MSC.Nastran 2001

Getting Started with MSC.Nastran
User’s Guide

MSC SOFTWARE

Corporate

MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707 USA
Telephone: (800) 345-2078
Fax: (714) 784-4056

Europe

MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Asia Pacific

MSC.Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjyku-Ku, Tokyo 160-0023, Japan
Telephone: (81) (03) 6911 1200
Fax: (81) (03) 6911 1201

Worldwide Web

www.mscsoftware.com

Disclaimer

C O N T E N T S

Getting Started with MSC.Nastran User's Guide

Preface

1 Some Preliminaries

2 Overview of the MSC.Nastran Finite Element Model

13 Example Problems

14

Overview of Other MSC.Nastran Capabilities

Preface

About this Book

How to Use this Guide

MSC.Nastran is a powerful general purpose finite element program that provides an extraordinary range of time-tested analysis capabilities. Problems that were intractable even a few years ago are routinely solved today using this technology.

Prerequisites. This book written exclusively for the new MSC.Nastran user. No prior experience with commercial finite element software is assumed and no finite element-specific university coursework is required. It is assumed that you have a bachelor's degree in any of the fields relevant to structural analysis: mechanical engineering, civil engineering, engineering mechanics, or the equivalent.

Goals of this Book. The principal goal of this book is to give new users the ability to model the basic types of beam, plate, and solid structures described in typical undergraduate mechanics-of-materials classes. Concentrating on linear static analysis, we will learn how to design an MSC.Nastran model, define its geometry, choose finite elements based on our knowledge of the mechanical behavior of the structure, define boundary conditions, apply loads, request and control program output, and interpret the results of an analysis. In addition, we will gain a basic understanding of some of MSC.Nastran's most popular intermediate and advanced analysis features.

Part I. This book is self-contained and designed to be read from beginning to end-many topics are introduced in general terms and expanded upon in subsequent discussions when the need for added detail arises. Part I of this book contains a broad overview of the finite element method, the MSC.Nastran program, and the makeup of an MSC.Nastran finite element model.

Part II. Part II describes the basic design, construction, and execution of MSC.Nastran models. **Basic Modeling Issues** (p. 7) describes several important elementary modeling concepts and MSC.Nastran issues. **Specifying the Type of Analysis** (p. 7) through **Additional Considerations** (p. 7) describe in detail the various components of an MSC.Nastran model and are organized the way engineers think about a structural problem (i.e., the type of analysis, geometry, boundary conditions, materials, loads, and so on). **Performing an MSC.Nastran Analysis Step-by-Step** (p. 7) contains a complete and fully narrated description of a simple MSC.Nastran analysis.

Part III. In Part III, **Example Problems** (p. 7) provides several additional example problems for further review and practice, and **Overview of Other MSC.Nastran Capabilities** (p. 7) gives an overview of MSC.Nastran's wide-ranging intermediate and advanced capabilities.

The Second Edition of the this book has been updated to correspond with MSC.Nastran Version 69.

Note to the Reader

This guide contains many excerpts from another MSC Software book, the ***MSC.Nastran Quick Reference Guide***. The ***MSC.Nastran Quick Reference Guide*** contains complete descriptions of all the finite element input data and options available in MSC.Nastran. Most of the excerpts have been edited-some extensively-to eliminate material that is not relevant to the topics covered in this book. After reading *Getting Started*, we recommend that you review the complete descriptions in the *Quick Reference Guide* in order to gain a broader understanding of MSC.Nastran's features and capabilities.

List of MSC.Nastran Books

Below is a list of some of the MSC.Nastran documents. You may order any of these documents from the MSC.Software BooksMart site at www.engineering-e.com.

Installation and Release Guides

- ❏ Installation and Operations Guide
- ❏ Release Guide

Reference Books

- ❏ Quick Reference Guide
- ❏ DMAP Programmer's Guide
- ❏ Reference Manual

User's Guides

- ❏ Getting Started
- ❏ Linear Static Analysis
- ❏ Basic Dynamic Analysis
- ❏ Advanced Dynamic Analysis
- ❏ Design Sensitivity and Optimization
- ❏ Thermal Analysis
- ❏ Numerical Methods
- ❏ Aeroelastic Analysis
- ❏ Superelement
- ❏ User Modifiable
- ❏ Toolkit

Technical Support

For help with installing or using an MSC.Software product, contact your local technical support services. Our technical support provides the following services:

- Resolution of installation problems
- Advice on specific analysis capabilities
- Advice on modeling techniques
- Resolution of specific analysis problems (e.g., fatal messages)
- Verification of code error.

If you have concerns about an analysis, we suggest that you contact us at an early stage.

You can reach technical support services on the web, by telephone, or e-mail:

Web

Go to the MSC.Software website at **www.mscsoftware.com**, and click on **Support**. Here, you can find a wide variety of support resources including application examples, technical application notes, available training courses, and documentation updates at the MSC.Software Training, Technical Support, and Documentation web page.

Phone and Fax

United States
Telephone: (800) 732-7284
Fax: (714) 784-4343

Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Rome, Italy
Telephone: (390) (6) 5 91 64 50
Fax: (390) (6) 5 91 25 05

Moscow, Russia
Telephone: (7) (095) 236 6177
Fax: (7) (095) 236 9762

Frimley, Camberley
Surrey, United Kingdom
Telephone: (44) (1276) 67 10 00
Fax: (44) (1276) 69 11 11

Tokyo, Japan
Telephone: (81) (3) 3505 02 66
Fax: (81) (3) 3505 09 14

Paris, France
Telephone: (33) (1) 69 36 69 36
Fax: (33) (1) 69 36 45 17

Gouda, The Netherlands
Telephone: (31) (18) 2543700
Fax: (31) (18) 2543707

Madrid, Spain
Telephone: (34) (91) 5560919
Fax: (34) (91) 5567280

Email Send a detailed description of the problem to the email address below that corresponds to the product you are using. You should receive an acknowledgement that your message was received, followed by an email from one of our Technical Support Engineers.

MSC.Patran Support	mscpatran.support@mscsoftware.com
MSC.Nastran Support	mscnastran.support@mscsoftware.com
MSC.Nastran for Windows Support	vn4w.support@mscsoftware.com
MSC.visualNastran Desktop 2D Support	vn2d.support@mscsoftware.com
MSC.visualNastran Desktop 4D Support	vndesktop.support@mscsoftware.com
MSC.Abaqus Support	mscabaqus.support@mscsoftware.com
MSC.Dytran Support	mscdytran.support@mscsoftware.com
MSC.Fatigue Support	mscfatigue.support@mscsoftware.com
MSC.Interactive Physics Support	ip.support@mscsoftware.com
MSC.Marc Support	mscmarc.support@mscsoftware.com
MSC.Mvision Support	mscmvision.support@mscsoftware.com
MSC.SuperForge Support	mscsuperforge.support@mscsoftware.com
MSC Institute Course Information	msctraining.support@mscsoftware.com

Training

The MSC Institute of Technology is the world's largest global supplier of CAD/CAM/CAE/PDM training products and services for the product design, analysis and manufacturing market. We offer over 100 courses through a global network of education centers. The Institute is uniquely positioned to optimize your investment in design and simulation software tools.

Our industry experienced expert staff is available to customize our course offerings to meet your unique training requirements. For the most effective training, The Institute also offers many of our courses at our customer's facilities.

The MSC Institute of Technology is located at:

2 MacArthur Place
Santa Ana, CA 92707
Phone: (800) 732-7211
Fax: (714) 784-4028

The Institute maintains state-of-the-art classroom facilities and individual computer graphics laboratories at training centers throughout the world. All of our courses emphasize hands-on computer laboratory work to facility skills development.

We specialize in customized training based on our evaluation of your design and simulation processes, which yields courses that are geared to your business.

In addition to traditional instructor-led classes, we also offer video and DVD courses, interactive multimedia training, web-based training, and a specialized instructor's program.

Course Information and Registration. For detailed course descriptions, schedule information, and registration call the Training Specialist at (800) 732-7211 or visit www.mscsoftware.com.

Internet Resources

MSC.Software (www.mscsoftware.com)

MSC.Software corporate site with information on the latest events, products and services for the CAD/CAE/CAM marketplace.

Simulation Center (simulate.engineering-e.com)

Simulate Online. The Simulation Center provides all your simulation, FEA, and other engineering tools over the Internet.

Engineering-e.com (www.engineering-e.com)

Engineering-e.com is the first virtual marketplace where clients can find engineering expertise, and engineers can find the goods and services they need to do their job

CATIASOURCE (plm.mscsoftware.com)

Your SOURCE for Total Product Lifecycle Management Solutions.

Permission to Copy and Distribute MSC Documentation

If you wish to make copies of this documentation for distribution to co-workers, complete this form and send it to MSC.Software Corporation. MSC will grant written permission if the following conditions are met:

- All copyright notices must be included on all copies.
- Copies may be made only for fellow employees.
- No copies of this manual, or excerpts thereof, will be given to anyone who is not an employee of the requesting company.

Please complete and mail to MSC for approval:

MSC.Software Corporation
Attention: Legal Department
2 MacArthur Place
Santa Ana, CA 92707

Name:____

Title: ____

Company: ____

Address:____

____

Telephone:_______________Email: ______________________________

Signature:___________________________ Date:____________________

Please do not write below this line.

APPROVED: MSC.Software Corporation

Name:____

Title: ____

Signature:___________________________ Date:____________________

Fold here

Place
Stamp
Here

MSC.Software Corporation

Attention: Legal Department

2 MacArthur Place

Santa Ana, CA 92707

Fold here

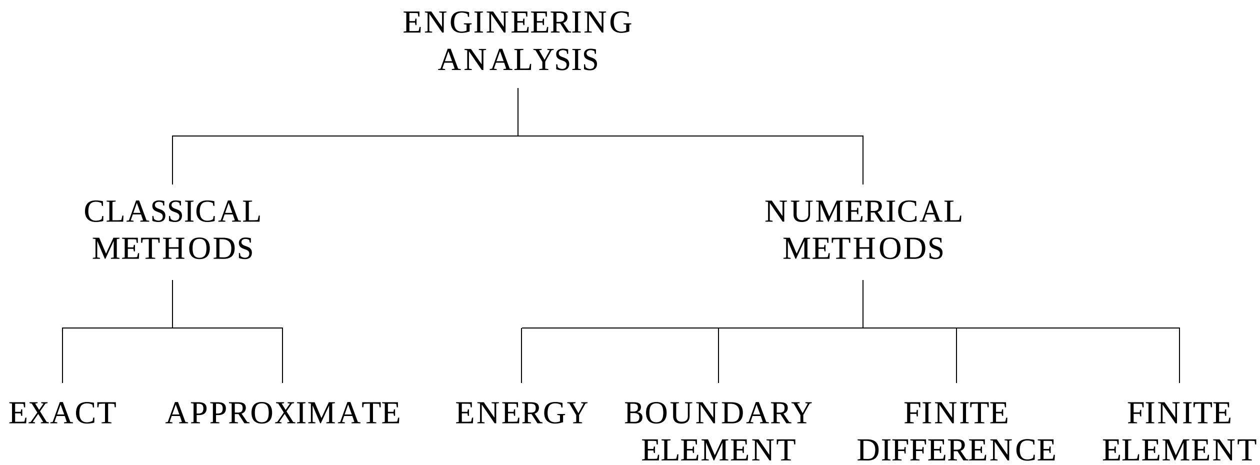
CHAPTER 1

Some Preliminaries

- The Role of the Finite Element Method in Engineering Analysis
- The Finite Element Process
- The Basic Matrix Equation of Linear Static Analysis
- Assumptions and Limitations of Linear Static Analysis

1.1 The Role of the Finite Element Method in Engineering Analysis

As a start, it is useful to see where the finite element method fits in with other methods of engineering analysis. Engineering analysis can be broadly divided into two categories: classical methods and numerical methods.



Classical Methods. Classical methods attempt to solve field problems directly by forming governing differential equations based on fundamental principles of physics. Exact solutions-those having closed forms-are possible only for the simplest cases of geometry, loading, and boundary conditions. A somewhat wider variety of classical problems can be solved using approximate solutions to the governing differential equations. These solutions take the form of series expansions which are truncated after a reasonable degree of convergence. In the structural world, many of Timoshenko's works and Roark's *Formulas for Stress and Strain* are essentially catalogs of these types of solutions. Like exact solutions, approximate solutions require regular geometric shapes, simple boundary conditions, and well behaved loads. Consequently, these solutions bear little resemblance to most practical engineering problems. The principal advantage of classical methods is the high degree of problem insight provided by solutions of this type.

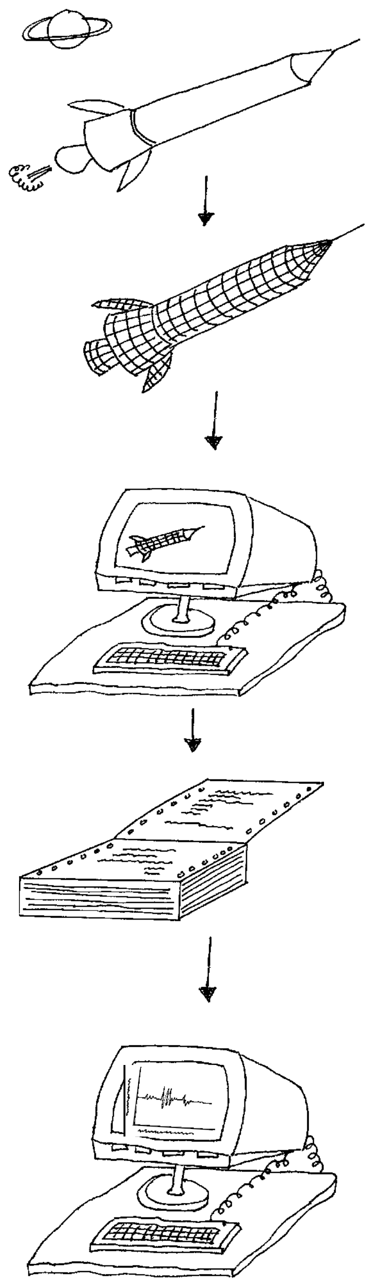
Numerical Methods. Numerical methods address a broad range of problems. The energy method seeks to minimize an expression for the potential energy of a structure over its entire domain. This approach works extremely well for certain problems, but it is not broadly applicable. The boundary element method approximates functions satisfying the governing differential equations, but not the boundary conditions. Problem size is reduced because elements represent only the boundary of the domain. However, the application of this method relies on knowing the fundamental solution to the governing equations, which can be difficult to obtain. The finite difference method replaces governing differential equations and boundary conditions with corresponding algebraic equations. This permits the representation of somewhat irregular problems, but complex geometry, boundary conditions, or loads become difficult to handle.

The Finite Element Method. The finite element method offers virtually unlimited problem generality by permitting the use of elements of various regular shapes. These elements can be combined to approximate any irregular boundary. In similar fashion, loads and constraints of any type can be applied. Problem generality comes at the expense of insight-a finite element solution is essentially a stack of numbers that applies only to the particular problem posed by the finite element model. Changing any significant aspect of the model generally requires a complete reanalysis of the problem. Analysts consider this a small price to pay, however, since the finite element method is often the only possible method of analysis. The finite element method is applicable to all classes of field problems-these include structural analysis, heat transfer, fluid flow, and electromagnetics. In this book, we will concentrate on linear static structural analysis.

1.2 The Finite Element Process

Finite element analysis seeks to approximate the behavior of an arbitrarily shaped structure under general loading and constraint conditions with an assembly of discrete finite elements. Finite elements have regular (or nearly regular) geometric shapes and known solutions. The behavior of the structure is obtained by analyzing the collective behavior of the elements.

The finite element process is illustrated in **Figure 1-1**.



A REAL STRUCTURE ...

A DISCRETIZED MATHEMATICAL MODEL containing finite elements, loads, constraints, and structural properties ...

YOUR COMPUTER, MSC.Nastran, and a few million calculations ...

DISPLACEMENTS, STRESSES, FORCES, MODE SHAPES ... Whatever you asked for and designed the model to calculate.

VISUALIZATION OF RESULTS

Figure 1-1 The Finite Element Process

1.3 The Basic Matrix Equation of Linear Static Analysis

The Displacement Method. This book is not about theory, but it helps to know what a stiffness matrix is and what equation is being solved by MSC.Nastran. The procedure described in this section is called the displacement method since displacements are the unknown quantities to be calculated.

The Elemental Stiffness Matrix

An element stiffness matrix $[k]$ relates loads acting on an element to displacements resulting from the loads. We will derive by hand the stiffness matrix of the simplest possible element, an extensional elastic rod. Consider an elastic rod of uniform cross sectional area A and length L connecting grid points 1 and 2, as shown in **Figure 1-2**. The rod is subjected to end loads (cross-sectional loads) and is in static equilibrium.

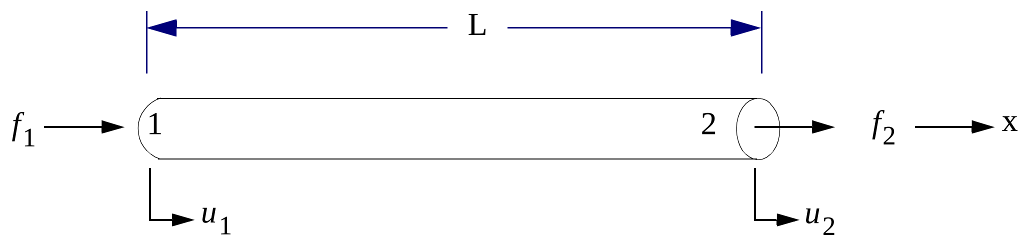


Figure 1-2 Extensional Elastic Rod

Axial translations u_1 and u_2 are the only permitted displacements at grid points 1 and 2. Thus, this element is said to have two degrees of freedom.

Relate Force to Displacement. Our goal is to find an equation relating force to displacement for each degree of freedom. For static equilibrium summing element forces in the x direction requires

$$\sum f_x = f_1 + f_2 = 0$$

or

$$f_2 = -f_1 \qquad \text{Eq. 1-1}$$

Assume that the rod changes length by an amount δ due to the axial load. Strain in the rod ε_x can be related to displacement by the definition of simple strain:

$$\varepsilon_x = \frac{\delta}{L} = \frac{u_2 - u_1}{L} \qquad \text{Eq. 1-2}$$

We assume that the material of the rod is homogeneous (has the same elastic properties at all points), isotropic (has the same elastic properties in all directions at any given point in the body), and linear. For such a material, axial strain ε_x is related to axial stress σ_x by

$$\sigma_x = E\varepsilon_x \qquad \text{Eq. 1-3}$$

where E is the modulus of elasticity of the material. By definition, axial (normal) stress is given by axial force divided by area. Thus,

$$\text{Grid 1:}\quad \sigma_x \quad = \quad -\frac{f_1}{A}$$

and

$$\text{Grid 2:}\quad \sigma_x \quad = \quad +\frac{f_2}{A} \qquad \text{Eq. 1-4}$$

Force can now be related to displacement by substituting **Eq. 1-2** and **Eq. 1-3** into **Eq. 1-4** as follows. For grid point 1,

$$\sigma_x = -\frac{f_1}{A}$$

$$-f_1 = \sigma_x A = E\varepsilon_x A = \frac{EA}{L}(u_2 - u_1)$$

$$-f_1 = \frac{EA}{L}u_2 - \frac{EA}{L}u_1 \qquad \text{Eq. 1-5}$$

Similarly for grid point 2,

$$f_2 = \frac{EA}{L}u_2 - \frac{EA}{L}u_1 \qquad \text{Eq. 1-6}$$

Eq. 1-5 and **Eq. 1-6** represent two linear equations in two unknowns. In matrix form, **Eq. 1-5** and **Eq. 1-6** can be expressed as

$$\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} = \underbrace{\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}}_{[k]}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \qquad \text{Eq. 1-7}$$

or

$$\{f\} = [k]\{u\} \qquad \text{Eq. 1-8}$$

where:

$[k]$ = element stiffness matrix

$\{f\}$ = vector of element forces

$\{u\}$ = vector of element displacements

$[k]$ is the individual rod's stiffness matrix and **Eq. 1-8** is the basic element equation generated and used in the solution process by MSC.Nastran in linear static analysis.

Each type of element has its own elemental stiffness matrix. Stiffness matrices for more complex elements (general beams, plates, and solids) are determined using procedures based on energy principles.

The Global Stiffness Matrix

A structure is modeled as a collection of individual finite elements. An assembly of elemental stiffness matrices representing the structure is called a global stiffness matrix $[K]$. This relationship is illustrated in **Figure 1-3**.

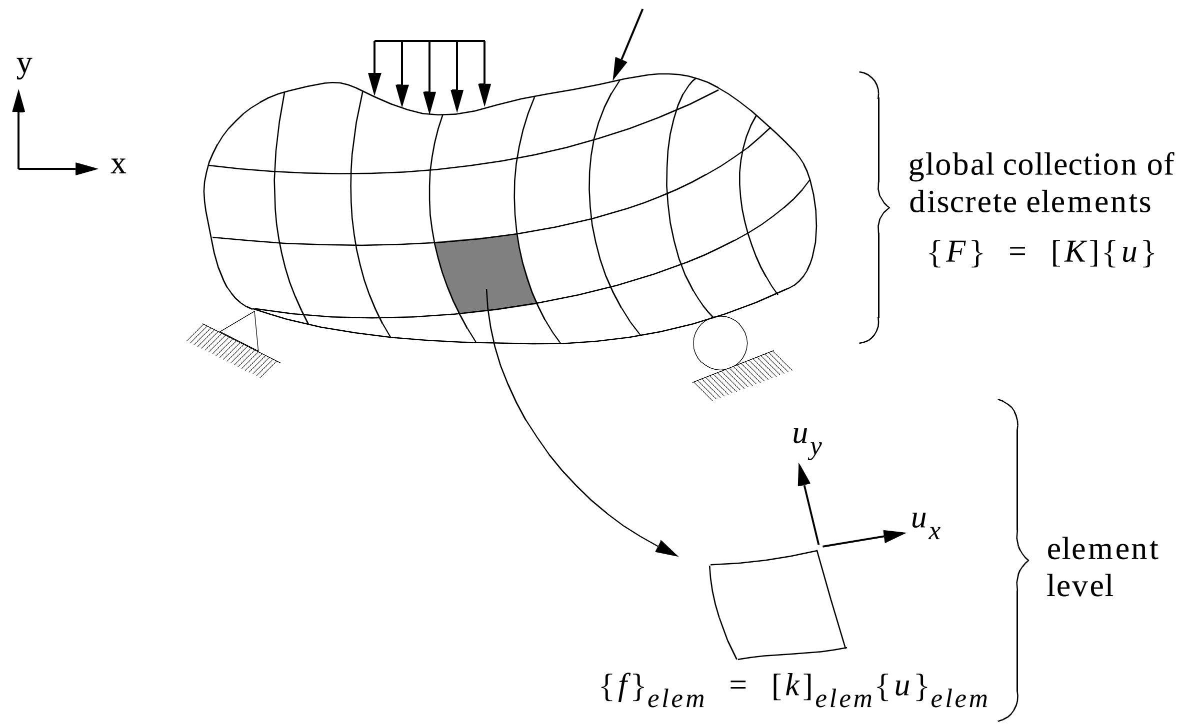


Figure 1-3 The Generalized Finite Element Model

As an example of a global system, consider a structure consisting of the two axial springs with stiffnesses k_a and k_b, and applied forces F_1, F_2, and F_3.

