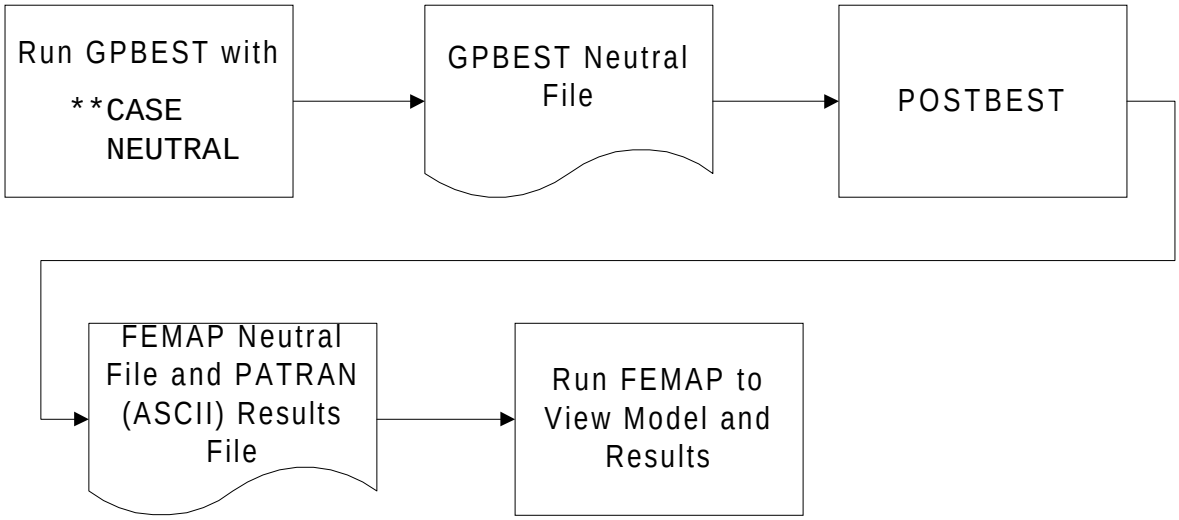
If FEMAP is used only to generate the boundary element model but loads and boundary conditions are left out, then BESTVIEW can be used to add loads and boundary conditions and write out the GPBEST input file:



If the user has a GPBEST data file and wants to view the model in FEMAP, POSTBEST can be used to generate an FEMAP neutral file:



Finally, if the user already has a GPBEST data file and wants to view the model and results in FEMAP, then POSTBEST can be used to create a FEMAP neutral file and PATRAN (ASCII) results files, which can be imported into FEMAP for post-processing:



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.

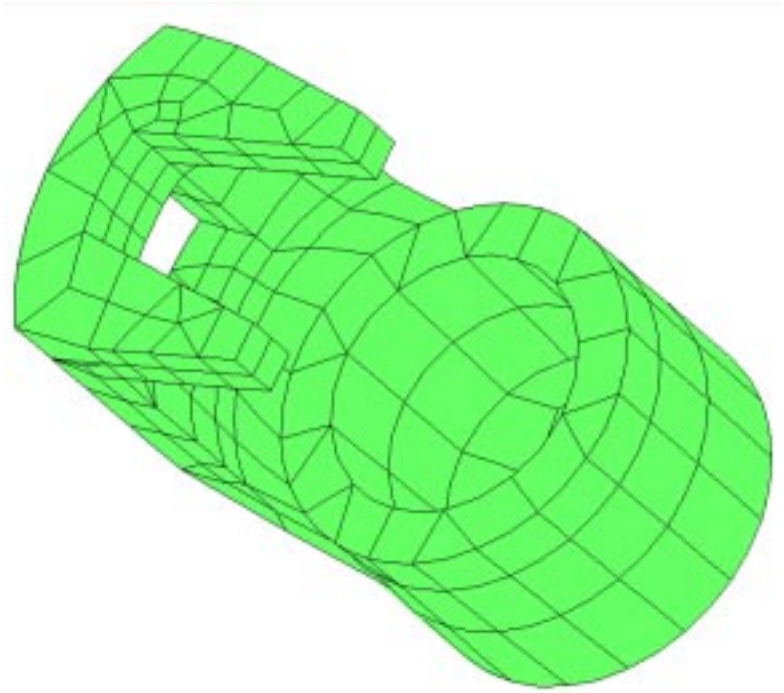
PRO/Engineer

PRO/Engineer has become extremely popular due to the broad range of programs that interface with it because of its extensive capabilities. PRO/Engineer concentrates on 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. Solid modeling 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.

Example of a GPBEST Surface Model Which Was *Automatically* Meshed Over the Solid Model by PRO/Engineer



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 need 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.

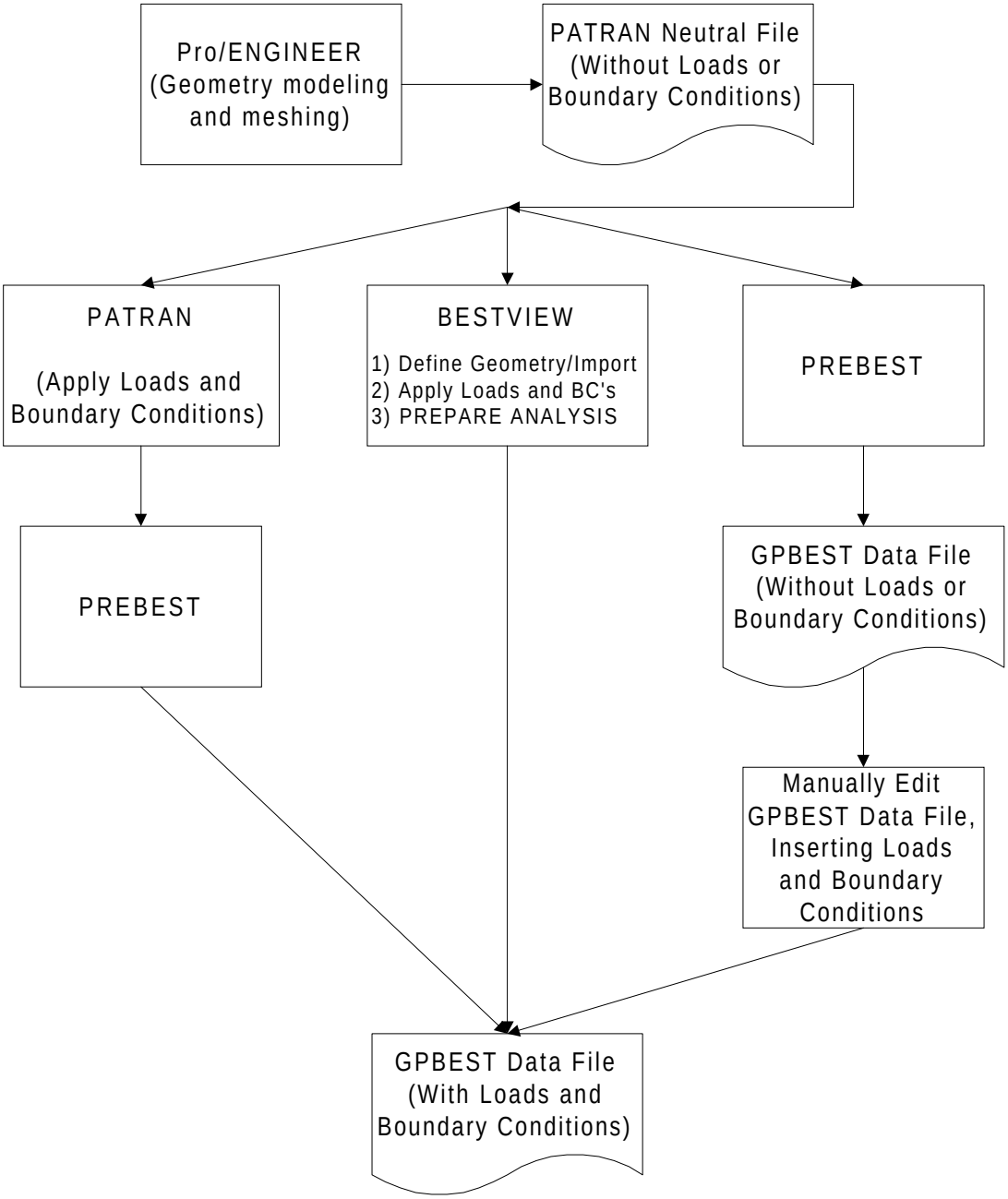
The overall process of generating a model in PRO/Engineer for GPBEST is:

1. Create the part geometry, which is typically a solid model.
2. Create the boundary element model:
    a) Define the model. This includes the material properties, mesh control, etc.
    b) Make the model (mesh generation).
    c) Output the model to a PATRAN neutral file. In carrying out this step, make sure that you use the following options: PATRAN, Parabolic, Accept.

Once the boundary element mesh is saved in a PATRAN neutral file, it is necessary to apply boundary conditions and loads. This can be done in the following ways:

1. In PATRAN.
2. In BESTVIEW.
3. Manually edited in the GPBEST data file.

Although the last option is the 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, it may only be necessary to make note of only a few elements to specify the loads and boundary conditions. These three paths are illustrated in the following figure:



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.

IGES

TIP: For further information on IGES, consult Section 3.9 in the BESTVIEW User's Manual (Volume II).

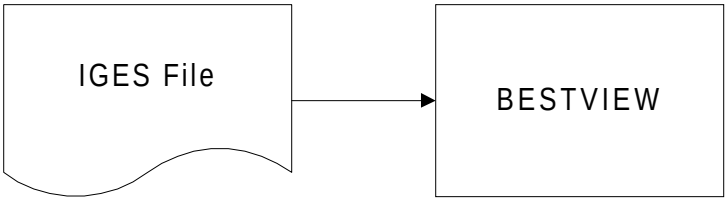
Most CAD and finite element modeling programs support some form of IGES (Initial Graphics Exchange Specification). There is no computer program named "IGES." IGES is merely a specification of a standard or neutral file format that can be used by numerous programs for the purpose of CAD entity data exchange.

Although IGES is a "standard," most programs only support a subset of IGES entities. Therefore, users must be very careful only to use entities that are:

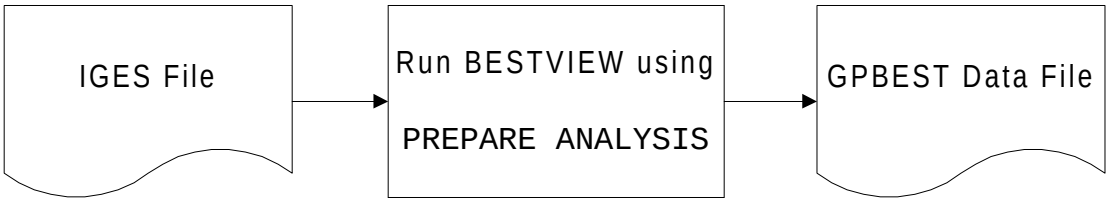
1. Supported by IGES.
2. Supported by the translators to and from IGES.

TIP:. Section 3.9 of the BESTVIEW User's Manual (Volume II) contains a list of the IGES entities supported by the GPBEST/IGES translator. The user is encouraged to consult this list to insure that the desired entities will translate properly.

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 it is necessary to use IGES, then BESTVIEW can read the IGES file directly:



In order to translate an IGES model to create a GPBEST data file, BESTVIEW can be used to 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:


IGES File
Run BESTVIEW using
PREPARE ANALYSIS
GPBEST Data File

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 or I-DEAS universal file.
2. FEMAP neutral file or HYPERMESH neutral 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 with relative ease.

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.