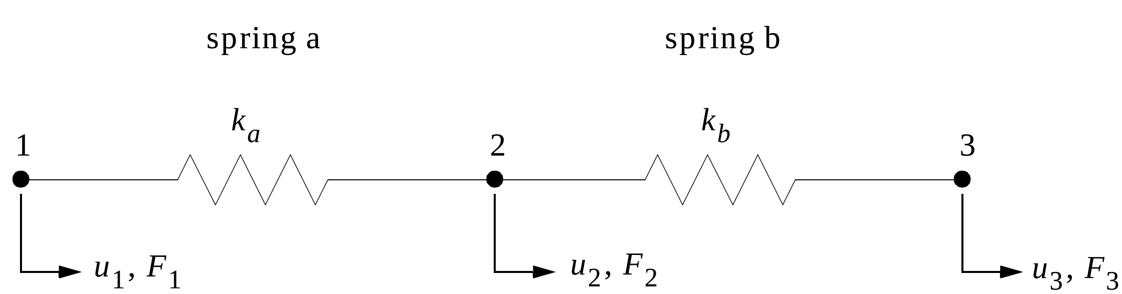
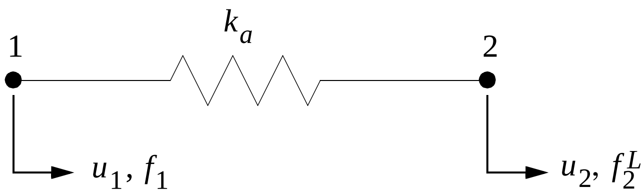


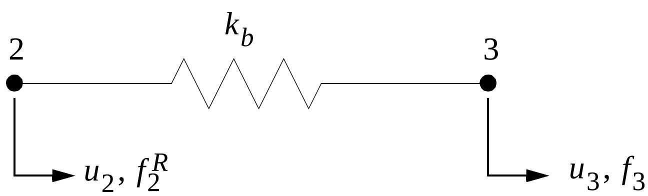
Figure 1-4 Structural System with Two Axial Springs

This assembly has a total of three degrees of freedom (u_1, u_2, and u_3). Create a free-body for each spring and use **Eq. 1-7** to obtain an element matrix equation for each model.



$$\begin{Bmatrix} f_1 \end{Bmatrix} \quad \begin{bmatrix} k_a & -k_a \end{bmatrix} \begin{Bmatrix} u_1 \end{Bmatrix}$$

Eq. 1-9



$$\begin{Bmatrix} f_2^R \\ f_3 \end{Bmatrix} = \begin{bmatrix} k_b & -k_b \\ -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

Eq. 1-10

Combine these last two equations by adding them as follows:

$$\begin{Bmatrix} f_1 \\ f_2^L + f_2^R \\ f_3 \end{Bmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

Eq. 1-11

From static equilibrium at each grid point (free body each grid point) a linear static equilibrium global equation for the system is found to be:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

Eq. 1-12

or

$$\{F\} = [k]\{u\}$$

Eq. 1-13

Of course, this structural system will need to be constrained. For some structural systems there are thousands of linear equations that need to be solved simultaneously.

Solution Flowchart

A flowchart summarizing what MSC.Nastran does in solving a linear static structural problem is shown in **Figure 1-5**.

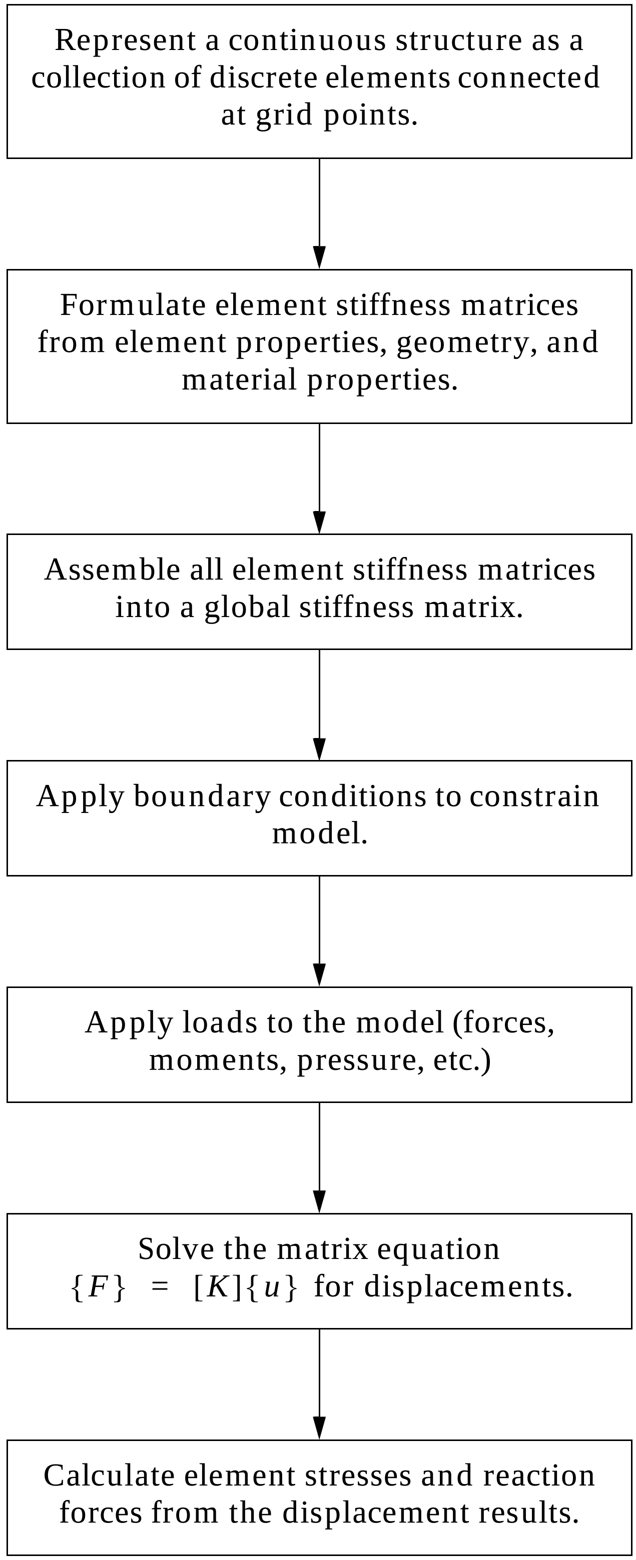


Figure 1-5 MSC.Nastran Solution Flow

1.4 Assumptions and Limitations of Linear Static Analysis

A number of important assumptions and limitations are inherent in linear static analysis. As a finite element analyst, you are responsible for ensuring that these restrictions are understood and accounted for. Failure to do so will result in an analysis that on the surface appears credible, but in reality is not faithful to the structure's physical behavior. Restrictions on linear static analysis are summarized as follows:

LINEAR ELASTIC MATERIAL – Our material is assumed to be homogeneous and isotropic. We are restricted to material in which stress is directly proportional to strain (linear) and to loads that do not take the material beyond its permanent yield point (the material remains elastic). In addition, we assume that the unloaded structure is free of initial or residual stress.

SMALL DISPLACEMENTS – We are restricted to the small displacement assumptions used in the formulation of governing equations for linear beam, plate, shell, and solid behavior and in MSC.Nastran element development. In practice, these assumptions mean lateral plate deflections substantially smaller than the thickness of the plate and beam deflections substantially less than the smallest dimension of the beam's cross section. Violating linear analysis restrictions on small displacements quickly leads to grossly inaccurate displacement results-large displacements require nonlinear analysis methods (see "**Overview of Other MSC.Nastran Capabilities**" on page 7 for further information about geometric nonlinear analysis).

SLOWLY APPLIED LOADS – In linear static analysis our structure is in static equilibrium. Loads must be "slowly applied," which means that they induce no dynamic effects. Some types of loads, such as impact loads, violate this restriction in an obvious way. Some loads are not as obvious. Suppose that you place a brick on the surface of a cantilever beam and then release the brick quickly. The resulting maximum deflection will be greater than the final static equilibrium position. Although impact is not involved, dynamic effects occur. Therefore, "slowly applied" can, for our purposes, be taken to mean a load that does not result in significant dynamic behavior.

CHAPTER
2

Overview of the MSC.Nastran Finite Element Model

This chapter is designed to give you a broad overview-without too much detail-of a simple MSC.Nastran finite element model. Many topics in this chapter will be taken up in greater detail later in the book.

2.1 MSC.Nastran-What Is It?

MSC.Nastran is a general purpose finite element analysis computer program. “General purpose” means that MSC.Nastran addresses a wide range of engineering problem-solving requirements (i.e. beam versus plate structures and various types of response such as statics or dynamics) as compared to specialty programs, which concentrate on particular types of analysis. MSC.Nastran is written primarily in FORTRAN and contains over one million lines of code. MSC Software’s clients lease or purchase executable-only versions of the program. MSC.Nastran is available on a large variety of computers and operating systems ranging from small workstations to the largest supercomputers. Regardless of the computer system used, MSC.Nastran is optimized to run efficiently and provide identical results on every system.

The MSC.Nastran program has evolved over many versions. Each new version contains significant enhancements in analysis capability and numerical performance. In addition, many errors from previous versions are corrected. (No computer program of any degree of sophistication is free of errors-MSC.Software Corporation maintains a detailed and frequently updated list of known errors, including suggestions for alternate approaches.)

MSC.Nastran is composed of a large number of building blocks called modules. A module is a collection of FORTRAN subroutines designed to perform a specific task-processing model geometry, assembling matrices, applying constraints, solving matrix problems, calculating output quantities, conversing with the database, printing the solution, and so on. The modules are controlled by an internal language called the Direct Matrix Abstraction Program (DMAP). Each type of analysis available in MSC.Nastran is called a solution sequence, and each solution sequence is a prepackaged collection of hundreds or thousands of DMAP commands. Once a solution sequence is chosen, its particular set of DMAP commands sends instructions to the modules that are needed to perform the requested solution. All of this happens automatically with no effort on your part beyond choosing a solution procedure (sequence).

2.2 Describing the Structure

This section presents an overview of the various categories of information needed to create a finite element model. They are:

- Coordinate Systems
- Model Geometry
- Finite Elements
- Loads
- Boundary Conditions
- Material Properties

Coordinate Systems

Basic Coordinate System. Before describing the geometry of a structure, you need a coordinate system to put it in. MSC.Nastran has a built-in rectangular Cartesian system called the basic coordinate system, also called the default coordinate system.

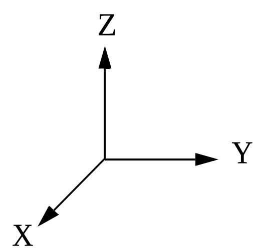


Figure 2-1 The Basic Coordinate System

Local Coordinate System. Depending on the structure, it may be more convenient to input model geometry in other types of coordinate systems. Therefore, MSC.Nastran allows the construction of alternate (local) rectangular, cylindrical, and spherical coordinate systems. As an example, consider the farm silo in **Figure 2-2**. It is cylindrical with a hemispherical dome, and is offset from the the origin of the basic coordinate system. Creating local cylindrical (r, θ, z) and spherical (r, θ, ϕ) systems makes entry of model geometry, or reviewing displacement results from the analysis, far more convenient.

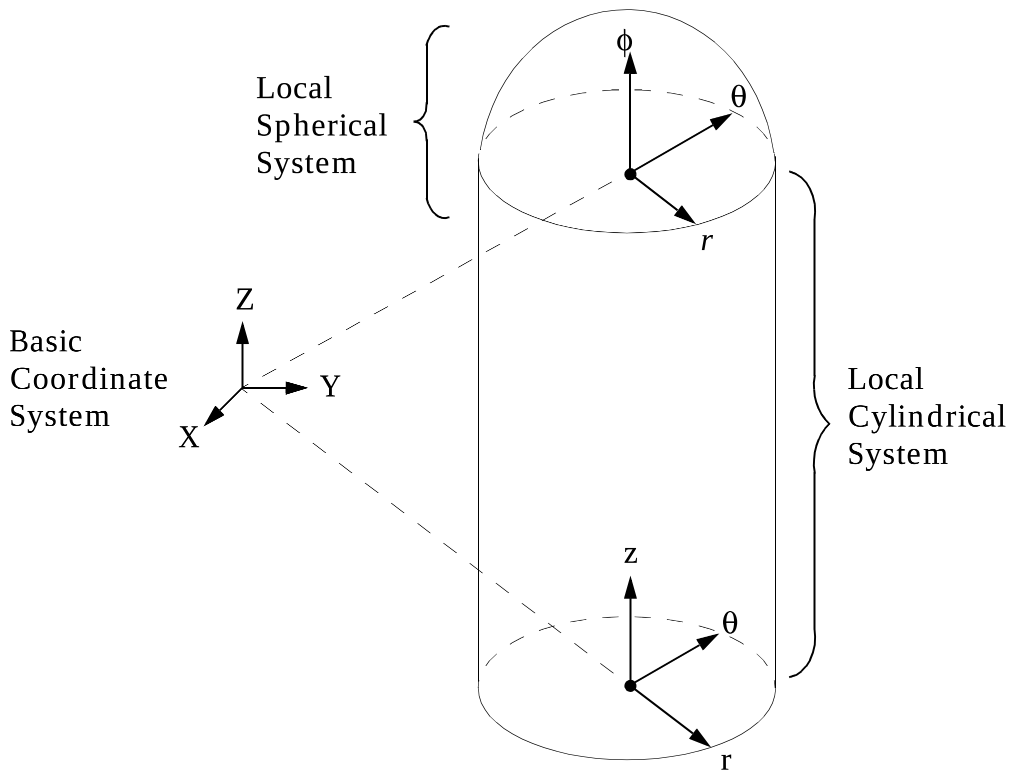


Figure 2-2 Example of Local Coordinate Systems to Aid in Establishing Model Geometry

Model Geometry

Some of the analyst's time is devoted to describing the correct geometry for the model. The information required may come from blueprints, a computer-aided engineering (CAE) system, a computer-aided design (CAD) system, or a sketch on the back of an envelope.

Model geometry is defined in MSC.Nastran with grid points. A grid point is a point on or in the structural continuum which is used to define a finite element. A simple model may have only a handful of grid points; a complex model may have many tens of thousands. The structure's grid points displace with the loaded structure. Each grid point of the structural model has six possible components of displacement: three translations (in the x-, y-, or z-directions) and three rotations (about the x-, y-, or z-axes). These components of displacement are called degrees of freedom (DOFs).

Example. In **Figure 2-3**, which looks down the z-axis, a grid point has undergone two translations (one in the positive x-direction and one in the positive y-direction) and one rotation (about the z-axis).

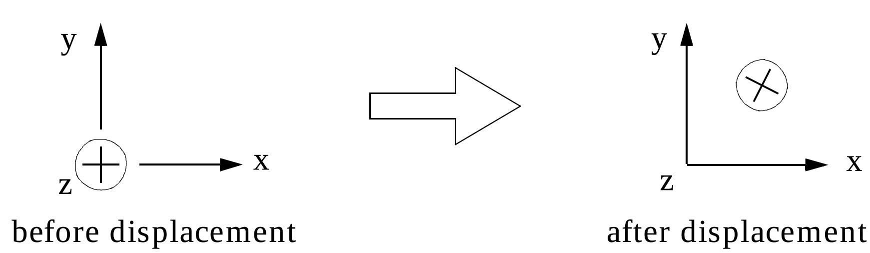


Figure 2-3 Grid Point Displacement

Finite Elements

Once the geometry (grid points) of the structural model has been established, the grid points are used to define the finite elements. In linear static analysis, each element is essentially an elastic spring whose mathematical behavior approximates the physical behavior of a small piece of the actual structure. The overall goal is to make the assembled collection of discrete spring-like elements behave as much like the actual structure as possible. A thorough understanding of the nature of the structure is required to properly choose the type and quantity of elements-no finite element program can relieve you of that responsibility.

MSC.Nastran has an extensive library of finite elements covering a wide range of physical behavior. In this book, we are concerned with the most popular elements used in linear static analysis. These elements and their names are shown in **Figure 2-4**. The C in front of each element name stands for “connection.”

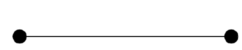
- Point Element (not a finite element, but can be included in the finite element model)

CMASS1 (Scalar mass connection)
CONM1 (Concentrated mass)

- Spring Elements (they behave like simple extensional or rotational springs)

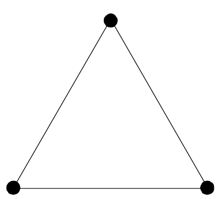
CELAS2

- Line Elements (they behave like rods, bars, or beams)

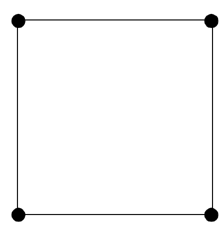


CROD, CBAR, CBEAM

- Surface Elements (they behave like membranes or thin plates)

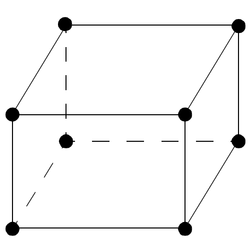


CTRIA3

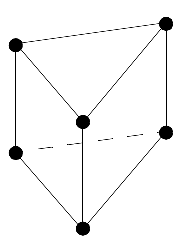


CQUAD4

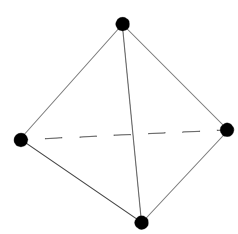
- Solid Elements (they behave like bricks or thick plates)



CHEXA



CPENTA



CTETRA

- Rigid Bar (infinitely stiff without causing numerical difficulties in the mathematical model)

RBE2

Figure 2-4 Some Basic Elements

Loads

MSC.Nastran is capable of modeling many types of loads from a variety of engineering disciplines (static loads, dynamic transients, oscillatory loads, thermal loads, seismic accelerations, and random loads, to name a few). We will concentrate on standard static loads-the types of loads described in typical mechanics of materials classes. These include:

- Concentrated forces and moments.
- Distributed loads on bars and beams.
- Pressure loads on plate and solid surfaces.
- Gravity loads-for example, the response of a structure to its own weight.
- Loads due to acceleration.
- Enforced displacements.

If you are lucky, the loads being applied to a structure are straightforward and well understood. It is often the case, however, that you are required to make assumptions or estimates regarding the nature of the loads. This is particularly true when trying to model, say, a contact patch on a machine part or gear tooth, or wind pressure loads on an irregular surface. It may even be the case that knowledge of the loads represent the largest single source of uncertainty in the model. Since the quality of the overall analysis is no better than the collected uncertainties in the model's input, great care must be taken to represent the loads as accurately as possible.

Boundary Conditions

Structures respond to loads by developing reactions at their point or points of constraint. Some simple Boundary Conditions are shown in **Figure 2-5**:

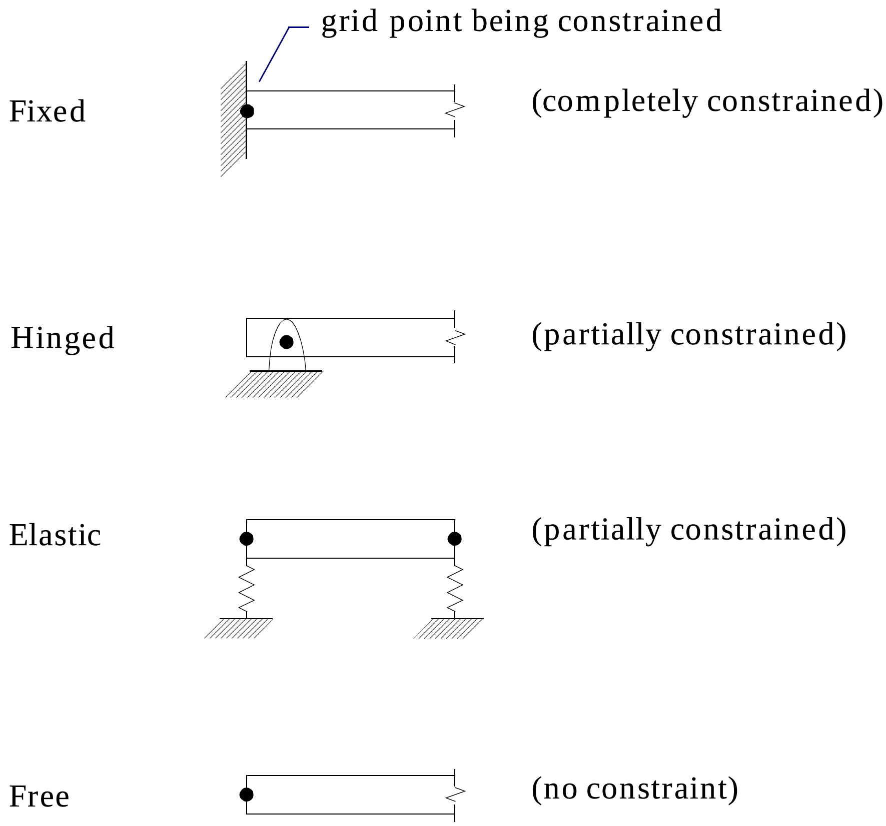


Figure 2-5 Some Simple Boundary Conditions

In most cases, boundary conditions are modeled in MSC.Nastran by constraining appropriate degrees of freedom to zero displacement.

Real-world structures often do not have ideal or simple boundary conditions. The choice of constraints greatly influences the structure's response to loading, so great care must be taken to represent boundary conditions as accurately as possible.

Material Properties

MSC.Nastran can model a wide range of material properties. Options include isotropic, anisotropic, orthotropic, nonlinear (stress-dependent), fluid, temperature-dependent, and composite behavior. In this book we will concentrate on modeling material that is homogenous, isotropic, and temperature independent. In addition, we require that the loading not cause the structure to exceed the elastic limit (i.e., the material remains linear). Fortunately, these restrictions cover a wide range of engineering materials and problems.

2.3 The Structure of the MSC.Nastran Input File

The MSC.Nastran input file contains a complete description of the finite element model, including:

- The type of analysis to be performed.
- The model's geometry.
- A collection of finite elements.
- Loads.
- Constraints (boundary conditions).
- Requests for the type of output quantities to be calculated.

The input file is a text file with a filename and a .DAT extension (e.g., MODEL1A.DAT). The input file can be created with a text editor or finite element preprocessor. To execute MSC.Nastran, the user types a system command followed by the name of the input file (the .DAT extension is not required-MSC.Nastran automatically assigns a .DAT extension to the filename being submitted). A typical command might be:

```
NASTRAN MODEL1A
```

The details of submitting an MSC.Nastran job are specific to your computer system-contact your computer system personnel or your ***MSC.Nastran 2003 Installation and Operations Guide*** for further information.

The input file contains five distinct sections (three of which are required) and three one-line delimiters. The structure of the input file is shown in **Figure 2-6**.

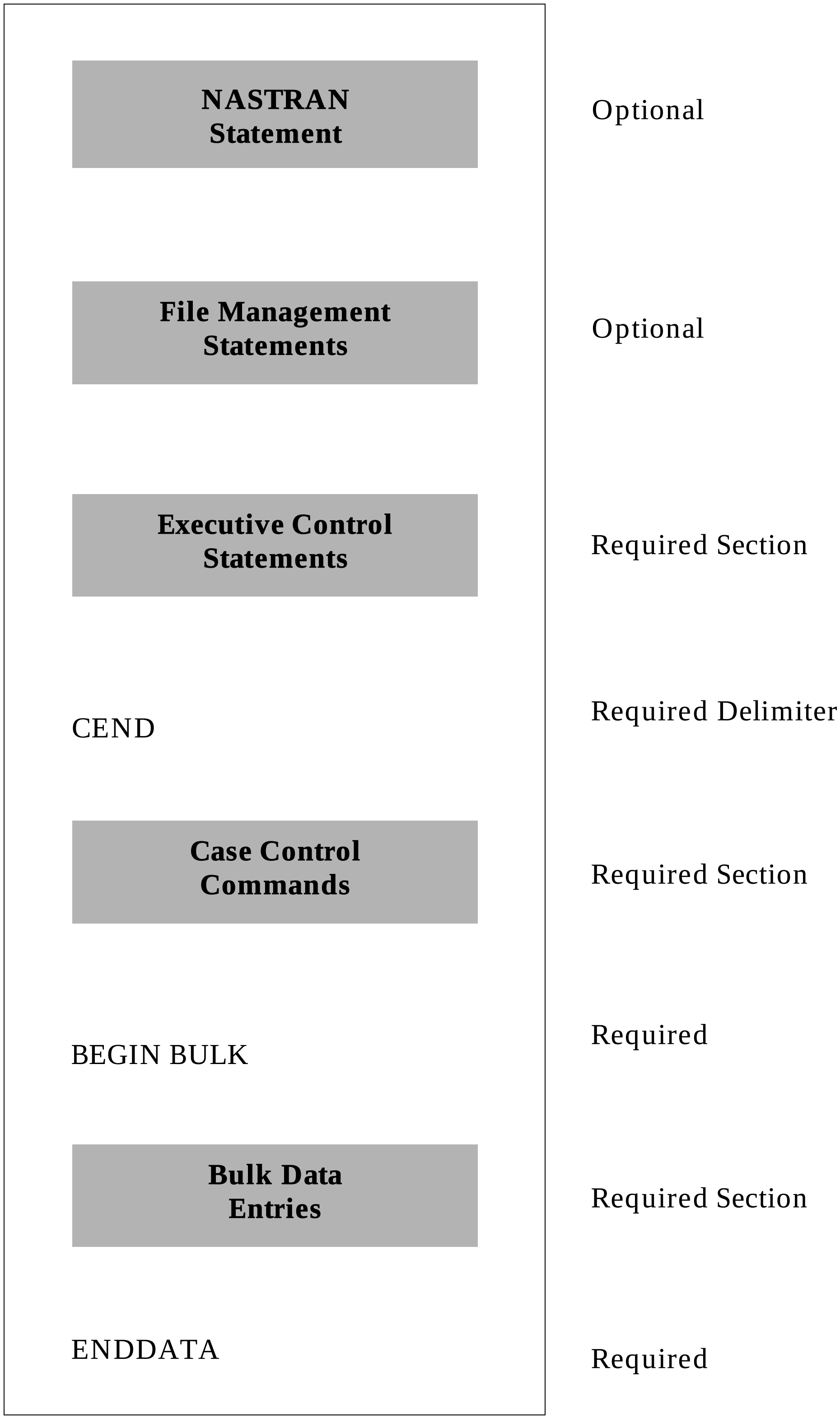


Figure 2-6 Structure of the MSC.Nastran Input File

The function of each section is summarized as follows:

NASTRAN Statement (optional)

The NASTRAN statement is optional and is used to modify certain operational parameters (also called system cells). Examples include aspects of working memory, datablock size, datablock parameters, machine specific issues, numerical methods, etc. The NASTRAN statement is used for exceptional circumstances and is not needed in most runs. Further information may be found in the "**nastran Command and NASTRAN Statement**" on page 1 of the *MSC.Nastran Quick Reference Guide*.

Note that as of Version 69, the NASTRAN statement may also be combined with the File Management Section.

File Management Section (optional)

The File Management Section (FMS) is also optional; it is used primarily to attach or initialize MSC.Nastran databases and FORTRAN files. The initialization of a database includes specification of its maximum size, member names, and physical filenames. The initialization of a FORTRAN file includes the specification of its filename, FORTRAN unit numbers, and FORTRAN attributes.

For many MSC.Nastran problems, no File Management statements are required because a default File Management Section is executed at the beginning of every run. A detailed discussion of the FMS is not required here; further information may be found in the "**File Management Statements**" on page 31 of the *MSC.Nastran Quick Reference Guide*.

Executive Control Section (required)

Entries in the Executive Control Section are called statements. The primary function of this section is to specify the type of analysis solution to be performed (required for all MSC.Nastran jobs). Other common functions include an optional ID statement to identify the job and a TIME statement which sets maximum time limits for execution. The end of the Executive Control Section is signaled by the CEND delimiter.

Case Control Section (required)

Entries in the Case Control Section are called commands. The Case Control Section is used to specify and control the type of analysis output required (e.g., forces, stresses, and displacements). Case Control commands also manage sets of Bulk Data input, define analysis subcases (multiple loadings in a single job execution), and select loads and boundary conditions. The Case Control Section always follows the Executive Control Section and precedes the Bulk Data Section.

Bulk Data Section (required)

The Bulk Data Section always follows the Case Control Section and begins with the BEGIN BULK delimiter. Bulk Data entries contain everything required to describe the finite element model-geometry, coordinate systems, finite elements, element properties, loads, boundary conditions, and material properties. In most analyses this section constitutes the vast majority of the total MSC.Nastran input file. Bulk Data entries can be entered in any order, but the last entry must be the ENDDATA delimiter (note the spelling).

2.4 A Simple MSC.Nastran Model

This section describes the simplest possible MSC.Nastran model-two grid points, one element (an extensional rod), and one concentrated force. You certainly don't need MSC.Nastran to solve a problem like this, but the point is to show you a complete model-the big picture-before moving on to more detail. $F = 20\ lb_f$

Problem Statement. A 0.25 inch diameter steel rod is fixed on one end, with a 20 lb axial force on the other. What is the elongation of the rod due to the axial force?

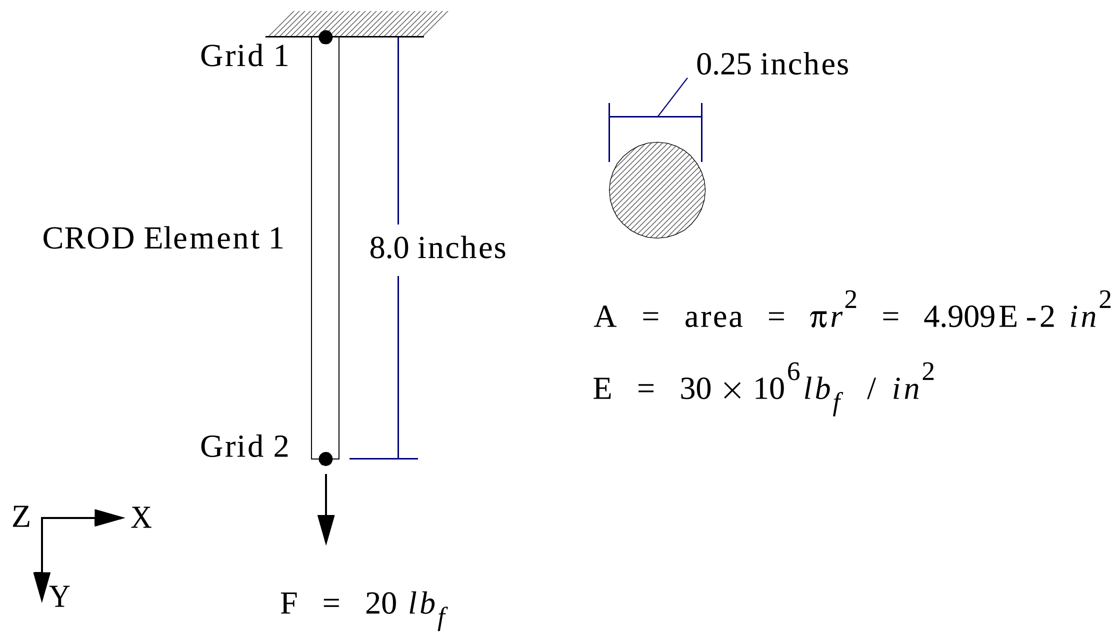


Figure 2-7 Simple Rod Model

The MSC.Nastran input file used to solve this problem is shown in **Figure 2-8**. The name of this input file is ROD.DAT. Don't worry about the details of the input or the result files yet-this material will be covered in later chapters.

Executive Control Section	ID ROD EXAMPLE SOL 101 TIME 5
Required Delimiter	CEND
Case Control Section	LOAD=8 DISP=ALL SPCF=ALL ECHO=BOTH
Required Delimiter	BEGIN BULK
Bulk Data Section	GRID,1,,0.,0.,0.,,123456 GRID,2,,0.,8.0,0.,, FORCE,8,2,,20.,0.,1.,0. CROD,1,15,1,2 PROD,15,5,4.909E-2 MAT1,5,30.E6,,0.3
Required Delimiter	ENDDATA

Figure 2-8 MSC.Nastran Input (.DAT) File

Submitting the Job. The ROD.DAT file is submitted to MSC.Nastran with the command

```
NASTRAN ROD SCR=YES
```

The Results. The following MSC.Nastran results are obtained from the .f06 file:

Listing 2-1 .f06 File Results

```
THIS PROGRAM IS CONFIDENTIAL AND A TRADE SECRET OF MSC.SOFTWARE CORPORATION.  THE RECEIPT OR
POSSESSION OF THIS PROGRAM DOES NOT CONVEY ANY RIGHTS TO REPRODUCE OR DISCLOSE ITS CONTENTS, OR TO
MANUFACTURE, USE, OR SELL ANYTHING HEREIN, IN WHOLE OR IN PART, WITHOUT THE SPECIFIC WRITTEN CONSENT
OF MSC.SOFTWARE CORPORATION.


* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
* *                                  * *
* *           MSC.SOFTWARE           * *
* *               CORP               * *
* *                                  * *
* *      M S C . N a s t r a n       * *
* *                                  * *
* *       VERSION - 2001.0.1         * *
* *                                  * *
* *          SEP 10, 2001            * *
* *                                  * *
* *                                  * *
* *              Intel               * *
* *                                  * *
* *MODEL PentiumIII/995 (MERCED.scm)* *
* *                                  * *
* *  Windows 2000 5.0 (Build 2195)   * *
* *                                  * *
* *                                  * *
* *                                  * *
* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
```

```
                                                                           SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     1

          N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O

     ID ROD EXAMPLE
     SOL 101
     TIME 5
     CEND
```

Listing 2-1 .f06 File Results (continued)

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     2

                                              C A S E    C O N T R O L   D E C K   E C H O
                  CARD
                  COUNT
                   1      LOAD=8
                   2      DISP=ALL
                   3      SPCF=ALL
                   4      ECHO=BOTH
                   5      BEGIN BULK
```

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3

                                                   I N P U T   B U L K   D A T A   D E C K   E C H O

                               .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                               GRID,1,,0.,0.,0.,,123456
                               GRID,2,,0.,8.,0.
                               FORCE,8,2,,20.,0.,1.,0.
                               CROD,1,15,1,2
                               PROD,15,5,4.909E-2
                               MAT1,5,30.E6,,0.3
                               ENDDATA
                               INPUT BULK DATA CARD COUNT =       7
```

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4

                                                     S O R T E D   B U L K   D A T A   E C H O
                  CARD
                  COUNT         .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                     1-         CROD    1       15      1       2
                     2-         FORCE   8       2               20.     0.      1.      0.
                     3-         GRID    1               0.      0.      0.              123456
                     4-         GRID    2               0.      8.      0.
                     5-         MAT1    5       30.E6           0.3
                     6-         PROD    15      5       4.909E-2
                                ENDDATA
                         TOTAL COUNT=         7
```

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     5

  USER INFORMATION MESSAGE
 ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.0 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC
COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES

                                                         OLOAD    RESULTANT

                  T1             T2             T3             R1             R2             R3
        1   0.0000000E+00  2.0000000E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
```

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     6

                                                G R I D   P O I N T   S I N G U L A R I T Y   T A B L E
                               POINT    TYPE   FAILED      STIFFNESS       OLD USET          NEW USET
                                ID             DIRECTION      RATIO     EXCLUSIVE  UNION   EXCLUSIVE  UNION
                                 2        G      1         0.00E+00        B        F         SB       SB     *
                                 2        G      3         0.00E+00        B        F         SB       SB     *
                                 2        G      4         0.00E+00        B        F         SB       SB     *
                                 2        G      5         0.00E+00        B        F         SB       SB     *
                                 2        G      6         0.00E+00        B        F         SB       SB     *
```

```
                                                                            SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     7

 *** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL         LOAD SEQ. NO.            EPSILON              EXTERNAL WORK
EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS

                1          2.2204461E-17          1.0864398E-03
```

Listing 2-1 .f06 File Results (continued)

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     8

  USER INFORMATION MESSAGE
 ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.0 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC
COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
                                                           SPCFORCE RESULTANT

                  T1             T2             T3             R1             R2             R3
        1   0.0000000E+00 -2.0000000E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
```

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     9

                                                    D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0            0.0            0.0            0.0            0.0            0.0
             2      G      0.0            1.086440E-04   0.0            0.0            0.0            0.0
```

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    10

                                       F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0           -2.000000E+01   0.0            0.0            0.0            0.0
```

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    11
```

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    12

 * * * *  D B D I C T   P R I N T  * * * *      SUBDMAP = PRTSUM  ,  DMAP STATEMENT NO.     13

                                          * * * *  A N A L Y S I S  S U M M A R Y  T A B L E  * * * *

  SEID  PEID PROJ VERS APRCH      SEMG SEMR SEKR SELG SELR MODES DYNRED SOLLIN PVALID SOLNL LOOPID DESIGN CYCLE SENSITIVITY
 ------------------------------------------------------------------------------------------------------------------------
     0     0    1    1 '        '    T    T    T    T    T     F      F      T      0     F     -1            0           F

 SEID = SUPERELEMENT ID.
 PEID = PRIMARY SUPERELEMENT ID OF IMAGE SUPERELEMENT.
 PROJ = PROJECT ID NUMBER.
 VERS = VERSION ID.
 APRCH = BLANK FOR STRUCTURAL ANALYSIS.  HEAT FOR HEAT TRANSFER ANALYSIS.
 SEMG = STIFFNESS AND MASS MATRIX GENERATION STEP.
 SEMR = MASS MATRIX REDUCTION STEP (INCLUDES EIGENVALUE SOLUTION FOR MODES).
 SEKR = STIFFNESS MATRIX REDUCTION STEP.
 SELG = LOAD MATRIX GENERATION STEP.
 SELR = LOAD MATRIX REDUCTION STEP.
 MODES = T (TRUE) IF NORMAL MODES OR BUCKLING MODES CALCULATED.
 DYNRED = T (TRUE) MEANS GENERALIZED DYNAMIC AND/OR COMPONENT MODE REDUCTION PERFORMED.
 SOLLIN = T (TRUE) IF LINEAR SOLUTION EXISTS IN DATABASE.
 PVALID = P-DISTRIBUTION ID OF P-VALUE FOR P-ELEMENTS
 LOOPID = THE LAST LOOPID VALUE USED IN THE NONLINEAR ANALYSIS.  USEFUL FOR RESTARTS.
 SOLNL = T (TRUE) IF NONLINEAR SOLUTION EXISTS IN DATABASE.
 DESIGN CYCLE = THE LAST DESIGN CYCLE (ONLY VALID IN OPTIMIZATION).
 SENSITIVITY = SENSITIVITY MATRIX GENERATION FLAG.
```

```
                                              * * * END OF JOB * * *
```

Interpreting the Results. Grid point displacements are shown on page 9 of the MSC.Nastran results (see the highlighted boxes). Grid point 1 has no displacement since it is fixed. Grid point 2 displaces in the y direction (T2), as expected.

The hand-calculated solution is shown below:

$$\text{Axial Elongation} = \frac{\mathrm{PL}}{\mathrm{AE}} = \frac{(20\ lb_f)(8\ in)}{(4.909\mathrm{E}-2in^2)(30\times 10^6 lb_f/in^2)}$$

$$= 1.0864\mathrm{E}-4\ inch$$

Thus, in this very simple case, MSC.Nastran agrees with the analytical solution.

2.5 Files Created by MSC.Nastran

In "**A Simple MSC.Nastran Model**" on page 19, an input text file called ROD.DAT was submitted to MSC.Nastran. Upon successful execution of the job, a variety of files were automatically created:

ROD.DBALL	Contains permanent data for database runs.
ROD.f04	Contains database file information and a module execution summary.
ROD.f06	Contains the MSC.Nastran analysis results, as listed in "**A Simple MSC.Nastran Model**" on page 19 example.
ROD.LOG	Contains system information and system error messages.
ROD.MASTER	The master directory for database runs.

In this book, we will concentrate on the .DAT and .f06 files.

SCR (scratch) Command. If no restarts or database manipulations are planned (which is the case throughout this book), then the MASTER and DBALL files can be automatically deleted (scratched) upon completion of the run by adding the statement SCR=YES to the execution command. For example:

```
NASTRAN   ROD   SCR=YES
```

Failure to delete these files may prohibit subsequent reruns of the same input file.

CHAPTER 3

Basic Modeling Issues

- Units
- Format of the Input Data
- Meshes and Mesh Transitions
- Designing the Model
- Using Test Models
- Pre- and Postprocessors

3.1 Units

MSC.Nastran (or your computer, for that matter) knows nothing about physical units. You must use a consistent set of units in developing a finite element model. It doesn't matter what the system is-English, SI, or a system of your own invention-as long as it agrees with itself.

Important: A natural tendency is to say, "The need for consistent units is obvious! Inconsistent units could never happen to me!" Our client support experience suggests otherwise: this problem has happened to almost everyone who has been in this business for any length of time. In particular, it tends to happen at a Departmental Design Committee meeting attended by (at least) your boss's boss, three weeks before your annual performance review, wherein your vocational tendencies may come into question. The moral is: BE COMPULSIVE ABOUT UNITS.

A common example of using inconsistent units is to specify model geometry in feet (perhaps taken directly from a civil engineering blueprint) and the elastic modulus of the material in pounds per square inch. MSC.Nastran cannot detect the existence of inconsistent units. If inconsistent units are used, wrong answers will occur and no user warning messages will be generated.

Example of consistent systems of units are shown in **Table 3-1** below:

Table 3-1 Consistent Systems of Units

Quantity	English	SI
Input:		
Grid Point Geometry	inch	meter
Elastic Modulus	lb_f / $inch^2$	Newton / $meter^2$
Applied Moment	inch-lb_f	Newton-meter
Applied Force	lb_f	Newton
Mass	lb_f-sec^2 / inch	Kilogram
Time	second	second
Output:		
Displacements	inch	meter
Stresses	lb_f / $inch^2$	Newton / $meter^2$

3.2 Format of the Input Data

Real, Integer, and Character Input Data

MSC.Nastran is quite particular about the input requirements for data entry. The three possible types of data entries are Integer, Real, and Character (sometimes called literal, or BCD-binary coded decimal). The three types of data are described as follows:

Integer	Cannot contain a decimal point.
Real	Must contain a decimal point.
Character	Can be alphanumeric, but must always start with an alpha character and be 8 characters or less in length.

The MSC.Nastran Quick Reference Guide describes the input requirements and default values for all MSC.Nastran data file input. MSC.Nastran will issue an error message if the wrong class of input is used.

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

7.0	.7E1	0.7+1
.70+1	7.E+0	70.-1

Free, Small, and Large Field Formats

MSC.Nastran has three different field formats for input data:

Free Field Format	Input data fields are separated by commas.
Small Field Format	Ten fields of eight characters each.
Large Field Format	Ten fields-fields containing actual data are sixteen characters each. Large fields are used when greater numerical accuracy is required.

The NASTRAN statement, File Management Section, Executive Control Section, and Case Control Section use free field format. The Bulk Data Section allows the use of any of the three formats.

MSC.Nastran Bulk Data contains ten fields per input data entry. The first field contains the character name of the Bulk Data item (e.g., GRID, CBAR, MAT1, etc.). Fields two through nine contain data input information for the Bulk Data entry. The tenth field never contains data-it is reserved for entry continuation information, if applicable.

Consider the format of a typical MSC.Nastran Bulk Data entry, the GRID entry, which is used in MSC.Nastran to describe the geometry of the structural model:

Field Number

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Fields containing input data for the GRID entry.

Character name of this Bulk Data entry.

Field 10 is used only for optional continuation information, when applicable. This shaded box means that a data continuation is not used for the GRID entry.

Example:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

We will now represent this example in free field, small field, and large field formats.

Free Field Format

In free field format, data fields are separated by commas or blanks (commas are strongly recommended). The following shows the GRID Bulk Data entry example in free field format:

```
GRID,2,,1.0,-2.0,3.0,,136
```

The null between the two commas specifies an empty field

The rules for free field format are as follows:

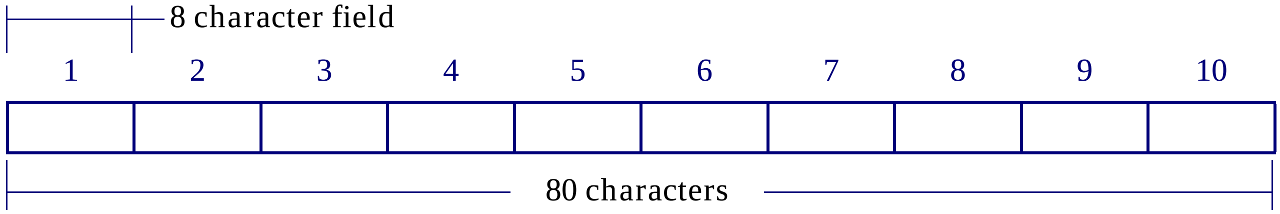
- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If more significant digits are needed, use the large field format.
- Free field data cannot contain embedded blanks. An example of a free field embedded blank is shown below:

```
GRID,2,,1 .0,-2.0,3.0,,136
```

Embedded blank not allowed

Small Field Format

Small field format separates a Bulk Data entry into ten fields of eight characters each:

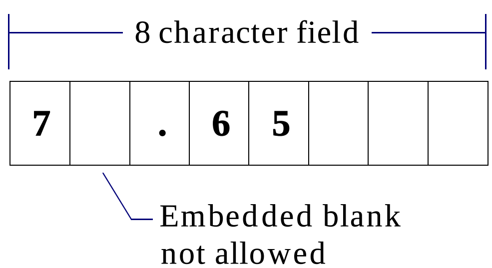


The following is an example of the GRID entry in small field format:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

The rules for small field format are as follows:

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right or left justified, although aligning the data fields is good practice.
- Small field input data cannot contain any embedded blanks. An example of a small field embedded blank is shown below:



Large Field Format

A high degree of numerical accuracy is required in some MSC.Nastran applications. Large field format is used when small field format does not provide enough significant digits (recall that a minus sign, decimal point, and the "E" in scientific notation count as characters).

Large field format requires (at least) two lines for each record, e.g. GRID data. The first and last field of each line contains eight columns, and the fields in between contain 16 columns. Large field entries are denoted by an asterisk (*) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

The following is an example of the GRID Bulk Data entry example in large field format:

First Line:

1A	2	3	4	5	6
GRID*	2		1.0	-2.0	*GRID10
8 Columns	16	16	16	16	8

Second Line:

1B	2	3	4	5	6B
*GRID10	3.0		136		
8 Columns	16	16	16	16	8

Continuations

Some Bulk Data entries require more than eight fields (72 columns) of data. Continuations are required in such cases. To do this, a parent entry (the first line) is followed by one or more continuation entries on subsequent lines. For example, consider the following PBAR simple beam property entry (do not worry about what each field represents-this will be explained later):

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	El	E2	F1	F2	
	K1	K2	I12						

Continuation Example:

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

+PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line. +PB2 in the second line points the third line, and so on.

Continuation fields can also be generated automatically by MSC.Nastran (this approach is the recommended practice). To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank. MSC.Nastran will then generate unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:

Listing 3-1

```
CHEXA,          1,      10,       3,       5,       7,       1,      15,      17,
     ,         19,      13,       4,       6,       8,       2,      10,      11,
     ,         12,       9,      16,      18,      20,      14
```

Output (.f06) file:

Listing 3-2

```
GEAR TOOTH EXAMPLE                                                      SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3

                                             S O R T E D   B U L K   D A T A   E C H O
             ENTRY
             COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..   10  .
                1-        CHEXA   1       10      3       5       7       1       15      17        +000001
                2-        ++00000119      13      4       6       8       2       10      11        +000002
                3-        ++00000212      9       16      18      20      14                        +000003
```

Note that for this example the first character in field 10 of the first line of output (.f06 file) is a blank; There is nothing below the “.”. This, the first field of the second line must be ++000001.

3.3 Meshes and Mesh Transitions

A mesh is the pattern formed by a collection of finite elements. Relatively few elements result in a coarse mesh. Adding more elements produces a finer mesh, which can more closely represent an irregularly shaped structure. In general, a finer mesh is more accurate, but is also more computationally expensive.

Mesh transition regions are required in most finite element models. Mesh transitions serve several purposes:

- They create transition areas that connect fine mesh regions to coarser mesh regions.
- They connect different element types (for example, beam elements to plate elements).
- They make the transitions required to model irregular structural geometry such as fillet welds in a solid element model, or the edge of a circular hole in a plate model.