Auto-Meshing

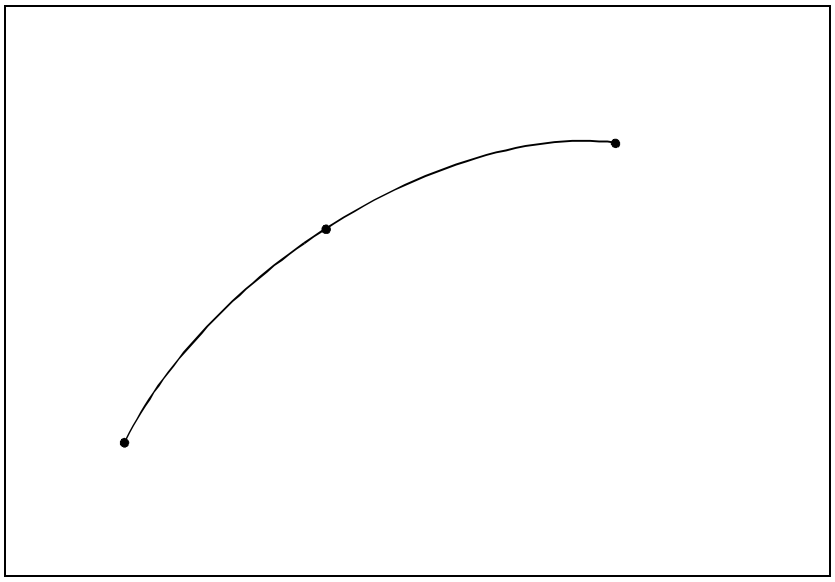
Many of the previously mentioned finite element modeling programs have auto-meshing capability. This refers to the ability to create a mesh automatically over surface geometry. Auto-meshing enables the analyst to spend less time in the mesh generation phase and get into the analysis phase more quickly. It has drastically reduced the overall time spent to complete finite element analyses.

The GPBEST user can take advantage of auto-meshing technology as well. However, there are some limitations that should be well understood in order to use this tool effectively.

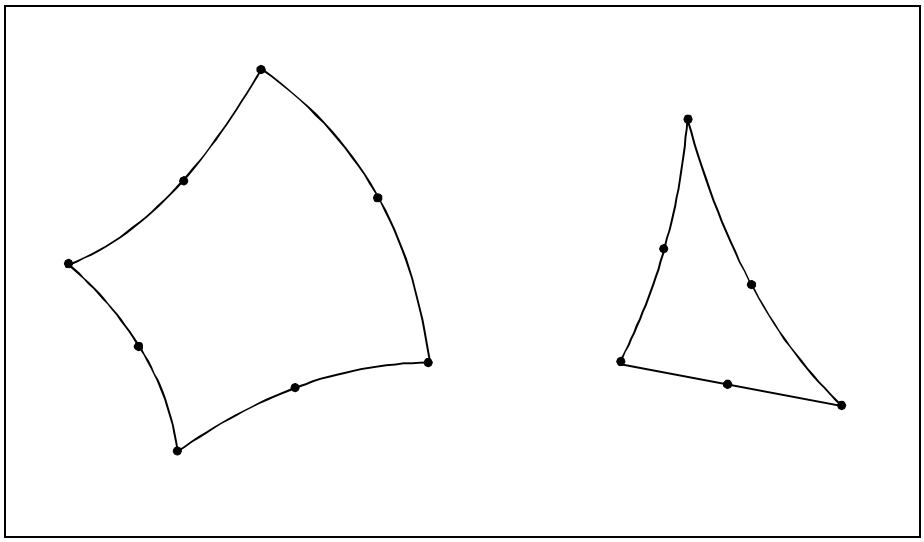
TIP: Many useful suggestions for building boundary element models are given in Chapter 9 of the BESTVIEW User's Manual (Volume II).

First, many auto-meshers default to creating *linear* elements: In the case of 2-D and axisymmetry, this involves 2-noded "beam" elements. In the case of 3-D, it involves 3-noded or 4-noded "shell" elements. Hence, very densely-meshed areas of the model arise around corners and complex edges. In the BEM, quadratic elements perform superior to linear elements, so they are preferred. The following figures illustrate these preferred elements for GPBEST:

2-D Boundary Element



3-D Boundary Elements



If possible, the user should direct that *quadratic* elements be generated by the auto-mesher.

The second consideration is that GPBEST models do not need to be as finely meshed as corresponding FEM models. There are two reasons for this: 1) GPBEST uses quadratic elements, which capture functional gradients much better than linear elements, and 2) The BEM is inherently more accurate than the FEM, especially in areas of steep stress gradients, such as corners and holes. Thus, when using auto-meshers, the GPBEST user will typically not require as fine a mesh as is typically required in a FEM model of the same part. However, no general rules can be given regarding element size. Experience will help the GPBEST user decide on proper levels of mesh refinement for different kinds of models.

TIP: The element quality checks performed by the PREBEST translator is discussed in detail in Chapter 9 of the BESTVIEW User's Manual (Volume II).

A third consideration regarding auto-meshing is that these routines often produce poorly-shaped elements. Elements with high aspect ratios, warping, etc. are not uncommon. The user can address this in two different ways. First, if the FEM program allows the user to set limits for "acceptable" elements, then this is the preferred method. However, if limits can not be defined, then the user can look at the output from the PREBEST translator step to determine which elements are poorly shaped for boundary element analysis. This output provides a "hit list," which can be used to modify the offending elements back in the modeling program. Once the elements are divided or fixed, the model can be translated into an acceptable GPBEST data file.

TIP: Do not ignore the messages from the PREBEST and POSTBEST translators. They may be directing you to potential problems with your model.

APPENDIX A - REFERENCES

General UNIX Literature

Introducing UNIX System V *by R. Morgan and H. McGilton, 1987, McGraw-Hill, Inc., New York, New York.*

UNIX Power Tools *by J. Peek, T. O'Reilly and M. Loukides, 1993, O'Reilly & Associates, Inc., Sebastopol, California.*

Essential System Administration *by A. Frisch, 1995, O'Reilly & Associates, Inc., Sebastopol, California.*

Boundary Element Literature

The Boundary Element Methods in Engineering *by P.K. Banerjee, 1994, McGraw-Hill, Inc., London, England.*

APPENDIX B - UNITS AND DIMENSIONS

A Consistent Set Of Units

Units are often confusing to a new user of any engineering analysis software. The units for all terms must be consistent. This means that the length, mass density, pressure, and gravitational acceleration must all be represented by a consistent set of units. For instance, if the length of the model is specified in millimeters (mm), and the gravitational acceleration is given as $9.8\left(\frac{m}{s^2}\right)$, then there is a mismatch in the units. Although GPBEST will execute properly, the results will not be correct. To be consistent, the gravitatational acceleration would be specified as $9800\left(\frac{mm}{s^2}\right)$.