As a general rule, avoid placing a mesh transition in an area of interest or in an area where there is a large variation in stress (high stress gradient). Ideally, mesh transitions are modeled far away from areas of interest, preferably in regions of relatively uniform stress (low stress gradient).

Transition between different element types (even a transition from CQUAD4 to CTRIA3 elements) can result in local stress anomalies. Normally these are localized and dissipate quickly as you move away from the transition, but be aware that results may be less accurate in an area in or near the transition.

A simple mesh transition in a plate element model is shown in **Figure 3-1**.

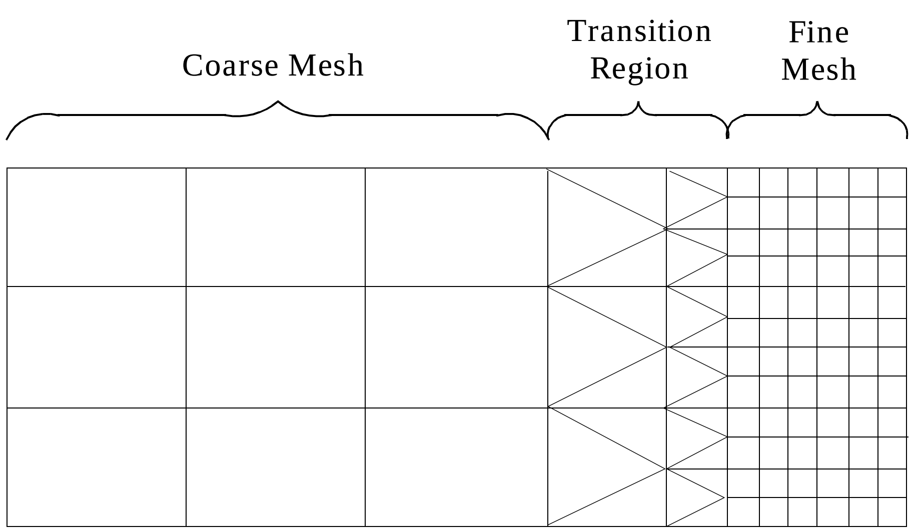


Figure 3-1 Typical Plate Model Mesh Transition

In this example, the transition region consisting of triangular plate elements (CTRIA3s) would tend to be too stiff compared to the adjacent CQUAD4 elements.

3.4 Designing the Model

Considerable engineering judgement about the behavior of the structure is required before the modeling process begins. Since finite element modeling of a complex structure can involve significant engineering and computer resources, a modeling plan is necessary before going anywhere near your computer. Various issues that must be addressed in designing a model are listed below:

- ESTABLISH A PROJECT BUDGET TO HELP YOU MAKE MODELING DECISIONS.

  A project budget takes into account the time available to do the job, labor hours available, and computer resources. Increasing the model's degrees of freedom increases computer costs, modeling time, and time required to interpret the results. For a model with N degrees of freedom, computer costs are divided roughly as follows:

  - Overhead (independent of N)
  - Matrix Assembly (proportional to N)
  - Solution (proportional to N^2)
  - Data Recovery (proportional to N)

  For a statics problem with 1000 degrees of freedom, these quantities are approximately equal. For larger problems, the solution component dominates.

- UNDERSTAND EXACTLY WHAT NEEDS TO BE SOLVED AND HOW ACCURATE THE SOLUTION NEEDS TO BE.

  The question of accuracy is critical to model design, and to a considerable extent involves experience and judgement. Increasing the number of elements generally increases accuracy. For example, 200 elements may be required to obtain a solution that is in error with theory by 15%, but an additional 100 elements may be required to improve the solution to 10%. Thus, it is important to understand how adding elements improves accuracy, but with diminishing returns as you converge toward a solution. More element detail (a finer mesh) is usually required in regions where high stress gradients are expected and where high accuracy is required.

- UNDERSTAND THE STRUCTURE'S PROBABLE MODE OF FAILURE.

  MSC.Nastran will only solve what you tell it to solve. In linear static analysis, for example, you can compress a long, slender column indefinitely, resulting in a short, highly stressed column. In reality, the physical structure will probably buckle under a small compressive load. Buckling analysis is a different solution sequence altogether, requiring eigenvalue methods.

- RECOGNIZE ALL LOADS, LOAD APPLICATION POINTS, AND REACTION POINTS.

  There is often considerable uncertainty associated with knowledge of loads, and boundary conditions are often less than ideal. Therefore, considerable care is required in this step.

- CONSIDER THE FUNCTION OF THE STRUCTURE UNDER LOAD.

This will help you establish the primary load paths for bending, torsion, shear and axial loads. Choose elements based on the expected behavior of the structure.

- IF NECESSARY, PERFORM SENSITIVITY STUDIES with small test models to determine the relationship between the number of elements, solution accuracy, and modeling cost.
- EXPLOIT MODEL SYMMETRY WHENEVER POSSIBLE.

  Symmetry permits the modeling of a single regular segment of the structure. The model's "connection" with the rest of the structure is represented with appropriate boundary conditions.

3.5 Using Test Models

Experiment With Small Test Models. One of the most useful and timesaving methods in finite element work is the use of small test models. When preparing a finite element model involving new or unfamiliar technology-typically an element or type of solution you have never worked with before-small test models involving only a handful of elements should be developed first. These “numerical experiments” give valuable insight into element behavior, proposed modeling techniques, sensitivity of the results to mesh size, or any other technical issue relevant to your analysis. Even the most sophisticated MSC.Nastran analysis methods can be learned and tested on simple models (ideally, create test models that have readily available theoretical solutions). The key point is that new methods should never be learned or tested on large, expensive, commercial models.

Inability to Predict Your Structure’s Response. A second fundamental reason to use test models occurs when you simply have no ability to predict the general behavior or critical regions of your structure, and therefore have no rational basis with which to design a detailed model. Perhaps you have no prior experience with a structure, or its behavior is so complex that engineering intuition is of little use. In such cases, an appropriate strategy is to build a crude initial model (using few elements, but with correct loads and boundary conditions) to gain insight into the gross behavior of the structure. The results of the initial model should aid in selecting elements and in determining where refined mesh regions should be located in the detailed model. When you are in doubt, it is always best to start with a small, inexpensive model.

Help Us Help You. A final note: MSC Software is frequently called upon to help clients with models that do not run correctly. Our technical support staff can help you much more efficiently and effectively if you are working with a small model, since debugging a small model is much easier, and the turnaround time to rerun a (hopefully) corrected test model is minutes rather than hours.

3.6 Pre- and Postprocessors

Developing a finite element model by hand is a time consuming, tedious, and error-prone activity. Making sense of a large stack of finite element computer output (which can easily contain millions of individual numbers in a medium size problem) is also a considerable challenge. A finite element pre- and postprocessor (such as MSC.Patran) is a graphics-based software package primarily designed to aid in the development of a finite element model (preprocessing) and to aid in the display and interpretation of analysis results (postprocessing). In addition, preprocessing software helps the analyst modify the original model if results show that changes and subsequent reanalysis are required. Some preprocessors are able to import geometric data from solids modeling or computer-aided design and manufacturing (CAD/CAM) software to be used as a basis for the finite element model. The processor might be integrated with the analysis software, or it might be a standalone software package. The role of pre- and postprocessors in finite element analysis is illustrated in **Figure 3-2**:

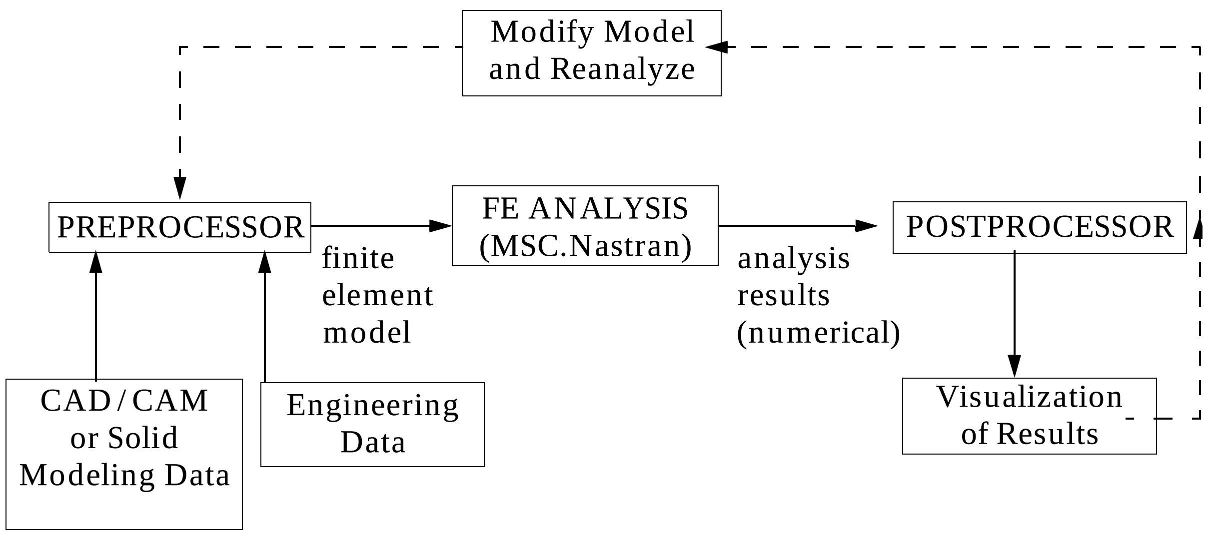


Figure 3-2 Pre- and Postprocessors in Finite Element Analysis

CHAPTER

4 Specifying the Type of Analysis

- Executive Control Section
- Solution Sequences

4.1 Executive Control Section

The Executive Control Section is the first required section in the MSC.Nastran input file. The entries, called statements, are written in free field format. The basic functions of this section are to identify the job, select the type of analysis to be performed, and set limits on the allowable CPU time.

The ID Statement

This optional statement is used to identify the job. If used, the ID statement must go first in the Executive Control Section. Its format is

ID i1, i2

where i1 and i2 are character strings. i1 may be 1 to 8 characters in length. i2 may be of any length. The first character of each string must be alphabetic.

The SOL Statement

This statement is required and is used to select the type of analysis (solution sequence) to be performed. The format of the SOL statement is

SOL n

where n is a positive integer identifying the solution type or the character name of the solution procedure. This book is concerned primarily with SOL 101 (or SOL SESTATIC)-linear static analysis. Many other solution types are available in MSC.Nastran. These are described in "**Solution Sequences**" on page 10.

The TIME Statement

This optional statement sets the maximum CPU time and I/O time for the job. Its format is

TIME t1, t2

where:

t1 = maximum allowed execution time in CPU minutes (real or integer; default=one minute).

t2 = maximum allowable I/O time in seconds (default is infinity)

Note that the default value for execution time is adequate only for very small jobs.

The CEND Statement

The CEND statement is a required statement that designates the end of the Executive Control Section (and the beginning of the Case Control Section). Its format is simply

CEND

A variety of other Executive Control Statements are available, most notably the DIAG statement which offers the user a variety of useful diagnostic information. Discussion of these statements is beyond the scope of this book, but more information can be found in the "**Executive Control Statements**" on page 99 of the *MSC.Nastran Quick Reference Guide.*

Example

Write an Executive Control Section for the linear static analysis of a simple model (a few elements).

```
ID SIMPLE,MODEL
SOL 101
TIME 5
CEND
```

4.2 Solution Sequences

An MSC.Nastran solution sequence (SOL) is a preprogrammed set of instructions designed to solve a particular type of engineering problem.

Some of MSC.Nastran's most commonly used solution sequences are listed in **Table 4-1**.

Table 4-1 Solution Sequences

SOL Number	SOL Name	Description
101	SESTATIC	Statics with options: Linear Heat Transfer Alternate Reduction Inertia Relief Design Sensitivity - Statics
103	SEMODES	Normal Modes with option: Design Sensitivity - Modes
105	SEBUCKL	Buckling with options: Static Analysis Design Sensitivity - Buckling
106	NLSTATIC	Nonlinear Statics
107	SEDCEIG	Direct Complex Eigenvalues
108	SEDFREQ	Direct Frequency Response
109	SEDTRAN	Direct Transient Response
110	SEMCEIG	Modal Complex Eigenvalues
111	SEMFREQ	Modal Frequency Response
112	SEMTRAN	Modal Transient Response
114	CYCSTATX	Cyclic Statics with Option: Alternate Reduction
115	CYCMODE	Cyclic Normal Modes
116	CYCBUCKL	Cyclic Buckling
118	CYCFREQ	Cyclic Direct Frequency Response
129	NLTRAN	Nonlinear Transient Response
144	AESTAT	Static Aeroelastic Response
145	SEFLUTTR	Aerodynamic Flutter
146	SEAERO	Aeroelastic Response
153	NLSCSH	Steady Nonlinear Heat Transfer
159	NLTCSH	Transient Nonlinear Heat Transfer

Table 4-1 Solution Sequences (continued)

SOL Number	SOL Name	Description
190	DBTRANS	Database Transfer
200	DESOPT	Design Optimization

CHAPTER

5

Model Geometry

- Grid Points
- Coordinate Systems

5.1 Grid Points

Grid points are used to define the geometry of a structure. As introduced on "**Model Geometry**" on page 11, each grid point has six degrees of freedom (DOF): three translations and three rotations. The six degrees of freedom are identified as 1, 2, 3, 4, 5, and 6, as shown in **Figure 5-1**.

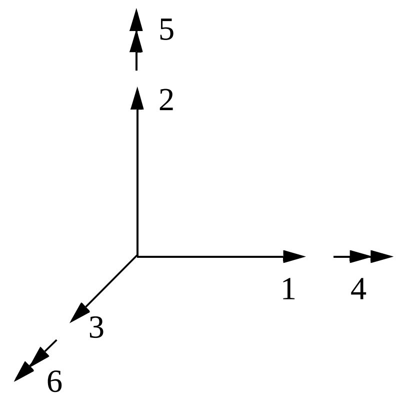


Figure 5-1 DOF Nomenclature

Other commonly used terms for the components of displacement at a grid point are

$$\text{DOF 1} = \text{T1} = U_1 = \text{translation in direction 1}$$

$$\text{DOF 2} = \text{T2} = U_2 = \text{translation in direction 2}$$

$$\text{DOF 3} = \text{T3} = U_3 = \text{translation in direction 3}$$

$$\text{DOF 4} = \text{R1} = \theta_1 = \text{rotation about direction 1}$$

$$\text{DOF 5} = \text{R2} = \theta_2 = \text{rotation about direction 2}$$

$$\text{DOF 6} = \text{R3} = \theta_3 = \text{rotation about direction 3}$$

The format of the GRID Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Field	Contents
ID	Grid point identification number. (0 < Integer < 100000000)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ –1 or blank)

Field	Contents
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank)
SEID	Superelement identification number. (Integer ≥ 0; Default = 0)

Each grid point refers to two coordinate systems. One system is used to locate the grid point (CP in field 3) and the other is used to establish the grid point's displacement (output) coordinate system (CD in field 7). The displacement coordinate system defines the direction of displacements, constraints, and other grid point related quantities such as reaction forces. The basic (default) coordinate system is indicated by a zero or blank in the CP and CD fields. CD and CP do not have to be the same coordinate system.

X1, X2, and X3 (fields 4 through 6) have the following meanings for different types of coordinate systems:

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ (degrees)	Z
Spherical	R	θ (degrees)	ϕ (degrees)

PS in field 8 allows you to apply constraints to any or all of the grid point's DOFs. See "**Single Point Constraints**" on page 10 for more information. Field 9 applies only to superelement analysis (superelement analysis is described in "**Techniques for Analyzing Large Models**" on page 8).

5.2 Coordinate Systems

The Basic Coordinate System

MSC.Nastran has a built-in rectangular coordinate system called the basic coordinate system, shown in **Figure 5-2**.

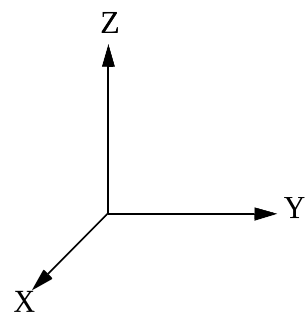


Figure 5-2 The Basic Coordinate System

All coordinate systems have a coordinate system identification number (CID). The basic coordinate system's identification number is zero (0) or blank. The orientation of this system is implied by the user by specifying the components of grid point locations. The basic system is used as a reference when establishing user-defined local coordinate systems.

Local Coordinate Systems

MSC.Nastran provides six Bulk Data entry options for defining local coordinate systems. Each local system must be related directly or indirectly to the basic coordinate system. The six options are:

CORD1R CORD2R	Rectangular
CORD1C CORD2C	Cylindrical
CORD1S CORD2S	Spherical

The CORD1R, CORD1C, and CORD1S entries define a local coordinate system by referencing three defined grid points. Be aware that if the model is modified and any of these reference grid point locations change, the coordinate system orientation will also change. The CORD2R, CORD2C, and CORD2S entries define a local coordinate system by specifying the location of three points.

We will examine the CORD2C Bulk Data entry in detail. Its format is as follows:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default (zero) is the basic coordinate system)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

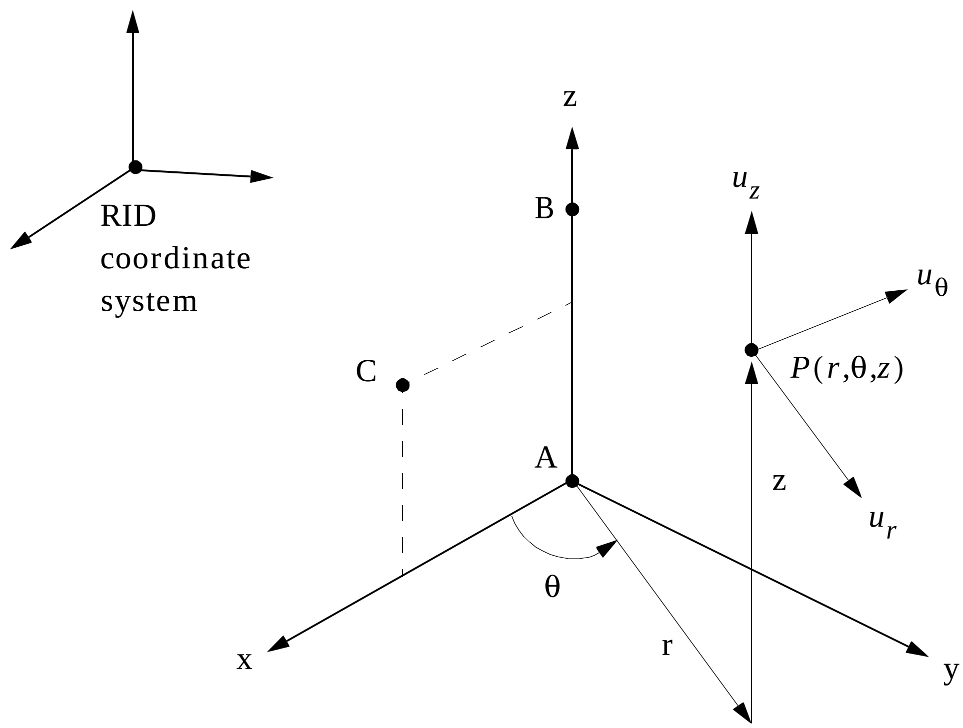


Figure 5-3 CORD2C Definition

The coordinate system identification number (CID) must all be unique with respect to all other CIDs. The three points A (A1, A2, A3), B (B1, B2, B3), and C (C1, C2, C3) must be unique and noncolinear. Noncolinearity is checked by MSC.Nastran's geometry processor. The first point (A) defines the origin. The second point (B) defines the direction of the z-axis. The third point (C) lies in the plane of the azimuthal origin. The reference coordinate system (RID coordinate system) must be independently defined. If RID is zero or blank, the basic coordinate system is used. The continuation entry is required. The location of a grid point (P in **Figure 5-3**) in this coordinate system is given by (r, θ, z) where θ is measured in degrees. If this coordinate system is used to specify the displacement directions they are given by u_r, u_θ and u_z (the displacements are u_r, u_θ and u_z).

Two final points:

1. In any coordinate system, all angular input for grid location is in degrees, but output (such as rotational displacement) is in radians.

2. MSC.Nastran refers to the collection of all displacement coordinate systems referenced by all grid point entries as the global coordinate system. Note that some other finite element programs use the term "global coordinate system" to refer to their equivalent of MSC.Nastran's basic coordinate system.

The following example illustrates the use of the CORD2C entry.

An arch with a semicircular top is shown in **Figure 5-4**. To facilitate grid point input, we wish to establish a local cylindrical coordinate system for grid points 3 through 7.

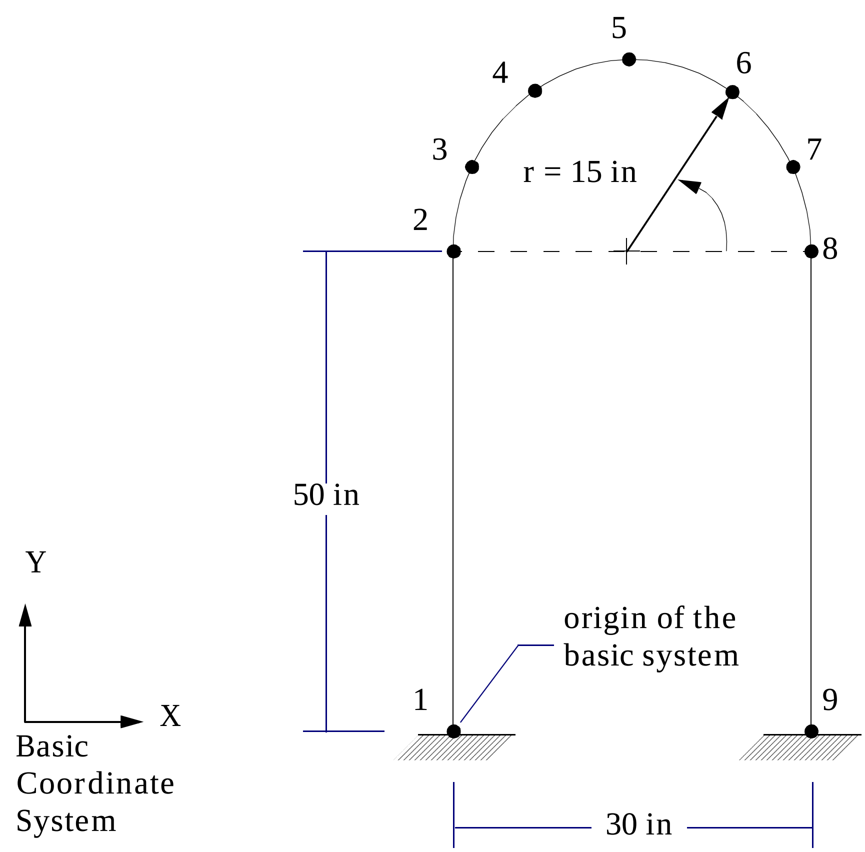


Figure 5-4 Arch Structure

A cylindrical coordinate system with an ID of 100 is defined as shown in **Figure 5-5**.

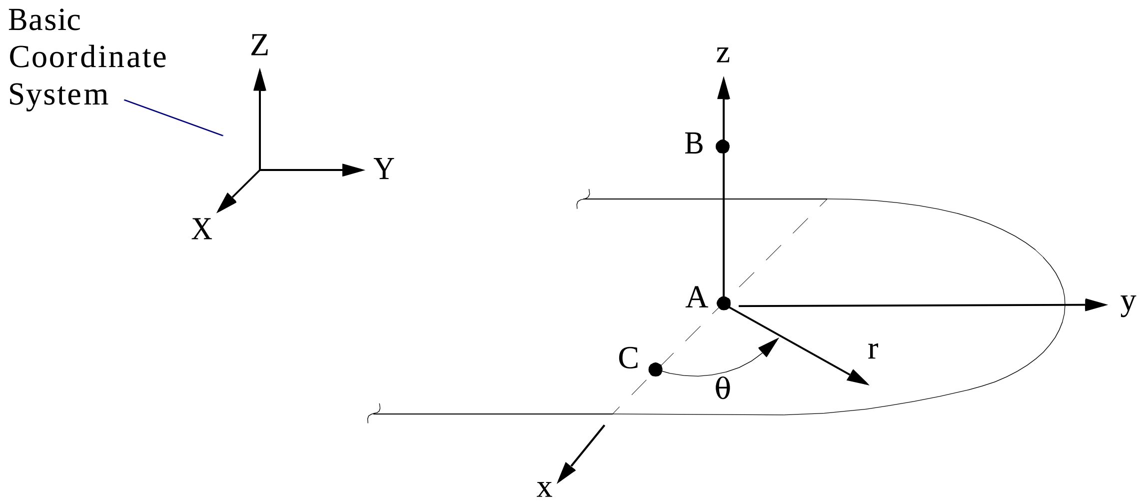


Figure 5-5 Definition of Local Cylindrical Coordinate System

The local cylindrical coordinate system is defined by referencing the basic coordinate system. Three points are required to define the local system. Point A is at the origin and point B lies on the z-axis of the new system. Point C defines the reference axis at which $\theta = 0°$. A CORD2C entry is used in which the last character C indicates its cylindrical nature (S indicates spherical and R rectangular). This coordinate system has an identification number in field CID which will be referenced by other entities such as the CP and CD fields of a GRID entry. In this example, the coordinates of points A, B, and C are in the basic coordinate system.

The cylindrical coordinate system is defined on a CORD2C entry as follows:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						
CORD2C	100		15.	50.	0.	15.	50.	1.	
	16.	50.	0.						

point C point A point B

The GRID entries are defined as follows:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.				
GRID	2		0.	50.	0.				
GRID	3	100	15.	150.	0.				
GRID	4	100	15.	120.	0.				
GRID	5	100	15.	90.	0.				
GRID	6	100	15.	60.	0.				

1	2	3	4	5	6	7	8	9	10
GRID	7	100	15.	30.	0.				
GRID	8		30.	50.	0.				
GRID	9		30.	0.	0.				

Grid points 3 through 7 use (r, θ, z) coordinates with r = 15.0 inches and θ varying from 30 degrees (GRID 7) to 150 degrees (GRID 3). The output for all grid points is in the basic rectangular coordinate system since field 7 is left blank.

For information regarding the other five coordinate system Bulk Data entries, refer to the "**Bulk Data Entries**" on page 849 of the *MSC.Nastran Quick Reference Guide*.

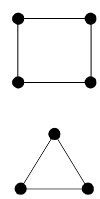
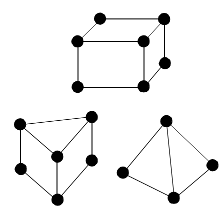
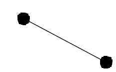
CHAPTER

6 The Basic Element Library

6.1 Introduction

MSC.Nastran offers an extensive variety of general purpose and specialty finite elements. We will discuss the essential aspects of the most common and widely used of these elements, which are categorized in **Table 6-1**:

Table 6-1 The Basic MSC.Nastran Elements

Category	Spring Elements	Line Elements	Surface Elements	Solid Elements	Rigid Elements
Physical Behavior	Simple Spring	Rod, Bar, Beam	Membrane, Thin Plate	Thick Plate, Brick	Rigid Bar
MSC.Nastran Element Name	CELAS2*	CONROD* CROD CBAR	CQUAD4 CTRIA3	CHEXA CPENTA CTETRA	RBE2*
Associated Property Entry	None Required	PROD PBAR	PSHELL	PSOLID	None Required
					

* A separate property entry is not required-it is built directly into the element entry.

Several general notes apply to all MSC.Nastran elements:

- All elements in your model should have unique element ID numbers. Do not reuse element IDs on different element types.
- The formulation of an element's stiffness matrix is independent of how you number the element's grid points.
- Each element has its own element coordinate system defined by connectivity order or by other element data. Element information (such as element force or stress) is output in the element coordinate system.
- MSC Software continually improves the performance of elements in MSC.Nastrans's library, consequently, you may observe changes in numerical results (for equivalent models) in subsequent versions of the program.
- Additional details concerning the features and use of each of MSC.Nastran's elements can be found in "**Bulk Data Entries**" on page 849 of the *MSC.Nastran Quick Reference Guide.*

6.2 Spring Element (CELAS2)

Spring elements, also called zero-dimensional or scalar elements, connect two degrees of freedom, with one at each grid point. They behave like simple extension/compression or rotational (e.g. clock) springs, carrying either force or moment loads. Forces result in translational (axial) displacement and moments result in rotational displacement.

CELAS2

The CELAS2 element defines a spring and includes spring property data directly on the element entry. The format of the CELAS2 entry is shown below:

Format:

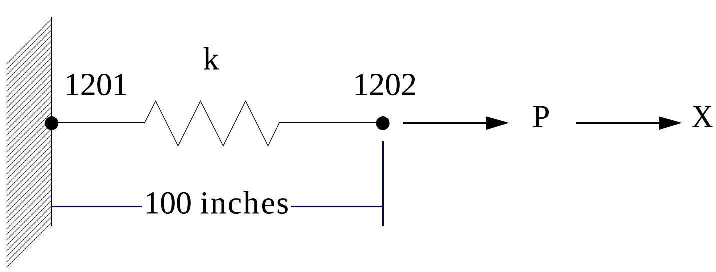
1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point or scalar identification number. (Integer ≥ 0)
C1, C2	Component number. ($0 \leq$ Integer ≤ 6; blank or zero if scalar point)
GE	Damping coefficient. (Real)
S	Stress coefficient. (Real)

Entering a zero or blank for either (Gi, Ci) pair indicates a grounded spring. A grounded point is a point whose displacement is constrained to zero. Also, G1 and G2 cannot be the same GRID point (same ID), but can be different grid points that occupy the same physical point in the structure (strange, but legal). Thus, spring elements do not have physical geometry in the same sense that beams, plates, and solids do, and that is why they are called zero dimensional.

Example:

Consider the simple extensional spring shown in the following figure. One end is fixed and the other is subjected to a 10 lb_f axial load. The axial stiffness of the spring (k) is 100 lb_f/inch. What is the displacement of GRID 1202?



The required Bulk Data entries are specified as follows:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	
CELAS2	1200	100.	1201	1	1202	1			

GRID	1201		0.	0.	0.		123456		
GRID	1202		100.	0.	0.		23456		

GRID 1201 at the fixed wall is constrained in all 6 DOFs. GRID 1202 is constrained in DOFs 2 through 6 since the element it is connected to only uses DOF 1 (translation in the X-direction). Recall that a grid point is free in all six DOFs until it is told otherwise. Leaving any DOF of any GRID point “unattached”-either unconnected to an element’s stiffness or unconstrained by other means-results in a rigid body motion singularity failure in static analysis. The PARAM,AUTOSPC feature of Solution 101 automatically constrains these unconnected DOFs.

Note also that damping (GE in field 8) is not relevant to a static analysis and is therefore not included on this entry. The stress coefficient (S) in field 9 is an optional user-specified quantity. Supplying S directs MSC.Nastran to compute the spring stress using the relation $P_s = S \cdot P$, where P is the applied load.

The grid point displacement and element force output is shown in **Figure 6-1**.

```
                                             D I S P L A C E M E N T   V E C T O R
POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
    1201      G      0.0            0.0            0.0            0.0            0.0            0.0
    1202      G      1.000000E-01   0.0            0.0            0.0            0.0            0.0
```

```
                            F O R C E S   I N   S C A L A R   S P R I N G S       ( C E L A S 2 )
ELEMENT        FORCE            ELEMENT        FORCE          ELEMENT        FORCE          ELEMENT        FORCE
  ID.                             ID.                           ID.                           ID.
   1200    -1.000000E+01
```

Figure 6-1 CELAS2 Spring Element Output

The displacement of GRID 1202 is 0.1 inches in the positive X-direction (the spring is in tension). The hand calculation is as follows:

$$u_x = \frac{P}{k} = \frac{10.lb_f}{100.lb_f\ /\text{inch}} = 0.1\ \text{inch}$$

The force in the spring element is calculated by MSC.Nastran as

$$f = k(u_x^1 - u_x^2) = 100.\ (0. - 0.1) = -10.0\ lb_f$$

where:

u_x^1 = displacement of G1

u_x^2 = displacement of G2

Note that reversing the order of G1 and G2 on the CELAS2 entry would reverse the sign of the element force.

This brings up the crucial point that THE SIGN OF FORCE (AND STRESS) OUTPUT FOR SCALAR ELEMENTS DEPENDS ON HOW THE GRID POINTS ARE LISTED (ORDERED) in defining an element, and not on a physical sense of tension or compression. This is not the case when using line (one-dimensional) elements such as rods and beams. Thus, be careful of how you interpret signs when using scalar elements.

6.3 Line Elements

Line elements, also called one-dimensional elements, are used to represent rod and beam behavior. A rod element supports tension, compression, and axial torsion, but not bending. A beam element includes bending. MSC.Nastran makes an additional distinction between “simple” beams and “complex” beams. Simple beams are modeled with the CBAR element and require that beam properties do not vary with cross section. The CBAR element also requires that the shear center and neutral axis coincide and is therefore not useful for modeling beams that warp, such as open channel sections. Complex beams are modeled with the CBEAM element, which has all of the CBAR’s capabilities plus a variety of additional features. CBEAM elements permit tapered cross-sectional properties, a noncoincident neutral axis and shear center, and cross-sectional warping. The CBEAM element’s additional capabilities are beyond the scope of this book-we will concentrate on rod and bar elements.

Rod Element (CONROD)

CONROD

The CONROD element shown in **Figure 6-2** has two grid points, one at each end, and supports axial force and axial torsion. Thus, stiffness terms exist for only two DOFs per grid point. All element connectivity and property information is contained directly on the CONROD entry-no separate property entry is required. This element is convenient when defining several rod elements having different properties.

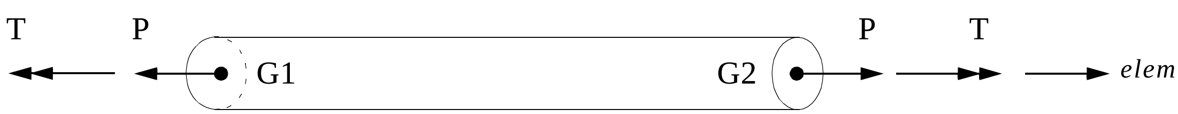


Figure 6-2 CONROD Element Convention

The CONROD element x-axis (x_{elem}) is defined along the line connecting G1 to G2. Torque T is applied about the x_{elem} axis in the right-hand rule sense. Axial force P is shown in the positive (tensile) direction.

The format of the CONROD entry is as follows:

Format:

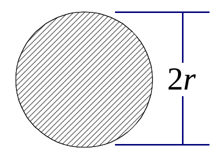
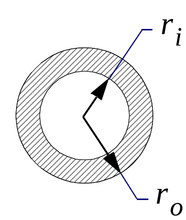
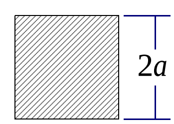
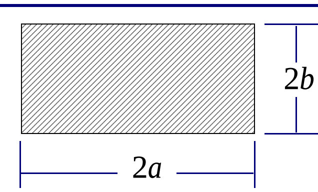
1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Field	Contents
EID	Unique element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; $G1 \neq G2$)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)

Field	Contents
C	Coefficient for torsional stress determination. (Real)
NSM	Nonstructural mass per unit length. (Real)

MID in field 5 points to a MAT1 material property entry. Equations used to calculate the torsional constant J (field 7) are shown in **Table 6-2** for a variety of cross sections.

Table 6-2 Torsional Constant J for Line Elements

Type of Section	Formula for J	Cross Section
Solid Circular	$J = \frac{1}{2}\pi r^4$	
Hollow Circular	$J = \frac{1}{2}\pi(r_o{}^4 - r_i{}^4)$	
Solid Square	$J = 2.25\ a^4$	
Solid Rectangular	$J = ab^3\left[\frac{16}{3} - 3.36\frac{b}{a}\left(1 - \frac{b^4}{12a^4}\right)\right]$	

The torsional stress coefficient C (field 8) is used by MSC.Nastran to calculate torsional stress according to the following relation:

$$\tau = C\frac{M_\theta}{J} \qquad \text{Eq. 6-1}$$

Rod Element (CROD)

CROD

The CROD element is the same as the CONROD element, except that its element properties are listed on a separate Bulk Data entry (the PROD rod element property). This element is convenient when defining rod elements having the same properties.

The CROD element is defined as follows:

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PROD entry. (Integer > 0; Default is EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; $G1 \neq G2$)

Rod Element Property (PROD)

PROD The PROD entry supplies element properties to CROD elements, and is defined as follows:

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)

An example rod element problem is given in "**A Simple MSC.Nastran Model**" on page 19.

Simple Beam Element (CBAR)

CBAR Element Characteristics

CBAR The CBAR element is a general purpose beam that supports tension and compression, torsion, bending in two perpendicular planes, and shear in two perpendicular planes. The CBAR uses two grid points, and can provide stiffness to all six DOFs of each grid point. The displacement components of the grid points are three translations and three rotations.

The characteristics and limitations of the CBAR element are summarized as follows:

- Its formulation is derived from classical beam theory (plane sections remain plane).
- It must be straight and prismatic (properties cannot vary along the length).
- The shear center and neutral axis must coincide (the CBAR element cannot model warping of open sections).
- Torsional stiffening of out-of-plane cross-sectional warping is neglected.
- It includes optional transverse shear effects (important for short beams).
- The principal axis of inertia need not coincide with the element axis.
- The neutral axis may be offset from the grid points (an internal rigid link is created). This is useful for modeling stiffened plates or gridworks.
- A pin flag capability is available to provide a moment or force release at either end of the element (this permits the modeling of linkages or mechanisms).

A careful discussion of offsets and pin flags is beyond the scope of this book-see the following entries ("**CBAR**" on page 1024 and "**CBEAM**" on page 1031 of the) or the *MSC.Nastran Linear Static Analysis User's Guide* for further details.

CBAR Format

Two formats of the CBAR entry are available, as shown below:

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Alternate Format:

CBAR	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PBAR entry. (Integer > 0 or blank; Default is EID unless BAROR entry has nonzero entry in field 3)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; $GA \neq GB$).
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real).

Field	Contents
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. (Integer > 0; G0 $\neq$ GA or GB)
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees of freedom of the bar. The degrees of freedom are defined in the element's coordinate system. The bar must have stiffness associated with the PA and PB degrees of freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0)
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$, respectively in displacement coordinate systems at points GA and GB, respectively. (Real or blank).

PID in field 3 points to a PBAR element property entry. Grid points GA and GB are connected by the element. X1, X2, and X3 are the components of orientation vector $\vec{v}$. Vector $\vec{v}$ describes how the beam's cross section is oriented with respect to the rest of the model. The continuation entry, which is optional, contains data for pin flags and offsets.

CBAR Element Coordinate System

The CBAR element coordinate system is a common source of difficulty for the new user. It is also critical to your model, since the beam's cross sectional moments of inertia are defined (on the PBAR entry) using the element coordinate system. For example, consider two possible orientations of the same rectangular beam (no offsets), as shown in **Figure 6-3**.

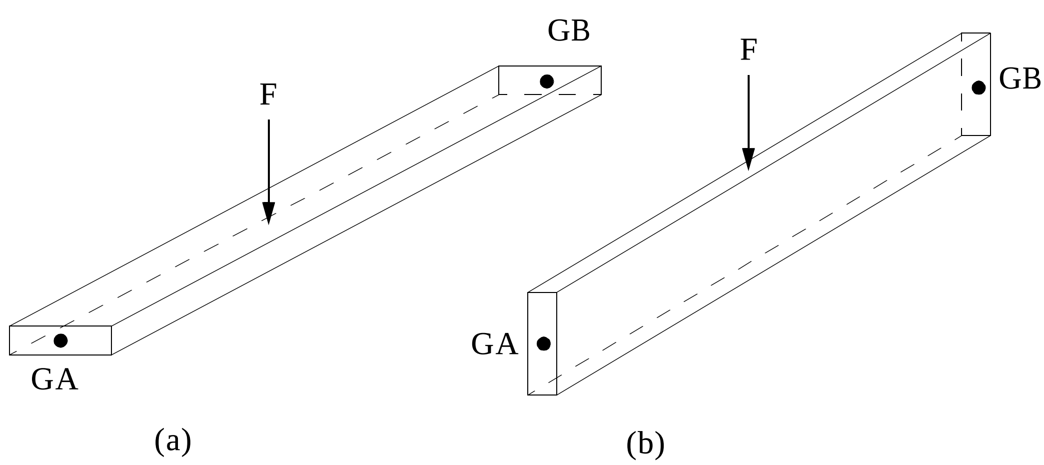
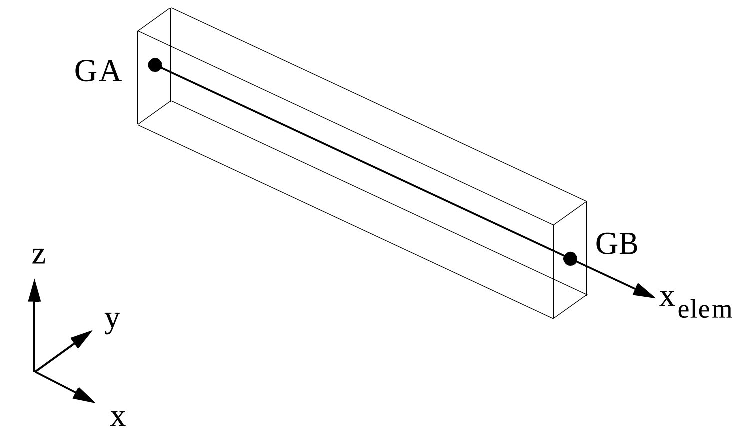


Figure 6-3 Two Orientations of the Same Beam

Although beams (a) and (b) are physically identical, their different orientations result in dramatically different load carrying abilities. In this sense, they are completely different structures. Therefore, it is critical to orient beam elements correctly.

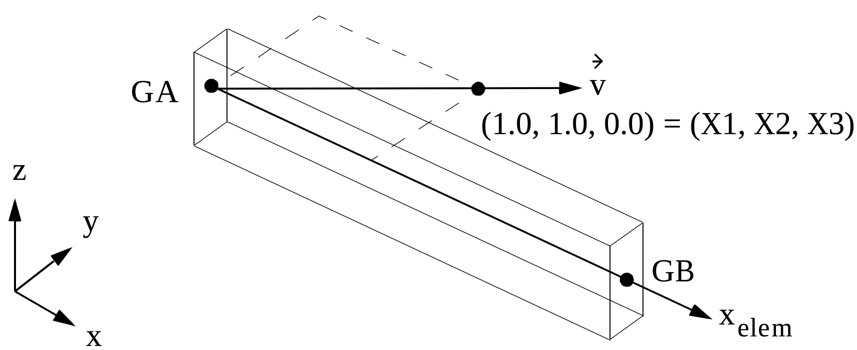
The CBAR element coordinate system is described as follows in **Figure 6-4**:

STEP 1 The element x-axis is automatically defined as the direction from GA to GB. The axis begins at GA:



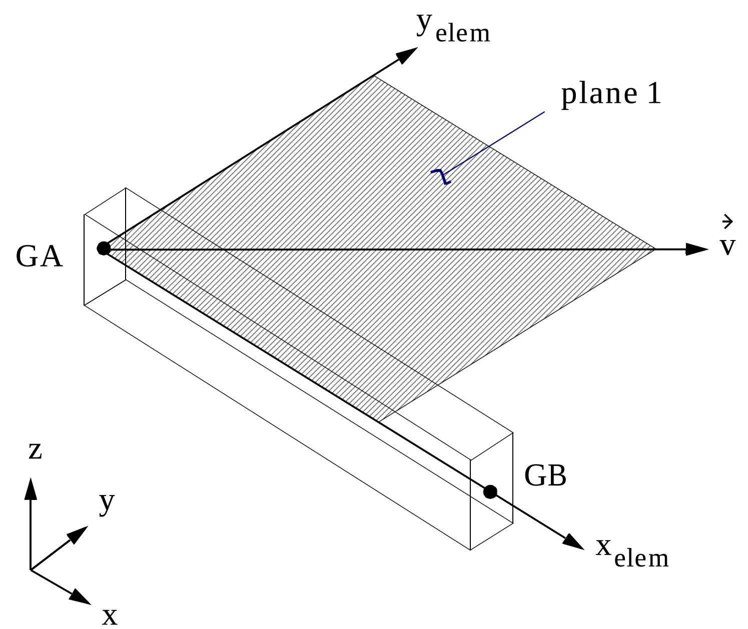
Displacement Coordinate System

STEP 2 Next, we choose a direction for the beam's orientation vector $\vec{v}$. Vector $\vec{v}$ starts at GA and contains the point (X1, X2, X3). X1, X2, and X3-which are defined in the displacement coordinate system of GA-are entered on fields 6, 7, and 8 of the CBAR entry. The direction of $\vec{v}$ with respect to the cross section is arbitrary, but $\vec{v}$ is normally aligned with one of the beam's principal planes of inertia. A choice of (1.0, 1.0, 0.0) gives the following $\vec{v}$:



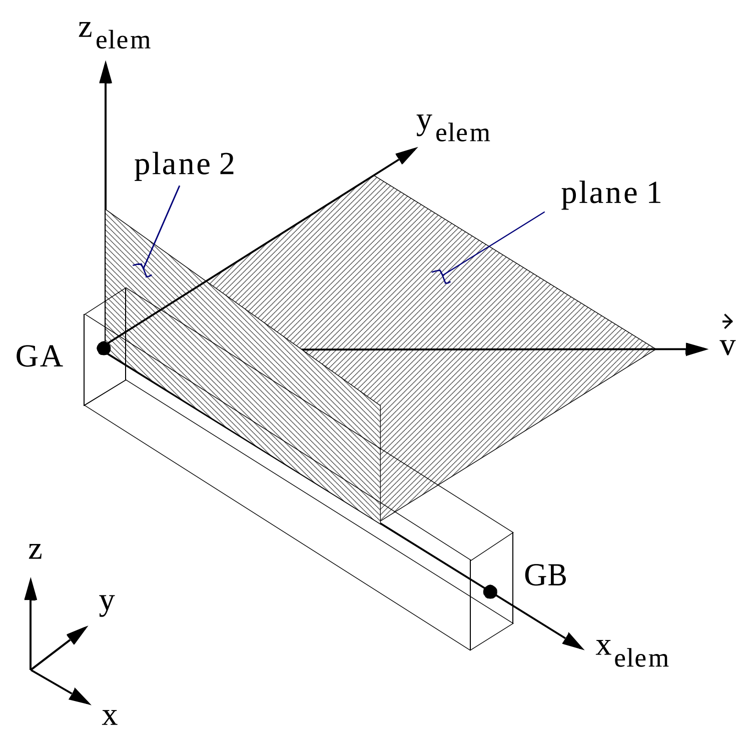
Displacement Coordinate System

STEP3 The plane formed by the element x-axis and orientation vector $\vec{v}$ is called plane 1. The element y-axis lies in plane 1 and is perpendicular to the element x-axis, as shown below:



Displacement Coordinate System

STEP 4 Finally, plane 2 is perpendicular to plane 1, and the element z-axis is formed by the cross product of the x and y element axes. Plane 2 contains the element x and z axes.



Displacement Coordinate System

Figure 6-4 CBAR Element Coordinate System

CBAR Force and Moment Conventions

The CBAR element force and moment conventions are shown in **Figure 6-5** and **Figure 6-6**.