This table is intended to take some of the burden off the user by suggesting a set of commonly-used units for each GPBEST card which have units implicitly associated with terms. Of course, any units can be used, but they must still be consistent. The following table is a starting point.

A Table Of Suggested Units

NOTE: This table lists GPBEST keywords relevant to all analysis types; not just elasticity. To determine which keywords apply only to elastic analysis in GPBEST, refer to Appendix E in this manual.

	GPBEST CARDS	SYMBOLS	ENGLISH	METRIC	CONVERSIONS
Model dimensions; length; thickness; displacement	POIN THIC DISP	*x,y,z,r* *l* *t* *u*	inch (*in*)	millimeter (*mm*)	$1m = 1000mm = 39.370in$ $1ft = 12in$ $1in = 2.540mm$
Time	TIME	*t*	second (*s*)	second (*s*)	$1hr = 60\min = 3600s$
Mass		*m*	$\left(\frac{lb_f s^2}{in}\right)$	$\left(\frac{N s^2}{mm}\right)$	$1kg = 1\frac{N s^2}{m} = .068522slug$ $1slug = 1\frac{lb_f s^2}{ft} = 32.174lb_m$ "1 *kg* weighs 9.8 *N*" "1 *slug* weighs 32.174 lb_f"

	GPBEST CARDS	SYMBOLS	ENGLISH	METRIC	CONVERSIONS
Mass density Fluid mass density	DENS FDEN	ρ	$\left(\frac{lb_f\, s^2}{in^4}\right)$	$\left(\frac{N\, s^2}{mm^4}\right)$	$1\frac{N\, s^2}{mm^4} = 93574\frac{lb_f\ \ s^2}{in^4}$
Force	FORC	*F,f*	pound (*lb*$_f$)	newton (*N*)	$1N = 1\frac{J}{m} = 1\frac{kg\, m}{s^2} = .22481 lb_f$
Velocity	SPEE VELO	*v*	$\left(\frac{in}{s}\right)$	$\left(\frac{mm}{s}\right)$	$1\frac{m}{s} = 2.2369\frac{mi}{hr} = 39.370\frac{in}{s}$ $c = 2.9979 \times 10^8\ \frac{m}{s}$
Acceleration	ACCE	*a,g*	$\left(\frac{in}{s^2}\right)$	$\left(\frac{mm}{s^2}\right)$	$g = 9800.\frac{mm}{s^2} = 386.1\frac{in}{s^2}$ "standard acceleration" (@ 45° latitude): $g = 9.80665\frac{m}{s^2} = 32.1740\frac{ft}{s^2}$
Angle	ORIE	*a1,a2,a3* *b1,b2,b3*	*degrees (°)*	*degrees (°)*	2π rad = 360°
Temperature	TEMP	*T,θ*	*°F*	*°C*	$°C = \frac{5}{9}(°F - 32)$ $°K = \frac{5}{9}°R = 273.15 + °C$ $°R = °F + 459.67$ For the purpose of "multiplying by one" to convert units, use: $1 = \frac{9°F}{5°C}$
Young's Modulus Traction Stress Pressure Bulk modulus	EMOD TRAC PRES SBUL	*E* *τ* *σ* *p*	$psi = \frac{lb_f}{in^2}$	$MPa = \frac{N}{mm^2}$	$1MPa = 10^6\, Pa = 1\frac{N}{mm^2}$ $= 145.04\, psi$ $1bar = .1\frac{N}{mm^2}$ $p_{atm} = .10132 MPa = 14.694\, psi$

	GPBEST CARDS	SYMBOLS	ENGLISH	METRIC	CONVERSIONS
Strain		ε	$\frac{in}{in}$	$\frac{mm}{mm}$	
Thermal conductivity	COND	k	$\frac{in\ lb_f}{s\ in\ °F}$	$\frac{N\ mm}{s\ mm\ °C}$	$1J = 1N\,m = 8.8507 in\,lb_f$ $1Btu = 9338 in\,lb_f = 1055J$
Specific heat	SPEC	c	$\frac{in^2\ lb_f}{s^2\ lb_f\ °F}$	$\frac{mm^2\ N}{s^2\ N\ °C}$	$1W = 1\frac{J}{s} = 1\frac{N\,m}{s} = .001360hp$ $= 9.485\times10^{-4} Btu/s$
Thermal diffusivity		α	$\frac{in^2}{s}$	$\frac{mm^2}{s}$	$\alpha = \frac{k}{\rho c}$
Permeability	PERM	κ	$\left(\frac{in}{s}\right)$	$\left(\frac{mm}{s}\right)$	
Coefficient of thermal expansion	ALPH	α	$\frac{1}{°F}$	$\frac{1}{°C}$	
Viscous damping coefficient	DAMP				
Thermal flux	FLUX	q	$\frac{in\ lb_f}{s\ in^2}$	$\frac{N\ mm}{s\ mm^2}$	
Spring constant	SPRI	k	$\frac{lb_f}{in^3}$	$\frac{N}{mm^3}$	The spring constant, *k*, is a force per unit length while being distributed over a unit area.
Convective film coefficient	CONV	h	$\frac{in\ lb_f}{s\ in^2\ °F}$	$\frac{N\ mm}{s\ mm^2\ °C}$	
Thermal resistance	RESI	R	$\frac{in\ s\ °F}{lb_f}$	$\frac{mm\ s\ °C}{N}$	
Impedance	IMPE	Z	$\frac{lb_f s}{in^3}$	$\frac{N\,s}{mm^3}$	
Admittance	ADMI	A	$\frac{in^3}{lb_f\ s}$	$\frac{mm^3}{N\ s}$	$A = \frac{1}{Z}$

	GPBEST CARDS	SYMBOLS	ENGLISH	METRIC	CONVERSIONS
Angular rotation	SPEE	ω	*rpm*	*rpm*	$1\frac{rad}{s} = .159154\frac{cycle}{s}$ $= 9.54930 rpm$ $1Hz = 1\frac{cycle}{s} = 1\frac{rev}{s}$
Point heat source	SOUR	ψ	$\frac{lb_f}{in^2\ s}$	$\frac{N}{mm^2\ s}$	
Point sound source	SOUR	I	$\frac{in^3}{s}$	$\frac{mm^3}{s}$	
Frequency	FREQ		$\frac{rad}{s}$	$\frac{rad}{s}$	
Energy rate	ENER	P	$\frac{in\ lb_f}{s}$	$\frac{N\ mm}{s}$	$1W = 1\frac{J}{s} = 1\frac{N\ m}{s}$