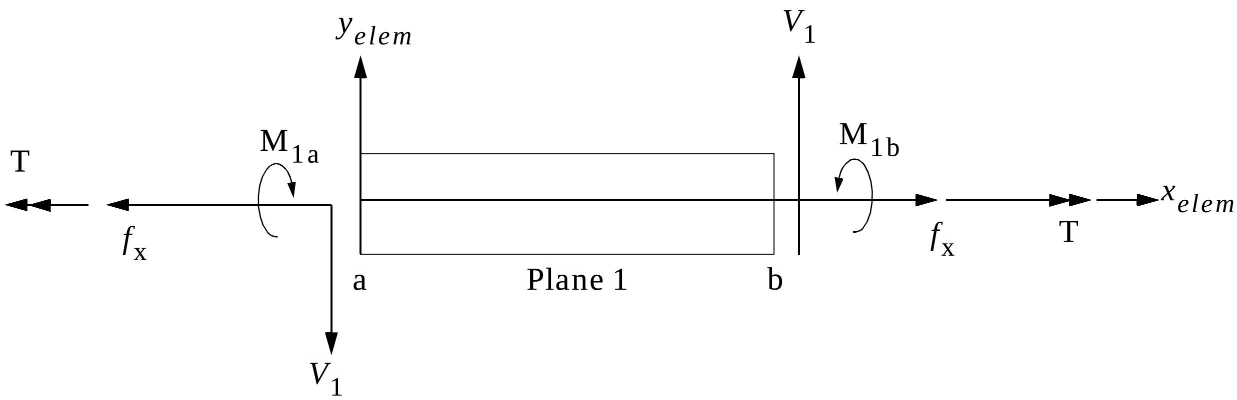


Figure 6-5 CBAR Element Internal Forces and Moments (x-y Plane)

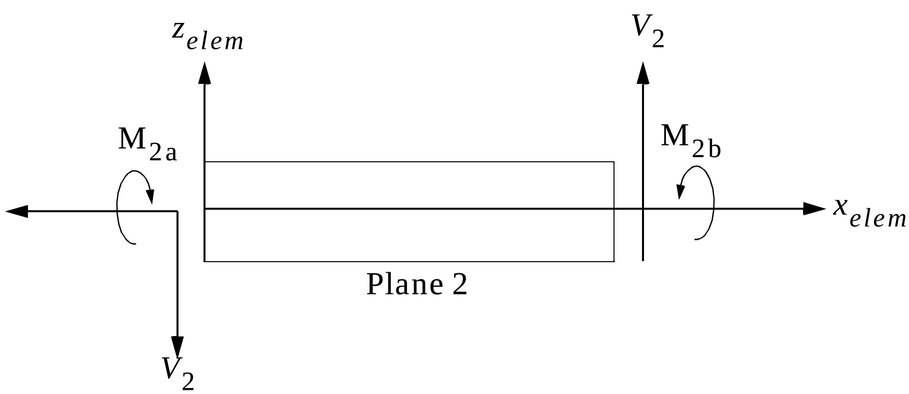


Figure 6-6 CBAR Element Internal Forces and Moments (x-z Plane)

Bar Element Property (PBAR)

The PBAR entry defines the properties of a CBAR element. The format of the PBAR entry is as follows:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of bar cross section. (Real)

Field	Contents
I1, I2, I12	Area moments of inertia. (Real; $I1 \geq 0.0$, $I2 \geq 0.0$, $I1 \cdot I2 > I12^2$)
J	Torsional constant. (Real)
NSM	Nonstructural mass per unit length. (Real)
K1, K2	Area factor for shear. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)

PID is the property's identification number from field 3 of the CBAR entry. MID references a MAT1 material property entry. I1 and I2 are area moments of inertia:

I1 = area moment of inertia for bending in plane 1 (same as Izz, bending about the z element axis)

I2 = area moment of inertia for bending in plane 2 (same as Iyy, bending about the y element axis)

J is the cross section's torsional constant (see **Table 6-2**). K1 and K2 depend on the shape of the cross section. K1 contributes to the shear resisting transverse force in plane 1 and K2 contributes to the shear resisting transverse force in plane 2.

Table 6-3 Area Factors for Shear

Shape of Cross Section	Value of K
Rectangular	$K1 = K2 = 5/6$
Solid Circular	$K1 = K2 = 9/10$
Thin-wall Hollow Circular	$K1 = K2 = 1/2$
Wide Flange Beams:	
Minor Axis	$\approx A_f / 1.2A$
Major Axis	$\approx A_w / A$

where:

A = Beam cross-sectional area

A_f = Area of flange

A_w = Area of web

The first continuation entry defines stress recovery coefficient points (Ci, Di, Ei, Fi) on the beam's cross section. These points are in the y-z plane of the element coordinate system as shown in **Figure 6-7**.

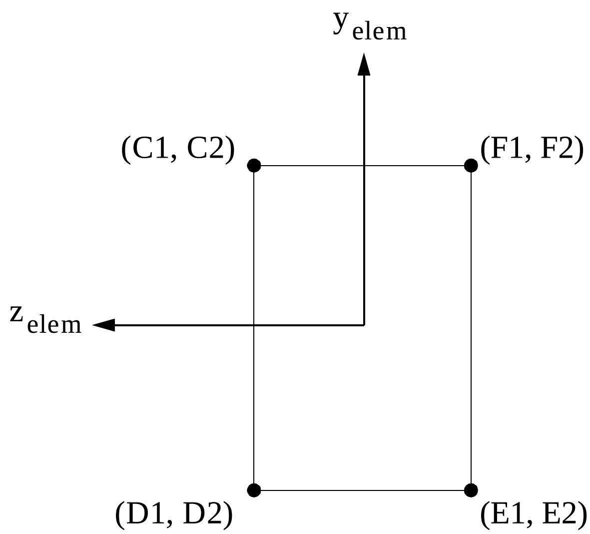


Figure 6-7 Stress Recovery Points on Beam Cross Section

By defining stress recovery points, you are providing c in the equation $\sigma = Mc/I$, thereby allowing MSC.Nastran to calculate stresses in the beam or on its surface.

6.4 Surface Elements

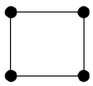
Surface elements, also called two-dimensional elements, are used to represent a structure whose thickness is small compared to its other dimensions. Surface elements can model plates, which are flat, or shells, which have single curvature (e.g. cylinder) or double curvature (e.g. sphere). For the grid points used to represent plate elements, stiffness terms exist for five of the possible six degrees of freedom per grid point. There is no stiffness associated to the rotation about the normal to the plate. This rotational DOF must be constrained in order to prevent stiffness singularities.

For linear analysis, MSC.Nastran plate elements assume classical assumptions of thin plate behavior:

- A thin plate is one in which the thickness is much less than the next larger dimension.
- The deflection of the plate's midsurface is small compared with its thickness.
- The midsurface remains unstrained (neutral) during bending-this applies to lateral loads, not in-plane loads.
- The normal to the midsurface remains normal during bending.

Quadrilateral Plate Element (CQUAD4)

CQUAD4



The CQUAD4 is MSC.Nastran's most commonly used element for modeling plates, shells, and membranes. The CQUAD4 can represent in-plane, bending, and transverse shear behavior, depending upon data provided on the PSHELL property entry. The CQUAD4 element is a quadrilateral flat plate connecting four grid points, as shown in **Figure 6-8**.

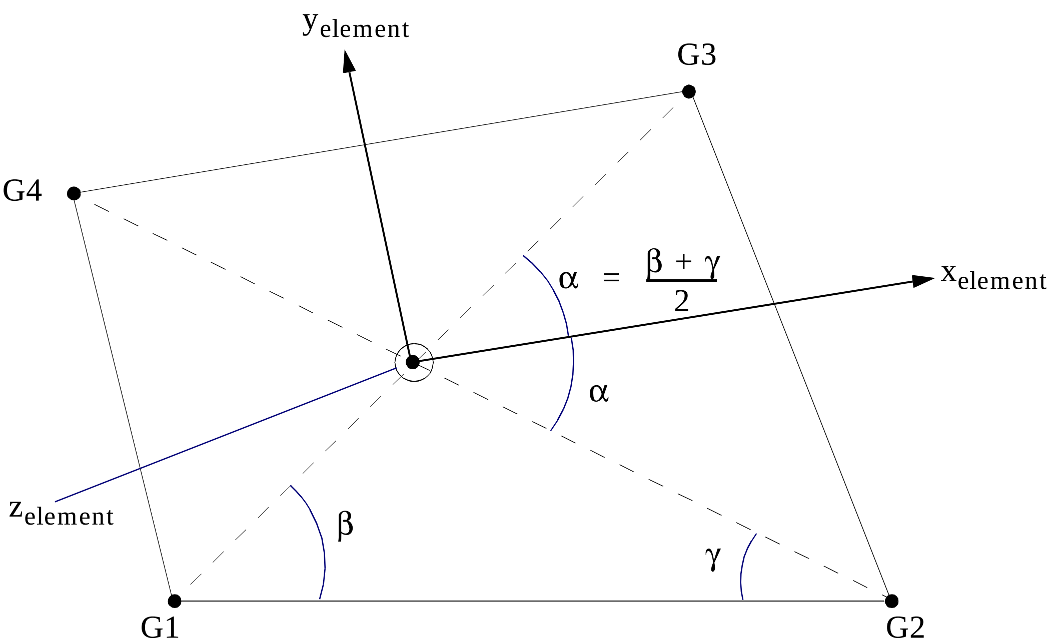


Figure 6-8 CQUAD4 Element Geometry and Element Coordinate System

CQUAD4 Format

The format of the CQUAD4 element entry is as follows:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
			T1	T2	T3	T4			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL or PCOMP entry. (Integer > 0; Default is EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real)
Ti	Membrane thickness of element at grid points G1 through G4. (Real ≥ 0.0 or blank, not all zero)

PID in field 3 points to a PSHELL element property entry. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element. THETA and MCID are not required for homogenous, isotropic materials. ZOFFS is used when offsetting the element from its connection point. For more information on THETA, MCID, and ZOFFS, see the “**CQUAD4**” on page 1141 of the . The continuation entry is optional. If not supplied, then corner thicknesses T1 through T4 will be set equal to the value of T (plate thickness) on the PSHELL entry. Finally, all interior angles of the CQUAD4 element must be less than 180°.

CQUAD4 Element Coordinate System

CQUAD4 element forces and stresses are output in the element coordinate system. The element coordinate system is established as follows:

- The element x-axis bisects the angle 2α. The positive direction is from G1 to G2.
- The element y-axis is perpendicular to the element x-axis and lies in the plane defined by G1, G2, G3, and G4. The positive direction is from G1 to G4.
- The element z-axis is normal to the x-y plane of the element. The positive z direction is defined by applying the right-hand rule to the ordering sequence of G1 through G4.

CQUAD4 Force and Moment Conventions

The following nomenclature is used when interpreting element output:

F_x, F_y	Membrane force per unit length
F_{xy}	Membrane shear force per unit length
M_x, M_y	Bending moments per unit length
M_{xy}	Twisting moment per unit length
V_x, V_y	Transverse shear forces per unit length

Element output is interpreted according to the conventions shown in **Figure 6-9**.

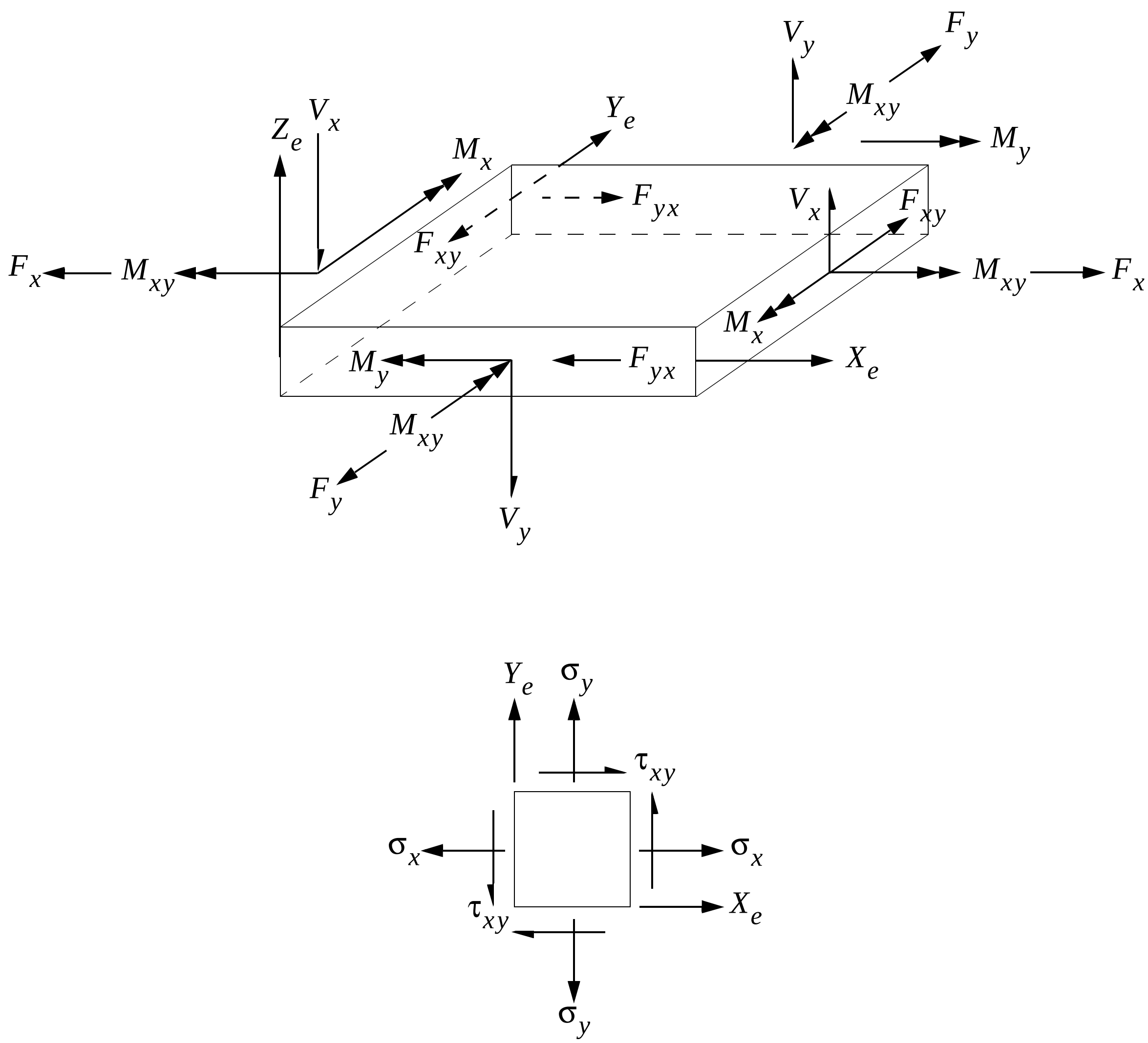
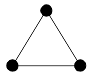


Figure 6-9 CQUAD4 Element Force, Moment, and Stress Conventions

Forces and moments are calculated at the element's centroid. Stresses are calculated at distances Z1 and Z2 from the element's reference plane (Z1 and Z2 are specified on the PSHELL property entry, and are normally specified as the surfaces of the plate; i.e., Z1, Z2 $= \pm \text{thickness}/2$).

Triangular Plate Element (CTRIA3)

CTRIA 3



The CTRIA3 element is a triangular plate connecting three grid points, as shown in **Figure 6-10**.

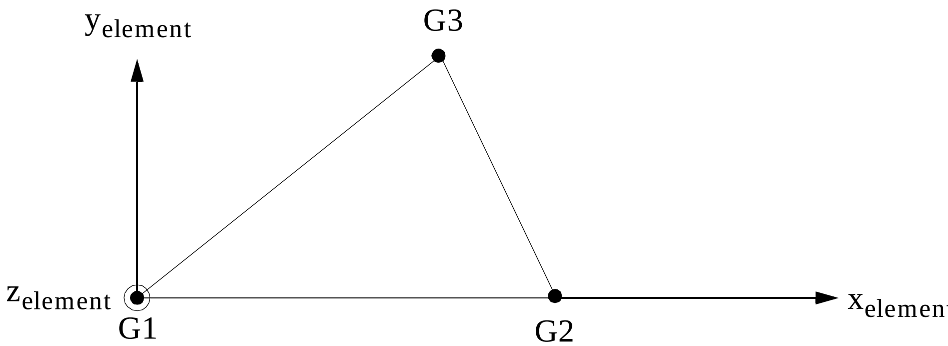


Figure 6-10 CTRIA3 Element Geometry and Element Coordinate System

The CTRIA3 is most commonly used for mesh transitions and filling in irregular boundaries. The element may exhibit excessive stiffness, particularly for membrane strain. Thus, as a matter of good modeling practice, CTRIA3s should be located away from areas of interest whenever possible. In other respects, the CTRIA3 is analogous to the CQUAD4.

CTRIA3 Format

The format of the CTRIA3 element entry is as follows:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
			T1	T2	T3				

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL or PCOMP entry. (Integer > 0; Default is EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real)
Ti	Membrane thickness of element at grid points G1, G2, and G3. (Real ≥ 0.0 or blank, not all zero)

PID in field 3 points to a PSHELL element property entry. We are not concerned at this time with material coordinate systems (THETA, MCID) or element offsets (ZOFFS); for more information, see the “**CTRIA3**” on page 1185 of the . The continuation entry is optional. If it is not supplied, then corner thicknesses T1 through T3 will be set equal to the value of T on the PSHELL entry.

CTRIA3 Element Coordinate System

CTRIA3 element forces and stresses are output in the element coordinate system. The element coordinate system (see **Figure 6-10**) is established as follows:

- The element x-axis lies in the direction from G1 to G2.
- The element y-axis is perpendicular to the element x-axis, and the positive x-y quadrant contains G3.
- The element z-axis is normal to the plane of the element. The positive z direction is established by applying the right-hand rule to the ordering sequence of G1 through G3.

Forces and moments are calculated at the element’s centroid. Stresses are calculated at distances Z1 and Z2 from the element reference plane (Z1 and Z2 are specified on the PSHELL entry).

Shell Element Property (PSHELL)

The PSHELL entry defines the membrane, bending, transverse shear, and coupling properties of thin plate and shell elements. The format of the PSHELL entry is as follows:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T	NSM	
	Z1	Z2	MID4						

Field	Contents
PID	Property identification number. (Integer > 0)
MID1	Material identification number for membrane. (Integer ≥ 0 or blank)
T	Default value for the membrane thickness. (Real)
MID2	Material identification number for bending. (Integer ≥ –1 or blank)
$12I/T^3$	Bending stiffness parameter. (Real > 0.0; Default = 1.0)
MID3	Material identification number for transverse shear. (Integer > 0 or blank; must be blank unless MID2 > 0)
TS/T	Transverse shear thickness divided by the membrane thickness. (Real > 0.0; Default = .833333)
NSM	Nonstructural mass per unit area. (Real)

Field	Contents
Z1, Z2	Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. (Real or blank; Default $= \pm t/2$)
MID4	Material identification number for membrane-bending coupling. (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2)

PID in field 2 is referenced by a surface element (e.g., CQUAD4 or CTRIA3). MID1, MID2, and MID3 are material identification numbers that normally point to the same MAT1 material property entry. T is the uniform thickness of the element. For solid homogenous plates, the default values of $12I/T^3$ (field 6) and TS/T (field 8) are correct.

The CQUAD4 element can model in-plane, bending, and transverse shear behavior. The element's behavior is controlled by the presence or absence of a material ID number in the appropriate field(s) on the PSHELL entry.

TO MODEL A MEMBRANE (i.e., no bending), fill in MID1 only. For example,

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1	204	.025						

TO MODEL BENDING ONLY, fill in MID2 only. For example,

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204					

TO ADD TRANSVERSE SHEAR FLEXIBILITY TO BENDING, fill in MID2 and MID3. For example,

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204		204			

Note: The phrase "add transverse shear flexibility" is standard but somewhat confusing. This is what it means: using MID3 adds a shear term in the element's stiffness formulation. Therefore, a plate element with an MID3 entry will deflect more (if transverse shear is present) than an element without an MID3 entry. For very thin plates, this shear term adds very little to the deflection result. For thicker plates, the contribution of transverse shear to deflection becomes more pronounced, just as it does with short, deep beams.

For a solid, homogeneous, thin, stiff plate, use MID1, MID2, and MID3 (all three MIDs reference the same material ID). For example:

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204		204			

Example

A 10 in x 10 in by 0.15 inch cantilever plate is subjected to in-plane tensile loads of 300 lb_f and lateral loads of 0.5 lb_f at each free corner. Find the displacements, forces, and stresses in the plate. A single CQUAD4 element is used to model the plate as shown in **Figure 6-11**.

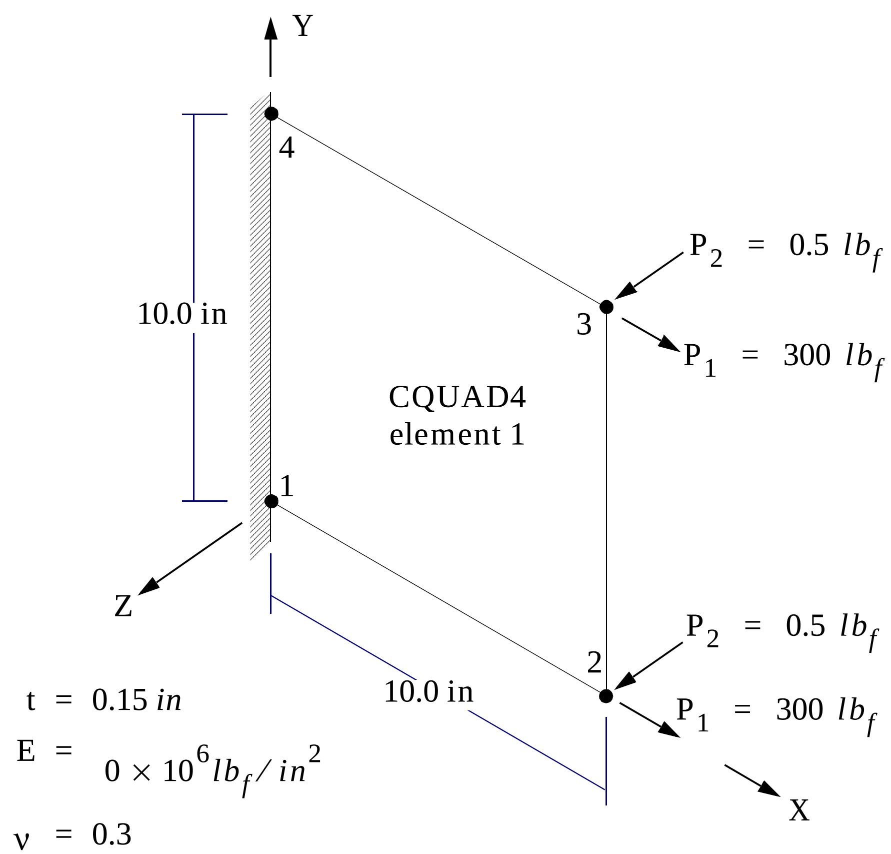


Figure 6-11 Cantilever Plate Example

The required element-related Bulk Data entries are specified as follows:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.		123456		
GRID	2		10.	0.	0.				

1	2	3	4	5	6	7	8	9	10
GRID	3		10.	10.	0.				
GRID	4		0.	10.	0.		123456		

CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
CQUAD4	1	5	1	2	3	4			

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T	NSM	
PSHELL	5	7	0.15	7		7			

MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	7	30.E6		0.3					

The Case Control commands required to obtain the necessary output are as follows:

```
FORCE=ALL
DISP=ALL
STRESS=ALL
```

The grid point displacement output is shown in **Figure 6-12**.

```
1    CANTILEVER QUAD PLATE                                                  SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    11

                                                D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0            0.0            0.0            0.0            0.0            0.0
             2      G      1.301162E-04   2.927614E-05   3.737665E-03   9.460688E-05  -5.676413E-04   0.0
             3      G      1.301162E-04  -2.927614E-05   3.737665E-03  -9.460688E-05  -5.676413E-04   0.0
             4      G      0.0            0.0            0.0            0.0            0.0            0.0
```

Figure 6-12 Grid Point Displacement

Here are some checks and observations you can make in examining the deflections shown in **Figure 6-12**:

1. The maximum deflections of 3.738E-3 inches are due to the 0.5 lb_f lateral loads and occur at grid points 2 and 3 (the free edge), as expected.
2. The free edge deflections are identical since the structure and loadings are symmetric.
3. Grid points 1 and 4 have exactly zero displacement in all DOFs, since they were constrained to be fixed in the wall.
4. The lateral deflections occur in the T3 (+z) direction, which correspond with the direction of lateral loading. Note that these displacements are reported in the displacement coordinate system, not the element coordinate system.
5. The maximum lateral deflection of 3.738E-3 inches is much less than the thickness of the plate (0.15 in). Therefore, we are comfortably within the range of small displacement linear plate theory.

The CQUAD4 element force output is shown in **Figure 6-13**.

```
CANTILEVER QUAD PLATE                                                       SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01    PAGE    12

                           F O R C E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )

ELEMENT                    - MEMBRANE  FORCES -                          - BENDING  MOMENTS -                - TRANSVERSE SHEAR FORCES -
  ID              FX            FY            FXY             MX            MY            MXY              QX            QY
   1       6.000000E+01  4.825737E+00  2.220446E-16    5.000000E-01  7.017544E-02 -1.214306E-17    1.000000E-01  3.469447E-18
```

In-plane element forces in element coordinate system (force/length)

Element internal moments in element coordinate system (moment/length)

Transverse shear forces (force/length)

Moment Diagram:

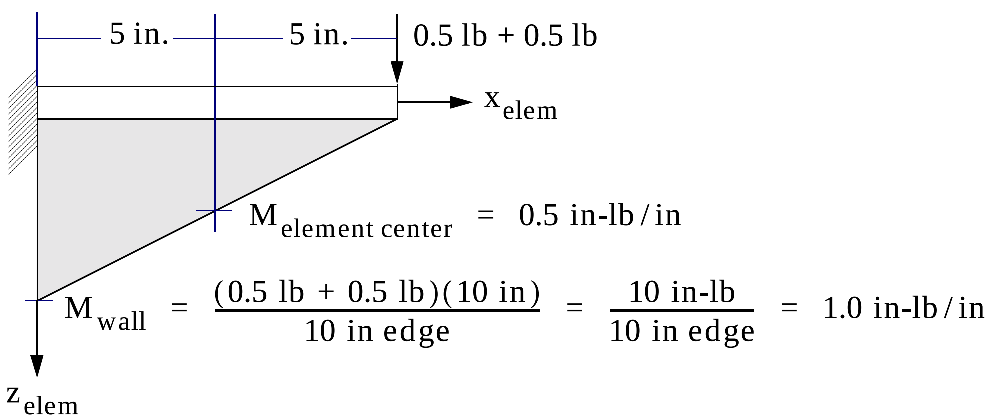


Figure 6-13 CQUAD4 Element Force and Moment Output

Note that numbers such as -1.214306E-17 (for bending moment MXY) are called “machine zeros”-they are zeros with slight errors due to computer numerical roundoff.

The CQUAD4 element stress output is shown in **Figure 6-14**.

```
1   CANTILEVER QUAD PLATE                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    14

                         S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )
  ELEMENT      FIBRE               STRESSES IN ELEMENT COORD SYSTEM             PRINCIPAL STRESSES (ZERO SHEAR)
    ID.      DISTANCE           NORMAL-X       NORMAL-Y      SHEAR-XY       ANGLE         MAJOR           MINOR         VON MISES

      1   -7.500000E-02     5.333333E+02   5.088503E+01  -1.757853E-15     0.0000    5.333333E+02    5.088503E+01    5.097990E+02
           7.500000E-02     2.666667E+02   1.345813E+01   4.718447E-15     0.0000    2.666667E+02    1.345813E+01    2.601988E+02
```

Axial and bending stresses at the centroid of the CQUAD4 element are given by:

$$\sigma_{\mathrm{AXIAL}} = \frac{P}{A} = \frac{300\ \mathrm{lb} + 300\ \mathrm{lb}}{(10\ \mathrm{in})(0.15\ \mathrm{in})} = 400\ \mathrm{lb/in}^2$$

$$\sigma_{\mathrm{BENDING}} = \frac{Mc}{I} = \frac{(.5\ \mathrm{in\text{-}lb/in})(10\ \mathrm{in\ edge})(.075\ \mathrm{in})}{(10\ \mathrm{in})(0.15\ \mathrm{in})^3 / 12} = \pm 133.33\ \mathrm{lb/in}^2$$

On the tensile surface,

$$\sigma_x^+ = \sigma_{\mathrm{AXIAL}} + \sigma_{\mathrm{BENDING}} = 400 + 133.33 = \underline{\underline{533.33\ \mathrm{lb/in}^2}}$$

On the compressive surface,

$$\sigma_x^- = \sigma_{\mathrm{AXIAL}} - \sigma_{\mathrm{BENDING}} = 400 - 133.33 = \underline{\underline{266.67\ \mathrm{lb/in}^2}}$$

The von Mises stress on the compressive surface is given by

$$\sigma_{\mathrm{von}}^- = \sqrt{\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2} = \sqrt{(266.67)^2 - (266.67)(13.46) + (13.46)^2 + \underbrace{3(4.718E\text{-}15)^2}_{0}}$$

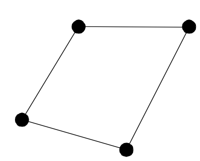
$$\sigma_{\mathrm{von}}^- = 260.2\ lb_f / in^2$$

Figure 6-14 CQUAD4 Element Stress Output

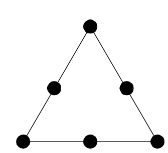
Other Surface Elements

Brief descriptions of MSC.Nastran's other surface elements are provided here for reference:

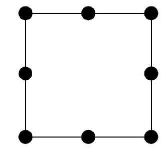
CSHEAR — Four-grid element that supports shear and extensional force only. Used for analyzing thin reinforced plates and shells. Commonly used with rod elements to analyze thin-skinned aircraft structures. Performs best if kept rectangular.



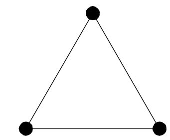
CTRIA6 — Isoparametric triangular element with three corner and three midside grid points. Used for transitioning meshes in regions with curvature.



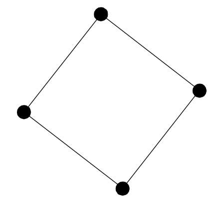
CQUAD8 Isoparametric element with four corner and four midside grid points. Useful for modeling singly-curved shells (e.g., a cylinder). The CQUAD4 element performs better for doubly-curved shells (e.g., a sphere).



CTRIAR Three-grid isoparametric flat element. Companion to the CQUADR element.



CQUADR Four-grid isoparametric flat plate element without membrane-bending coupling. Better performance for modeling planar structures with in-plane loads (i.e., membrane behavior). Less sensitive to distortion and extreme values of Poisson's ratio than the CQUAD4. Not recommended for curved surfaces.

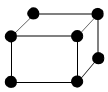


6.5 Solid Elements

MSC.Nastran solid (three-dimensional) elements are used to represent the behavior of thick plates and solids. The principal solid elements are the six-sided CHEXA, the five-sided CPENTA, and the four-sided CTETRA. Solid elements have only translational degrees of freedom-no rotational DOFs are used to define the solid elements.

Six-Sided Solid Element (CHEXA)

CHEXA



The CHEXA element is recommended for general use. The CHEXA's accuracy degrades when the element is skewed. When it is used to simulate bending behavior it is important to take special precautions. In some modeling situations, it has superior performance to other 3-D elements.

The CHEXA has eight corner grid points and up to twenty grid points if midside grid points are included. Element stresses (σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, and τ_{zx}) are calculated at the center, and are also extrapolated out to the corner grid points. The element's connection geometry is shown in **Figure 6-15**.

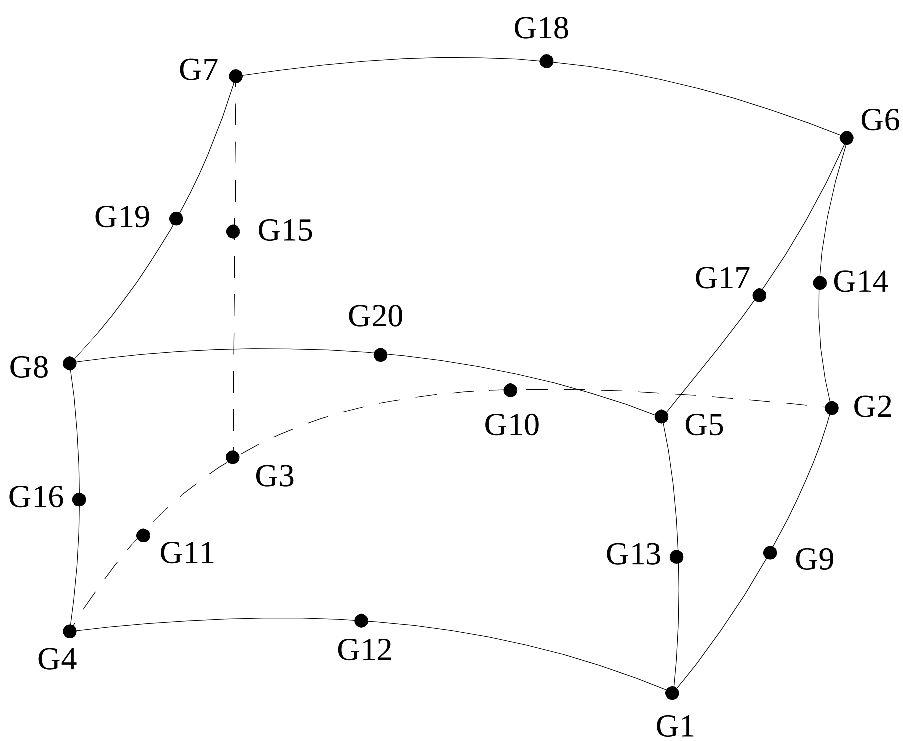


Figure 6-15 CHEXA Element Connection

CHEXA Format

The format of the CHEXA element entry is as follows:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 or blank)

Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc. The midside nodes, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any midside node is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. Components of stress are output in the material coordinate system. The material coordinate system is defined on the PSOLID property entry. The second continuation entry is optional.

The CHEXA element coordinate system is shown in **Figure 6-16**.

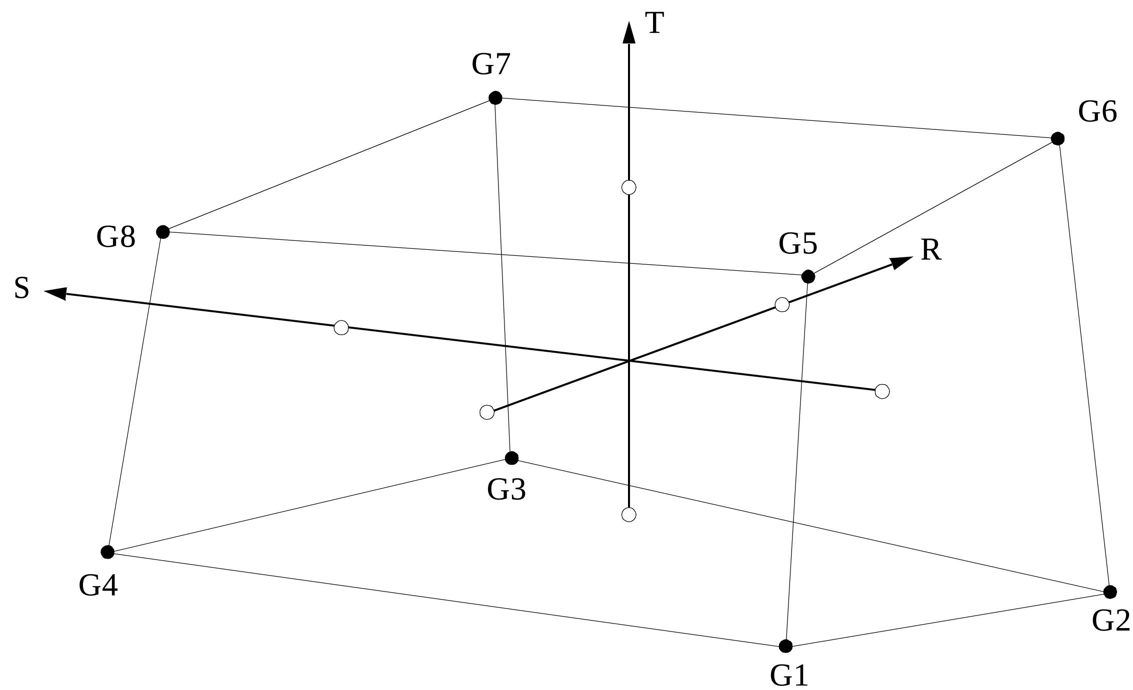


Figure 6-16 CHEXA Element Coordinate System

The CHEXA element coordinate system is defined in terms of vectors R, S, and T which join the centroids of opposite faces.

The R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.

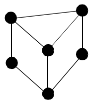
The S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.

The T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction.

Five-Sided Solid Element (CPENTA)

CPENTA



The CPENTA element is commonly used to model transitions from solids to plates or shells. If the triangular faces are not on the exposed surfaces of the shell, excessive stiffness can result.

The CPENTA element uses from six to fifteen grid points (six with no midside grid points; up to fifteen using midside grid points). Element stresses (σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, and τ_{zx}), are calculated at the center and are also extrapolated out to the corner grid points. The element's connection geometry is shown in **Figure 6-17**.

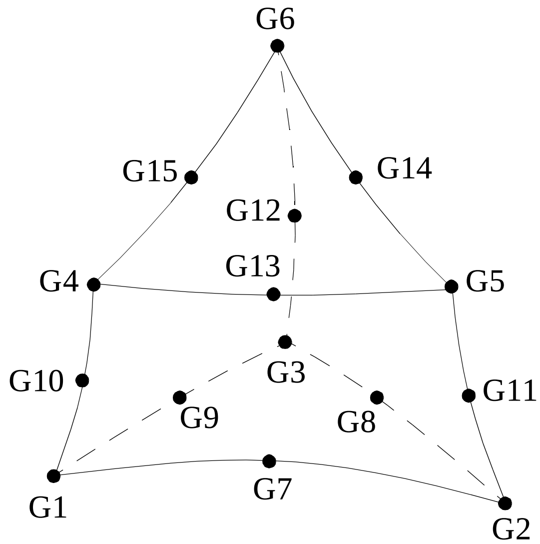


Figure 6-17 CPENTA Element Connection

CPENTA Format

The format of the CPENTA element entry is as follows:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank)

Grid points G1, G2, and G3 define a triangular face. Grid points G1, G10, and G4 are on the same edge, etc. The midside nodes, G7 to G15, are optional. Any or all midside nodes may be deleted. The continuations are not required if all midside nodes are deleted. Components of stress are output in the material coordinate system. The material coordinate system is defined on the PSOLID property entry.

The CPENTA element coordinate system is shown in **Figure 6-18**.

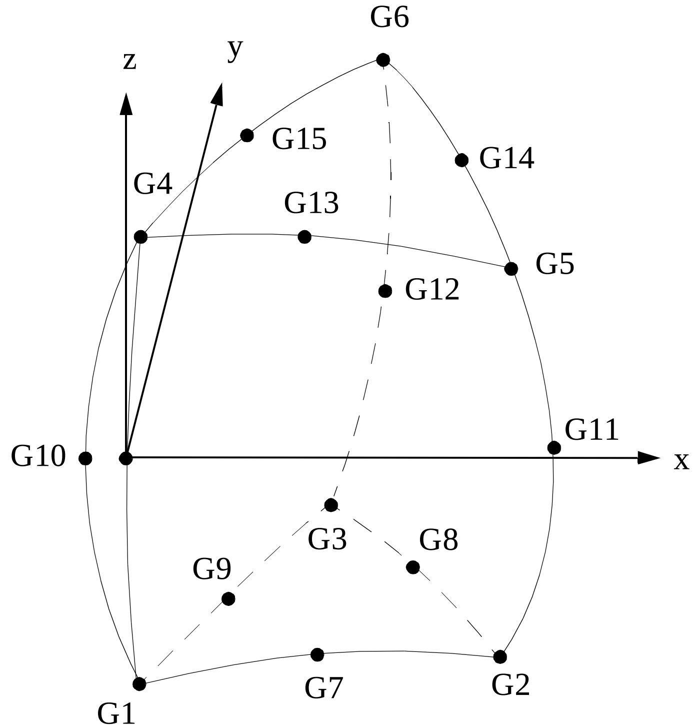
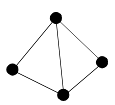


Figure 6-18 CPENTA Element Coordinate System

The origin of the CPENTA element coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 to G5 and G3 to G6, respectively.

Four-Sided Solid Element (CTETRA)

CTETRA



The CTETRA solid element is used widely to model complicated systems (i.e. extrusions with many sharp turns and fillets, turbine blades). The element has a distinct advantage over the CHEXA when the geometry has sharp corners. For this situation it is possible to have CTETRAs that are much better shaped than CHEXAs. However, it is important to use CTETRAs with ten grid points for all structural simulations (e.g. solving for displacement and stress). The CTETRA with four grid points is overly stiff for these applications. It is acceptable to use CTETRAs with four grid points for heat transfer applications.

The CTETRA has four grid points without midside nodes, or up to ten grid points with midside nodes. Element stresses (σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, and τ_{zx}), are calculated at the center, and are also extrapolated out to the corner grid points. The element's connection geometry is shown in **Figure 6-19**.

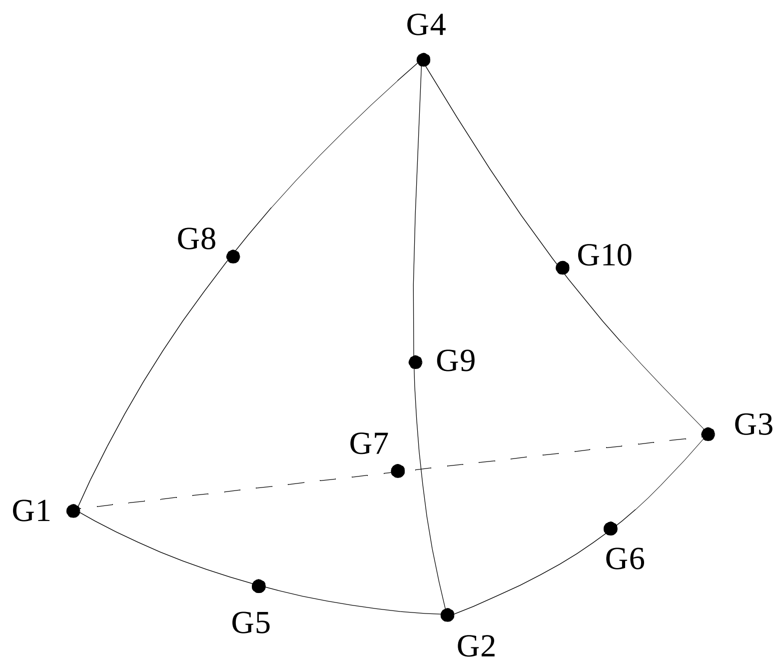


Figure 6-19 CTETRA Element Connection

CTETRA Format

The format of the CTETRA element is as follows:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSOLID property entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank)

Grid points G1, G2, and G3 define a triangular face. The midside nodes, G5 to G10, must be located as shown on **Figure 6-19**. If the ID of any midside node is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. Components of stress are output in the material coordinate system. The material coordinate system is defined on the PSOLID property entry.

The CTETRA element coordinate system is shown in Figure 6-20.

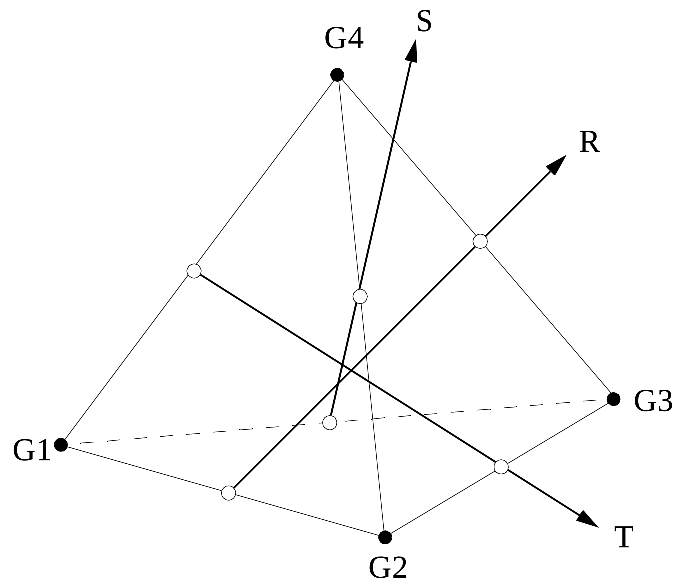


Figure 6-20 CTETRA Element Coordinate System

The CTETRA element coordinate system is derived from the three vectors R, S, and T which join the midpoints of opposite edges.

The R vector joins the midpoints of edges G1-G2 and G3-G4.

The S vector joins the midpoints of edges G1-G3 and G2-G4.

The T vector joins the midpoints of edges G1-G4 and G2-G3.

The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction.

Solid Element Property (PSOLID)

The PSOLID entry defines the properties of CHEXA, CPENTA, and CTETRA solid elements. The format of the PSOLID entry is as follows:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, or MAT10 entry. (Integer > 0)
CORDM	Identification number of the material coordinate system. (Integer; Default = -1)
IN	Integration network. (Integer, Character, or blank)
STRESS	Location selection for stress output. (Integer, Character, or blank)

Field	Contents
ISOP	Integration scheme. (Integer, Character, or blank)
FCTN	Fluid element flag. (Character: “PFLUID” indicates a fluid element, “SMECH” indicates a structural element; Default = “SMECH”)

The property identification number (PID) in field 2 points to one or more solid element entries. MID in field 3 references a MATi material property entry. CORDM in field 4 defines the material coordinate system. The material coordinate system may be the basic system (0), any defined system (Integer > 0), or the element coordinate system (-1 or blank). IN in field 5 concerns options for the integration network used by the element; new users are advised to use the default value (i.e., leave field 5 blank). ISOP in field 7 concerns the element’s integration scheme; new users should use the default value. FCTN in field 7 is only used for fluid elements.

6.6 Rigid Bar Element (RBE2)

As a general modeling principle, adjacent elements that differ greatly in relative stiffness-several orders of magnitude or more-can cause numerical difficulties in the solution of the problem. If you were to use, say, a CBAR element with extremely large values of I1 and I2 to simulate a rigid connection, a so-called numerically ill-conditioned problem would likely occur. The RBE2 element defines a rigid body whose independent degrees of freedom are specified at a single point and whose dependent degrees of freedom are specified at an arbitrary number of points. The RBE2 element does not cause numerical difficulties because it does not add any stiffness to the model. The RBE2 element is actually a constraint element that prescribes the displacement relationship between two or more grid points.

The format of the RBE2 element entry is shown below:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-					

Field	Contents
EID	Element identification number. (Integer > 0)
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees of freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks)
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned. (Integer > 0)

Example

A stiffened plate is modeled with two CQUAD4 elements and a CBAR element representing the stiffener, as shown in **Figure 6-21**. Two RBE2 elements are used to connect the CBAR stiffener to the plate elements.