Some Notes About Units And Conversions

1. Traditional units for length involve the foot (ft) in English units and the meter (m) in metric. However, for common mechanical components, inches (in) and millimeters (mm) are preferred. If you desire to work in feet or meters, merely substitute these for inches and millimeters respectively in the above table.
2. Note that some terms contain length within them. For instance, specific heat and thermal flux both contain length dimensions. It is important that the GPBEST user match consistent units among all terms.
3. Traditional units for heat transfer problems involve BTUs and Watts. However, these are not listed in the above table as primary quantities. For one reason, BTUs and Watts contain embedded reference to feet and meters which can be confusing. Also, the mechanical equivalent of heat has been used to convert these quantities to basic quantities involving force and length. This is essential for coupled thermoelasticity where mixed units between heat transfer and elasticity components would wreak havoc.
4. For all coupled analyses, it is absolutely imperative to use a consistent set of units. The above table should help.

5. The above table shows units and dimensions for 3-D models. For planar (2-D) models, it is important to reduce some of the length quantities by one magnitude, essentially giving quantities *per unit thickness*. For example, in 3-D analysis, the spring constant is $k = N/mm^3$. However, in a 2-D analysis it would be $k = N/mm^2$ where it is understood that this represents the spring constant per unit length.
6. The metric system (SI) fundamental units are meter, second, kilogram, and Kelvin degree. However, many engineers in industry use millimeter, second, Newton, and Celsius degree.
7. In converting from one system of units to another, it is useful to "multiply by one." The initial quantity is multiplied successively by a series of "ones," where each "one" is a conversion factor. Units are canceled until the desired units remain. For instance, the "one" to convert meters to mm is $1 = \left(\frac{1000mm}{1m}\right)$. The following examples illustrate the procedure:

   Example 1: The mass density of aluminum is given in tables as $169\frac{lb_m}{ft^3}$. Convert to the suggested English units given in the above table.

$$\rho_m = 169\frac{lb_m}{ft^3} \times \left(\frac{1\frac{lb_f\ s^2}{ft}}{32.174 lb_m}\right) \times \left(\frac{1ft}{12in}\right)^4 = \underline{\underline{2.533\times10^{-4}\ \frac{lb_f\ s^2}{in^4}}}$$

   Example 2: The specific heat of steel is given in tables as $465\frac{J}{kg\,°C}$. Convert to the suggested metric units given in the above table.

$$c = 465\frac{J}{kg\,°C} \times \left(\frac{1N\,m}{J}\right) \times \left(\frac{1kg}{\left(\frac{N\,s^2}{m}\right)}\right) \times \left(\frac{1000mm}{m}\right)^2 = \underline{\underline{4.65\times10^8\ \frac{mm^2\ N}{s^2\ N\,°C}}}$$

Approximate Properties for Steel (including appropriate units)

Material Property	Metric Units	English Units
E – Young's Modulus	$2.07\times10^{5}\ \frac{N}{mm^{2}}$	$3.00\times10^{7}\ psi$
ν - Poisson's Ratio	0.291	0.291
ρ - Mass Density	$7.80\times10^{-9}\ \frac{N\,s^{2}}{mm^{4}}$	$7.30\times10^{-4}\ \frac{lb_f\,s^{2}}{in^{4}}$
α - Coefficient of Thermal Expansion	$1.31\times10^{-5}\ \frac{1}{°C}$	$7.30\times10^{-6}\ \frac{1}{°F}$
k – Thermal Conductivity	$54.0\ \frac{N\ mm}{s\ mm\ °C}$	$6.70\ \frac{in\ lb_f}{s\ in\ °F}$
c – Specific Heat	$4.65\times10^{8}\ \frac{mm^{2}\ N}{s^{2}\ N\ °C}$	$4.00\times10^{5}\ \frac{in^{2}\ lb_f}{s^{2}\ lb_f\ °F}$

Approximate Properties for Aluminum (including appropriate units)

Material Property	Metric Units	English Units
E – Young's Modulus	$7.10\times10^{4}\,\frac{N}{mm^{2}}$	$1.03\times10^{7}\,psi$
ν - Poisson's Ratio	0.334	0.334
ρ - Mass Density	$2.64\times10^{-9}\,\frac{N\,s^{2}}{mm^{4}}$	$2.47\times10^{-4}\,\frac{lb_f\,s^{2}}{in^{4}}$
α - Coefficient of Thermal Expansion	$2.39\times10^{-5}\,\frac{1}{°C}$	$1.33\times10^{-5}\,\frac{1}{°F}$
k – Thermal Conductivity	$204.\,\frac{N\ mm}{s\ mm\ °C}$	$25.5\,\frac{in\ lb_f}{s\ in\ °F}$
c – Specific Heat	$8.96\times10^{8}\,\frac{mm^{2}\ N}{s^{2}\ N\ °C}$	$7.72\times10^{5}\,\frac{in^{2}\ lb_f}{s^{2}\ lb_f\ °F}$

APPENDIX C - USEFUL UNIX SCRIPTS

Script: fromto.sh

```
#!/bin/sh
#
#  Script to search through a file and output only lines from
#  string1 to string2.
#
#  Basic syntax:  fromto.sh string1 string2 filename
#
#  e.g.:  fromto.sh ’ST RE’ PAGE file1.res<RET>
#
#  Notes:
#   1) If you have more than one occurrence of the
#      string1/string2 combination, this will output
#      each occurrence.
#   2) If string2 is not found, everything from string1 to
#      the end of the file will be output.
#
if [ $# -ne 3 ]
then
  echo Must input three arguments:
  echo   e.g. fromto.sh string1 string2 filename
  exit
fi
#
sed -n /"$1"/,/"$2"/p $3
exit
```

Script: runbest.sh

```
#!/bin/sh
#
# runbest.sh - script to execute GPBEST.
#
# To execute:  runbest.sh job1 & <CR>; where data file is
job1.dat.
#
# NOTES:
#   1) This assumes data file is named *.dat and results are
named *.res
#
```

```
#

if [ $# -ne 1 ]
then
  echo Must input root of filename to run: e.g.  runbest.sh beam2
  exit
fi

if [ -r $1.dat ]
then
     # Delete neutral file if it exists:
     if [ -f NEUTRAL.OUT ]
     then
          rm NEUTRAL.OUT
     fi

     echo Running GPBEST with input file:  $1.dat
     # Execution time is tacked on the end of the result file.
     echo 'Start:' `date` > junk.date
     nohup time < $1.dat  best-s > $1.res  2> junk.time
     echo 'End:  ' `date` >> junk.date
     # cat junk.date >> $1.res
     # cat junk.time >> $1.res

     # Rename neutral file if it exists:
     if [ -f NEUTRAL.OUT ]
     then
          mv NEUTRAL.OUT $1.neu
     fi

     # Search for key messages:
     grep "ERROR" $1.res > $1.abr
     grep "FATAL" $1.res >> $1.abr
     grep "WARN"  $1.res >> $1.abr
     grep "INFORMATIONAL MESSAG"  $1.res >> $1.abr
     grep "ITERATIONS"  $1.res >> $1.abr
     fromto.sh 'TOTAL LOAD BALANCE' 'JOB TITLE' $1.res >> $1.abr
     cat junk.date >> $1.abr
     cat junk.time >> $1.abr
     rm junk.date junk.time
     ls -lrt | tail >> $1.abr

else
     echo Filename $1.dat does not exist!
fi

# beep
exit
# end of script
```

APPENDIX D - GPBEST/BESTVIEW PASSWORD REQUEST

A password file is required in order to be able to execute GPBEST or BESTVIEW. If the password information was not included with your GPBEST package, please make a copy of the 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.

In order for us to create a password for you, we will need your hostname and system ID. You will need this information to fill out the password request form on the next page. To determine your hostname, simply execute the UNIX hostname command. To determine your system ID, please enter the appropriate UNIX command for your system:

Computer System	***System ID Command***
SUN Solaris systems	/usr/bin/showrev
IBM RS6000 systems	uname -m
Silicon Graphics systems	/etc/sysinfo -s
Hewlett Packard 9000/700 series systems	uname -i
Digital DEC ALPHA	netstat -i | grep 08:00 | awk '{ print $4 }'

NOTE: If any of these commands are not found, you may have to consult your system administrator or search some of the above directories to find the proper command and/or location. If you need more than one license for a specific computer type, please fill out another request form with only model, hostname and system ID information. Multiple computers can be incorporated into one password file, and a single password file can be used for GPBEST and BESTVIEW.

GPBEST AND BESTVIEW PASSWORD REQUEST FORM

Please print in large, distinct letters. Fax your password request forms to (716) 639-1919.

Date: _____ Fax: _____

Name: _____ Phone: _____

E-mail Address: _____
(Passwords will be sent via E-mail if an E-mail address is provided. This is usually more convenient.)

Company/Organization: _____

Address: _____

_____

_____

Check one: ☐ New Order ☐ Renewal ☐ Trial ☐ Upgrade ☐ Info. Change

Computer Type: ☐ SUN Sparc ☐ IBM RS6000 ☐ Silicon Graphics
(check one) ☐ HP9000/700 ☐ DEC Alpha

Operating System Version: _____

Model: _____

Hostname: _____
(Use hostname command; see previous page)

System ID: _____
(See instructions on previous page)

Starting Date: _____ Ending Date: _____

System Administrator: _____ Phone: _____

Any questions? Please fax, call, write, or send E-Mail:

Fax: (716) 639-1919 Phone: (716) 639-1920 E-Mail: support@gpbest.com

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

Example Of The GPBEST password File

Here is an example of a GPBEST password file. This is typically mailed to users via E-Mail, so the mail headings must be stripped off. Only lines like the following should remain in the password file. It is installed according to the instructions given in Chapter 2 of this manual.

```
WWK44J3G2WCXJQE446LW
CXRGLWGJCBWGXRJJ29J3
GQBEWEQX642GCK4GX9LY
2GGKCGC2CGGCWJB2RJKY
BPCER2ERE2JGCGPGG2L3
4G4GCR2Q4C22W4YKYWJW
WRG2J2WG2622JJCGYWKC
3JJXJBP6K2GCC4XWEWJW
QWJWJ24BPB2898643EN3
RYCXE2GGRYR2E8XKRYNE
```

... (etc., etc., etc.) ...

```
PBJBCERY4CGW4G4XWBB9
EG42JGX2L3462CYJBG2Q
2LQGKCGQ2898XERYGGPN
J8XPRJE222JK2JJRJBLW
GYQ4X2L3YJ26RWEG2WNL
22BEE64W2W6W6CXRYPL2
JQEEEWBGGK2RGB4L8JL9
K4GWGWBGWCK2G6RWPBBG
JWGWJGWJEG3BJRWEWEB3
```

APPENDIX E - GPBEST ELASTIC ANALYSIS KEYWORDS

Introduction

Chapter 5 of the GPBEST User's Manual contains all of the input cards for a GPBEST input data file. However, that chapter contains all possible cards, some of which do not apply to basic elastic analysis. This appendix has been provided in order to aid the user to wade through the masses of material there. *The purpose of this appendix is to list the cards that are relevant for elastic analysis and eliminate the cards that do not apply.* By using this section as reference, the user will be able to concentrate on cards that are supported in elastic analysis instead of attempting to use cards intended for other analysis types.
The section numbers and page numbers refer to sections and pages in the GPBEST User's Manual (Volume I), June, 1998.

Section 5.1 - Case Control

SECTION	KEYWORD	PURPOSE	PAGE
5.1.1	Case Control Input Card		
	**CASE	Start of case control input	5.1.5
5.1.2	Title		
	TITL	Title of job	5.1.6
5.1.3	Dimensionality of the Problem (default is 3-D)		
	AXIS	Axisymmetric flag	5.1.7
	AXIS GENE	Axisymmetric body with general (non-axisymmetric) loading	5.1.7
	PLAN STRE	Plane stress flag	5.1.10
	PLAN STRA	Plane strain flag	5.1.10
	MIXED	Dimensionality is region-dependent	5.1.11
5.1.4	Type of Analysis		
	ELAS	Elastic analysis	5.1.20
5.1.6	Times/Frequencies for Solution and Output		
	TIME	Times for solution output	5.1.29

SECTION	KEYWORD	PURPOSE	PAGE
5.1.7	Geometric and Loading Symmetry Control		
	SYMM HALF	Half Symmetry	5.1.36
	SYMM QUAR	Quarter Symmetry	5.1.36
	SYMM OCTA	Octal Symmetry	5.1.36
5.1.8	Restart Facility		
	REST WRIT	Save integration files for future runs	5.1.43
	REST READ	Use integration files from previous run	5.1.43
	REST HOLE	Use surface integration files from previous run, but permit changes in hole elements	5.1.43
	REST SOLV	Use the decomposed system matrix from previous run	5.1.43
	REST GMR	Use a GMR-based restart facility	5.1.43
	SAVE SOLV	Save the decomposed system matrix and right-hand side matrices for future runs	5.1.47
5.1.9	Efficiency and Precision Control		
	PREC	Precision of analysis	5.1.49
	STOR	Precision for storage of equation systems	5.1.50
	SOLV SING	Use single precision variables in solver	5.1.51
	SOLV DOUB	Use double precision variables in solver	5.1.51
	SOLV ELIM	Use a solver algorithm based on gaussian elimination	5.1.52
	SOLV ITER	Use an efficient iteration-based solver algorithm	5.1.52
	MEMO	Specify dynamic memory allocation for use in the system equation solver	5.1.56
5.1.10	Output Options		
	ECHO	Produce echo of input data	5.1.61
	PRIN BOUN	Print displacement and traction results	5.1.62
	PRIN NODA	Print boundary displacement, stress, and strain at nodal points	5.1.62
	PRIN LOAD	Print load calculation	5.1.62
	PRIN DATA	Print input data	5.1.62

SECTION	KEYWORD	PURPOSE	PAGE
	PRIN ALL	Provide the maximum printed output file	5.1.62
	PRIN NONE	Provide only a minimal printed output file	5.1.62
	PRIN LIMI	Print current GPBEST limits	5.1.62
	PRIN FEAT	Print current implementation status of GPBEST special features	5.1.62
	NEUT	Produce formatted GPBEST neutral file (All quantities)	5.1.64
	NEUT BOUN	Print boundary quantities in neutral file	5.1.64.
	NEUT NODA	Print nodal quantities in neutral file	5.1.64
	NEUT LOAD	Print load balance in neutral file	5.1.64
	NEUT DATA	Print input data in neutral file	5.1.64
	NEUT ALL	Print all quantities in neutral file	5.1.64
	NEUT READ	Write a sequential neutral file	5.1.64
	CONT	Create output in neutral file for contour plots of results in post-processing 2-D problems	5.1.69
5.1.11	Miscellaneous Control Options		
	CHEC	Check input data only	5.1.73
	INCL	Include another file in the GPBEST data set	5.1.77
	COOR	Specifies coordinate system for input	5.1.79

Section 5.2 - Material Property Definition

SECTION	KEYWORD	PURPOSE	PAGE
5.2.1	Material Property Input Card		
	**MATE	Beginning of a material property input set	5.2.7
5.2.2	Material Identification		
	ID	Identifier of a material type	5.2.8
5.2.3	Mass Parameter		
	DENS	Material mass density	5.2.9
5.2.4	Isotropic Elastic Parameters		
	EMOD	Young's modulus	5.2.10
	POIS	Poisson's ratio	5.2.11
5.2.6	Isotropic Temperature-dependent Thermoelastic Parameters		
	TEMP	Temperature values at which elastic material properties will be defined	5.2.15
	EMOD	Young's modulus	5.2.17
	ALPH	Coefficients of thermal expansion	5.2.18
5.2.7	Anisotropic Elastic Parameters		
	ANIS	Identifier for anisotropic material	5.2.19
	STIF	Material stiffness	5.2.21
	COMP	Material compliances	5.2.24
	TECH	Technical constants for an orthotropic material	5.2.27
	ALPH	Coefficients of thermal expansion	5.2.29
	ORIE	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	5.2.31

Section 5.3 - Definition of Geometry

SECTION	KEYWORD	PURPOSE	PAGE
5.3.1	Geometry Input Card		
	**GMR	Start of geometric modeling region input	5.3.16
5.3.2	Region Identification		
	ID	Region ID	5.3.18
	AXIS	Axisymmetric region	5.3.19
	PLAN STRE	Plane stress region	5.3.20
	PLAN STRA	Plane strain region	5.3.20
	THIC	Thickness of a 2-D plane or plate/shell region	5.3.21
	MATE	Material property (set) for region	5.3.25
	REST	Restart option for region	5.3.26
	SAVE	Save reduced system matrix and right hand side for future analyses	5.3.28
	TINT	Temperature used to determine material properties for integration	5.3.29
	TREF	Reference (initial) temperature of region	5.3.30
	EXTE	Region is an infinite body	5.3.31
	HALF	Region is a half-space	5.3.33
	CONT	GMR control of contour plot information	5.3.35
5.3.3	Nodal Point Definition		
	POIN	Nodal points for boundary and volume discretization (coordinates)	5.3.37
5.3.4	Surface Element Definition		
	SURF	Beginning of surface discretization	5.3.40
	TYPE LINE	Linear surface variation of field quantities	5.3.42
	TYPE QUAD	Quadratic surface variation of field quantities	5.3.42
	TYPE QUAR	Quartic surface variation of field quantities	5.3.42
	GEOM LINE	Linear geometry specified	5.3.47
	GEOM QUAD	Quadratic geometry specified	5.3.47
	ELEM	Element list (element connectivity)	5.3.49
	TRAN	Surface generation by translation	5.3.59

SECTION	KEYWORD	PURPOSE	PAGE
	REFE	Axis of rotation for surface generation	5.3.60
	DIRE	Direction of axis of rotation	5.3.61
	ROTA	Angle of rotation for surface generation	5.3.62
	CRAC	Defines the nodes at the crack front	5.3.63
5.3.5	Normal Definition		
	NORM	Defines outer normal of surface	5.3.64
5.3.6	Enclosing Element Definition		
	ENCL	Enclosing element list (enclosing element connectivity)	5.3.70
5.3.7	Discontinuity Element Definition		
	DISC	Discontinuity element list (discontinuity element connectivity)	5.3.74
5.3.8	Line Element Definition		
	LINE	Start of line definition	5.3.78
	POIN	Nodal points for hole discretization (coordinates)	5.3.79
	TYPE LINE	Linear variation of field quantities for 3-noded holes	5.3.81
	TYPE QUAD	Quadratic variation of field quantities for 3-noded holes	5.3.81
	HOLE	Start of hole definition	5.3.82
	ELEM	Hole element list (element connectivity and radius of hole)	5.3.91
5.3.9	Sampling Points		
	SAMP POINT	Start of definition of sampling points (coordinates)	5.3.96
5.3.10	Sampling Surface		
	SAMP SURF	Beginning of sampling surface discretization	5.3.98
	ELEM	Sampling surface element list (sampling surface element connectivity)	5.3.100

Section 5.4 - Definition of GMR Compatibility (Interfaces)

SECTION	KEYWORD	PURPOSE	PAGE
5.4.1	Interface Definition Input Card		
	**INTE	Start of interface compatibility condition	5.4.7
	GAP	Indicates an initial gap or overlap	5.4.8
5.4.2	Definition of interface surface 1		
	GMR	Name of first GMR	5.4.14
	SURF	Surface on first GMR	5.4.15
	ELEM	Element of surface	5.4.17
	POIN	Nodal point of surface for gap definition	5.4.18
5.4.3	Definition of interface surface 2		
	GMR	Name of second GMR	5.4.19
	SURF	Surface on second GMR	5.4.20
	ELEM	Element of surface	5.4.22
	POIN	Nodal point of surface for gap definition	5.4.23
5.4.4	Type of interface conditions		
	BOND	Bonded interface connection	5.4.28
	SLID	Sliding interface connection	5.4.30
	FIT	Interference fit	5.4.32
	CONT	Nonlinear contact interface	5.4.36
	SPRI	Spring condition	5.4.39
5.4.5	Additional interface input		
	FRIC	Coefficient of static friction	5.4.45
	OFFS	Offset for nonlinear contact	5.4.46
5.4.6	Cyclic Symmetry interface definition		
	CYCL	Cyclic symmetry interface definition	5.4.51
	ANGL	Angle for cyclic interface	5.4.54
	DIRE	Axis of rotation for cyclic interface	5.4.56

Section 5.5 - Definition of Boundary Conditions

SECTION	KEYWORD	PURPOSE	PAGE
5.5.1	Boundary Condition Set Card		
	**BCSET	Start of the B.C. definition	5.5.17
5.5.2	Boundary Condition Identification		
	ID	Name of B.C. set	5.5.18
5.5.3	Identification of Boundary Condition Type		
	VALU	For specified B.C. value input	5.5.20
	RELA	For B.C. relation between boundary quantities	5.5.21
	VARI	For specified B.C. value input which may change during subsequent RESTART runs	5.5.22
	INTE	Boundary condition on interface	5.5.26
	LOCA	For local definition of B.C.	5.5.29
	DIRE	Input for local tangential directions in 3-D	5.5.37
	NODE	Explicit definition for local directions	5.5.39
5.5.4	Definition of Surface for Application of Boundary Conditions		
	GMR	Identifies a GMR	5.5.43
	SURF	Identifies the surface for this B.C. set	5.5.44
	ELEM	Identifies surface elements	5.5.47
	POIN	Identifies surface points	5.5.48
	TIME	Defines the time for input	5.5.50
5.5.5	Value Boundary Condition for Surface Elements		
	TRAC	Traction B.C. input	5.5.53
	LOAD	Load B.C. input	5.5.59
	TORQ	Torque B.C. input	5.5.62
	DISP	Displacement B.C. input	5.5.65
5.5.8	Definition of Space/Time Variation		
	SPLI	Source (field variable) point list	5.5.80

SECTION	KEYWORD	PURPOSE	PAGE
	T	Nodal value of B.C.	5.5.82
5.5.9	Relation Boundary Condition		
	SPRI	Spring relation (between displacement and traction)	5.5.86
5.5.11	Examples of Boundary Conditions		

Section 5.6 - Body Force Definition

SECTION	KEYWORD	PURPOSE	PAGE
5.6.1	Body Force Input Card		
	**BODY	Start of body force input	5.6.4
5.6.2	Centrifugal body force		
	CENT	Centrifugal load input	5.6.6
	POINT	Point on the axis of rotation	5.6.7
	DIRE	Direction of axis of rotation	5.6.9
	TIME	Time for input	5.6.10
	SPEE	Speed input	5.6.11
5.6.3	Inertial body force		
	INER	Inertial body force input	5.6.12
	DIRE	Direction of acceleration	5.6.13
	TIME	Time of input	5.6.14
	ACCE	Acceleration input	5.6.15
5.6.5	Generalized Plane Stress/Plane Strain Input		
	GENE	Generalized plane stress/strain input	5.6.25
	TIME	Times for input	5.6.26
	GMR	Identifies GMR	5.6.27
	CONS	Constant Coefficient	5.6.28
	XVAL	X coefficient	5.6.29
	YVAL	Y coefficient	5.6.30
5.6.6	Point Source / Force		
	CONC	Concentrated (point) source/force	5.6.31
	TIME	Times for input	5.6.33
	GMR	Identifies GMR	5.6.34
	POINT	List of points at which values are specified	5.6.35
	FORCE	Force input	5.6.37
	T	Values at points	5.6.40
5.6.7	Discontinuity Surface Input		
	DISC	Discontinuity surface input	5.6.41
	TIME	Times for input	5.6.50
	GMR	Identifies GMR	5.6.52
	DISP	Displacement discontinuity input	5.6.53

Section 5.7 - Definition of Boundary Condition Changes

This section does not apply to elastic analysis.