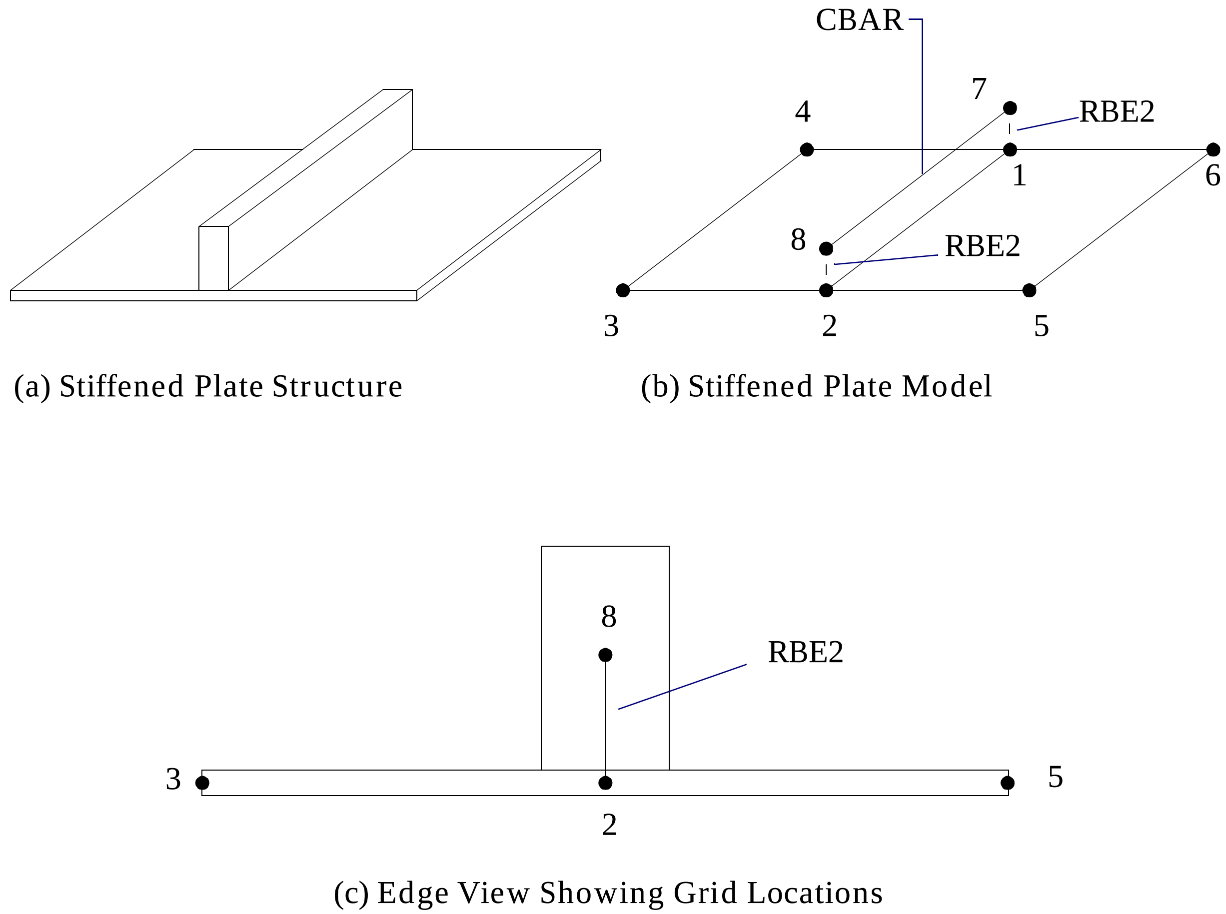


Figure 6-21 RBE2 Example

Grid points 7 and 8 are at the ends of the CBAR element and lie along the stiffener's neutral axis. An RBE2 element connects all six dependent degrees-of-freedom at grid point 7 (on the beam) to all six independent degrees-of-freedom at grid point 1 (on the plate). There is a similar element at the other end of the beam. Remember that an RBE2 element is not a finite element, but a set of equations that define a kinematic relationship between different displacements.

The required RBE2 entries are written as follows:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4		
RBE2	12	1	123456	7					
RBE2	13	2	123456	8					

CHAPTER 7

Applying Constraints

7.1 Introduction

A constraint is the enforcement of a prescribed displacement (i.e., component of translation or rotation) on a grid point or points. There are two basic types of constraints in MSC.Nastran: single point constraints (SPCs) and multipoint constraints (MPCs). A single point constraint is a constraint applied to an individual grid point. Single point constraints can enforce either zero displacement or nonzero displacement. A multipoint constraint is a mathematical constraint relationship between one grid point and another grid point (or set of grid points). User-defined multipoint constraints are beyond the scope of this book-we will concentrate on single point constraints.

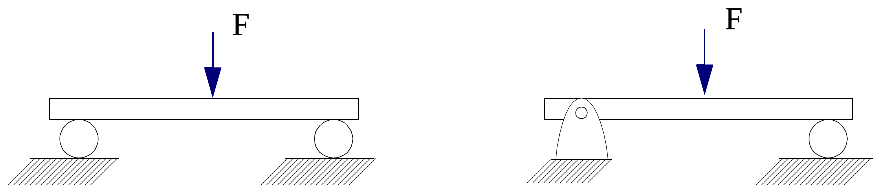
The boundary conditions of a static structure (fixed, hinged, roller support, etc.) typically require that various degrees of freedom be constrained to zero displacement. For example, consider a grid point fixed in a rigid wall. All six displacement degrees of freedom-three translational directions and three rotational directions-must be constrained to zero in order to mathematically describe the fixed boundary condition. Structural reaction forces called single point constraint forces (SPCFs) may be obtained at grid points constrained by SPCs and are listed in the .f06 output file.

Section 7.2 describes how boundary conditions are used to prevent rigid body motion-the presence of rigid body motion in an MSC.Nastran static analysis will cause the run to fail. **Section 7.3** describes several ways to apply SPCs to grid points. **Section 7.4** shows how MSC.Nastran automatically detects and constrains singularities using a feature called AUTOSPC. Finally, **Section 7.5** gives several examples of modeling simple boundary conditions.

7.2 Rigid Body Motion and Mechanisms

The solution of the equation $\{F\} = [K]\{u\}$ requires that $[K]$, the stiffness matrix, be nonsingular (the determinant of a nonsingular matrix cannot be equal to zero). If any unconnected degrees of freedom exist in the model, the stiffness matrix will be singular and the solution of $\{F\} = [K]\{u\}$ cannot proceed. Rigid body motion occurs when the structure can move freely in one or more displacement directions (displacement without strain). Rigid body motion causes singularities and must therefore be constrained in linear static analysis.

Consider the two beams shown in **Figure 7-1**:



(a) Rigid Body Motion (b) Adequate Constraints

Figure 7-1 Rigid Body Motion vs. Adequate Constraints

Static Equilibrium. The beam shown in (a) is in “real world” static equilibrium-no displacement in the beam’s axial direction occurs because no actual forces act on the beam to cause axial displacement. The constraints shown in (a) are not adequate for MSC.Nastran, which sees this structure as unstable since an infinite number of displacement solutions are possible (MSC.Nastran evaluates structural stability independent of the applied load). The beam shown in (b) is adequately constrained for MSC.Nastran.

Mechanisms. A “subclass” of rigid body motion occurs when part of an otherwise constrained structure is capable of rigid body motion. This is called a mechanism. In linear static analysis, the presence of a mechanism also produces a singularity failure in the solution.

Inertia Relief. A special technique called inertia relief is available to perform quasi-static analysis on unconstrained (free) structures under uniform (i.e., zero or constant) acceleration. Examples of such structures include an aircraft in flight or a satellite in orbit. A discussion of inertia relief is beyond the scope of this book; for further information, see ***MSC.Nastran Common Questions and Answers*** or the ***MSC.Nastran Linear Static Analysis User’s Guide***.

7.3 Single Point Constraints

Single point constraints (SPCs) are constraints applied to the displacements of individual grid points. We will learn how to apply SPCs using the GRID, SPC, and SPC1 Bulk Data entries.

Permanent Constraint on the GRID Entry

The GRID Bulk Data entry format is shown below:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

An entry in field 8 (PS) defines permanent single point constraints associated with this grid point. You can enter any combination of the integers 1 through 6 with no embedded blanks. Applying constraints directly on the GRID entry is an easy method if only a few grid points in your model need to be constrained. The constraints are in the displacement coordinate system of the grid point (i.e., the coordinate system defined in field 7). It is not necessary to define a Case Control constraint set to use this method of applying constraints.

Single Point Constraint (SPC)

The SPC Bulk Data entry is used to apply a set of single point constraints or enforced displacements (i.e., nonzero values of displacement) for static analysis. The format of the SPC entry is as follows:

SPC	SID	G1	C1	D1	G2	C2	D2		

Field	Contents
SID	Identification number of the single point constraint set. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ci	Component number. ($0 \le$ Integer ≤ 6; Up to six unique Integers 1 through 6 may be placed in the field with no embedded blanks)
Di	Value of enforced displacement for degrees of freedom designated by Gi and Ci. (Real)

One SPC entry can handle two sets of (G,C,D) values. The default value for enforced displacement (D) is zero (i.e., for zero displacement, leave the field blank). Any number of SPC entries may be used to define a set of constraints.

The constraints are specified in the grid point's displacement coordinate system.

SID is the SPC set identification number and is selected in the Case Control Section with the command

```
SPC = SID
```

Single Point Constraint (SPC1)

The SPC1 Bulk Data entry is used to define single point constraints of zero displacement. The formats of SPC1 allow you to efficiently apply the same constraint to a large number of grid points.

The formats of the SPC1 entry are as follows:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

Alternate Format:

SPC1	SID	C	G1	"THRU"	G2				

Field	Contents
SID	Identification number of single point constraint set. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for "THRU" option, G1 < G2)

Any number of SPC1 entries may be used to define a set of constraints. The constraints are given in the grid point's displacement coordinate system.

SID is the SPC set identification number and is selected in the Case Control Section with the command

```
SPC = SID
```

7.4 Automatic Identification and Removal of Singularities (AUTOSPC)

MSC.Nastran automatically identifies and constrains singularities in the stiffness matrix using a parameter called AUTOSPC. A Grid Point Singularity Table is printed in the output (.f06) file showing which DOFs were constrained. If this table is present in the output, it is critical that you inspect the results carefully and understand which DOFs were singular and why-it is possible that singular DOFs are due to modeling errors.

A sample Grid Point Singularity Table is shown in **Figure 7-2**. This table is from the single element cantilever rod model in **“A Simple MSC.Nastran Model”** on page 19 (see **Figure 2-7**).

```
                                                                       SEPTEMBER   10,  2001  MSC.NASTRAN  9/10/01   PAGE     7

                                         G R I D   P O I N T   S I N G U L A R I T Y   T A B L E

                           POINT     TYPE   FAILED      STIFFNESS       OLD USET           NEW USET
                            ID              DIRECTION     RATIO     EXCLUSIVE  UNION   EXCLUSIVE  UNION
                             2        G       1         0.00E+00        B        F         SB       SB     *
                             2        G       3         0.00E+00        B        F         SB       SB     *
                             2        G       4         0.00E+00        B        F         SB       SB     *
                             2        G       5         0.00E+00        B        F         SB       SB     *
                             2        G       6         0.00E+00        B        F         SB       SB     *
```

Figure 7-2 Grid Point Singularity Table

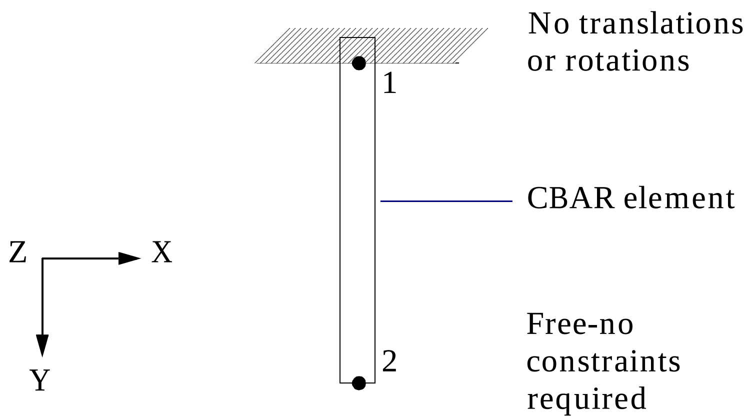
In this table, grid point 2, the free end of the rod, failed (i.e. had no stiffness connection) in five of its six possible DOFs. DOF 2 (translation in the Y direction) of grid point 2 was not singular and must therefore have been attached to something-the extensional stiffness of the rod in this case. CROD elements support extension and torsion, but in this example no value of J, the torsional constant, was provided in field 5 of the PROD element property entry. As a result, the CROD element had no torsional stiffness, and the resulting unconnected DOF was detected and constrained by AUTOSPC.

For further information on singularities and the grid point singularity table, see the ***MSC.Nastran Linear Static Analysis User’s Guide***.

7.5 Boundary Condition Examples

The following examples show how the GRID, SPC1, and SPC Bulk Data entries can be used to model several common structural boundary conditions.

Example 1 - Cantilever Beam



Grid 1 must be constrained in all six DOFs. This constraint can be conveniently specified directly in field 8 (permanent single point constraint) of the GRID Bulk Data entry:

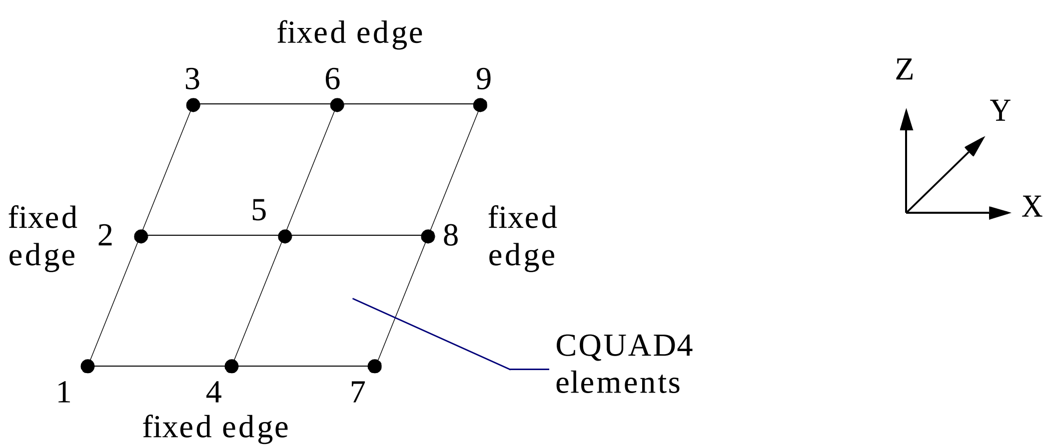
1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.		123456		

Or, in free field format,

```
GRID,1,,0.,0.,0.,,123456
```

Note that we do not need to specify a constraint set in the Case Control Section.

Example 2 - Fixed Plate



Grids 1, 2, 3, 4, 6, 7, 8, and 9 must be constrained in all six DOFs. Assume that a single point constraint selection is defined in the Case Control Section (SPC = 42). Applying the same constraint on numerous individual GRID Bulk Data entries is not particularly convenient-an SPC1 entry is a better choice:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

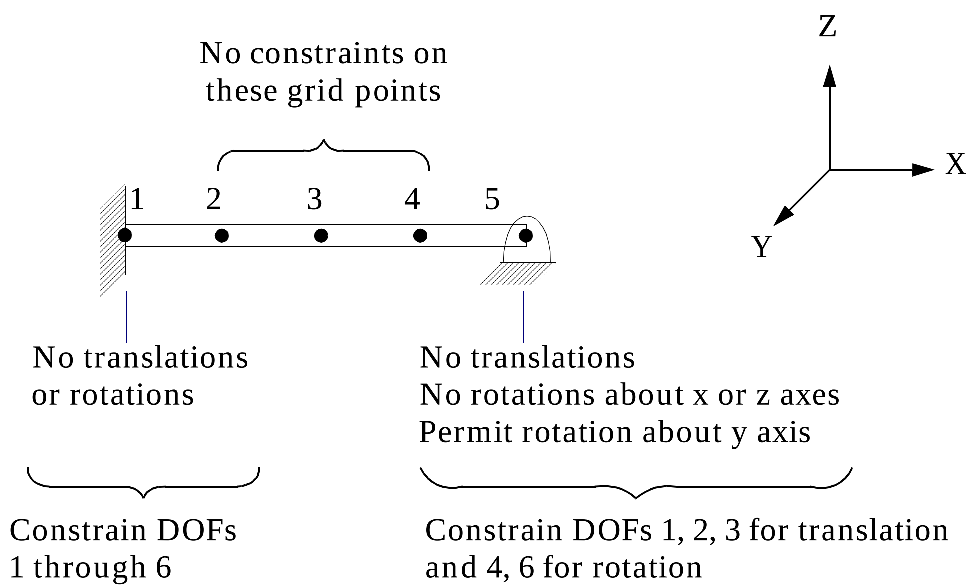
SPC1	42	123456	1	2	3	4	6	7	
	8	9							

Or, in free field format,

```
SPC1,42,123456,1,2,3,4,6,7
    ,8,9
```

Note that if the “THRU” (alternate format) option were used (1 THRU 9), grid point 5 would have been incorrectly fixed, and no displacement would occur anywhere in the model.

Example 3 - Fixed-Hinged Beam



An SPC entry will be used to constrain both ends of the beam. Assume that a single point constraint set selection is defined in the Case Control Section (SPC = 42). The values of enforced displacement (Di) are left blank since zero displacement is the default.

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

SPC	42	1	123456		5	12346			

Or, in free field format,

```
SPC,42,1,123456,,5,12346
```

One final note: real-world structures often do not have ideal or simple boundary conditions. The choice of constraints greatly influences the structure's response to loading, so care must be taken to model boundary conditions as accurately as possible.

CHAPTER

8

Material Properties

- Basic Material Property Definitions
- Material Definition (MAT1)
- Other Material Types Available in MSC.Nastran

8.1 Basic Material Property Definitions

In this book our discussion is limited to material that is linear, elastic, homogeneous, and isotropic. In addition, we assume that the material is a continuum (contains no gaps or voids) and that all material properties remain constant (many material properties are functions of temperature; thus, we infer that the temperature of the structure is also constant).

Definitions of these restrictions and of related material properties are given as follows:

- LINEAR

  Deformations are directly proportional to the applied load (i.e., strain is directly proportional to stress).

- ELASTIC

  An elastic structure returns to its original, undeformed shape when the load is removed.

- HOMOGENEOUS

  The material is the same throughout-material properties are independent of location within the material.

- ISOTROPIC

  Material properties do not change with the direction of the material.

- MODULUS OF ELASTICITY (YOUNG'S MODULUS) E

  E is the constant of proportionality relating stress to strain in the linear region. The greater the value of E, the stiffer the material.

- SHEAR MODULUS (MODULUS OF RIGIDITY) G

  G is the constant of proportionality relating shear stress to shear strain in the linear region.

- POISSON'S RATIO ν

  Poisson's ratio is the absolute value of the ratio of lateral linear strain to axial linear strain.

A typical stress-strain curve for structural steel is shown in **Figure 8-1**:

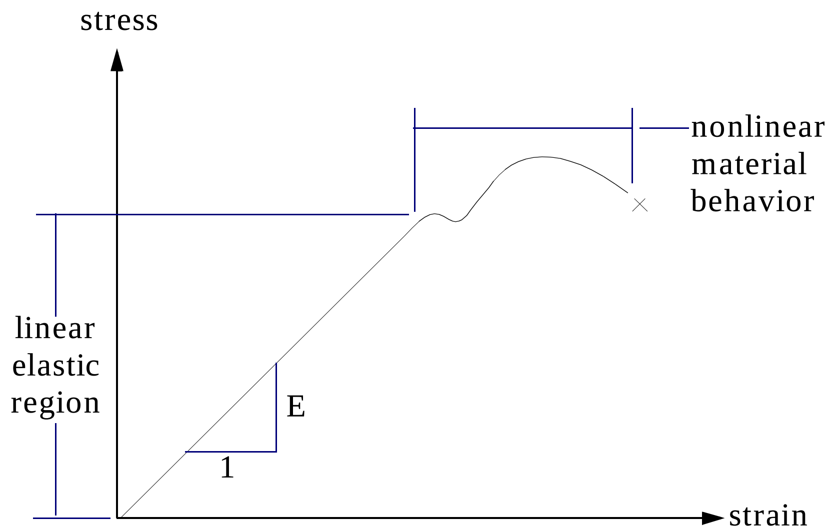


Figure 8-1 Typical Stress-Strain Curve for Structural Steel

If the loading on a structure is sufficient to exceed the linear elastic limit of the material, then nonlinear methods are required to predict the nature of the plastically (permanently) deformed state. See "**Nonlinear Analysis**" on page 15 for further information on nonlinear analysis. Nonlinear methods are also required if the material is nonlinear in its elastic range.

8.2 Material Definition (MAT1)

Linear, elastic, homogeneous, isotropic materials are modeled in MSC.Nastran with the MAT1 Bulk Data entry. The MAT1 entry has the following format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Field	Contents
MID	Material identification number. (Integer > 0)
E	Young's modulus. (Real ≥ 0.0 or blank)
G	Shear modulus. (Real ≥ 0.0 or blank)
NU	Poisson's ratio. ($-1.0 <$ Real ≤ 0.5 or blank)
RHO	Mass density. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. (Real; Default = 0.0 if A is specified)
GE	Structural element damping coefficient. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear used only to compute margins of safety in certain elements; they have no effect on the computational procedures. (Real)
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See the "**Parameters**" on page 601 of the *MSC.Nastran Quick Reference Guide*. (Integer ≥ 0 or blank)

The material identification number (MID) connects the MAT1 entry to an element property entry (e.g., PBAR, PSHELL, or PSOLID).

You need only supply two of the three properties E, G, and NU-the remaining value, if required, is calculated automatically according to the following relation:

$$G = \frac{E}{2(1+\nu)} \qquad \text{Eq. 8-1}$$

E and G may not both be blank.

It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements, as indicated in **Table 8-1**.

Table 8-1 Material Property Usage Versus Element Types

Element Entry	E	NU	G
CROD CBEAM CBAR	Extension and Bending	Not Used	Torsion Transverse Shear
CQUAD4, CQUAD8 CTRIA3, CTRIA6	Membrane and Bending		Transverse Shear
CSHEAR	Not Used		Shear
CHEXA CPENTA CTETRA	All Terms		Not Used

Mass density RHO is used for GRAV loads as well as problems in dynamic analysis. The thermal expansion coefficient A and reference temperature TREF are used only in thermal analysis problems. Structural damping GE is not used in static analysis. Optional margin of safety calculations can be performed by supplying yield values for the material on the ST, SC, and SS entries. These are extra arithmetic calculations and have no effect on the basic MSC.Nastran results. MCSID concerns material coordinate systems-material coordinate systems are used with two- and three-dimensional elements to orient orthotropic and anisotropic material axes with respect to the element coordinate system.

Example

Create a MAT1 entry for a linear static problem using mild structural steel. The material properties are $E = 30 \times 10^6 lb_f / in^2$, $\nu = 0.3$ and mass density $= 7.0 \times 10^{-4}\ \mathrm{lb_f{-}sec^2/in}$

1	2	3	4	5	6	7	8	9	10
MAT1	5	30.E6		0.3	7.0E-4				

Or in free field format,

```
MAT1,5,30.E6,,0.3,7.0E-4
```

8.3 Other Material Types Available in MSC.Nastran

Although discussion of other material models is beyond the scope of this book, the following additional material types are listed here for reference:

MAT2	Anisotropic material for two-dimensional elements.
MAT3	Orthotropic material for axisymmetric solid elements.
MAT4	Thermal properties for temperature-independent isotropic materials.
MAT5	Thermal properties for temperature-independent anisotropic materials.
MAT8	Orthotropic material for two-dimensional elements.
MAT9	Anisotropic material for solid elements.
MAT10	Material properties for fluid elements in fluid-structure interaction analysis.
MATHP	Hyperelastic material properties for use in fully nonlinear analysis.
MATS1	Stress-dependent material properties (nonlinear materials).
MATTi	Temperature-dependent material properties.
MFLUID	Fluid volume properties for virtual mass.

CHAPTER

9

Applying Static Loads

- Introduction
- Concentrated Loads
- Distributed Load on a 1-D Element (PLOAD1)
- Pressure Loads
- Acceleration Loads (GRAV)
- Enforced Displacements
- Combining Loads (LOAD)
- Using Subcases (SUBCASE, SUBCOM, SUBSEQ)

9.1 Introduction

Overview of Basic Static Loads

MSC.Nastran offers an extensive library of load types. The most commonly used loads for linear static analysis are summarized in **Table 9-1**:

Table 9-1 Basic Static Loads

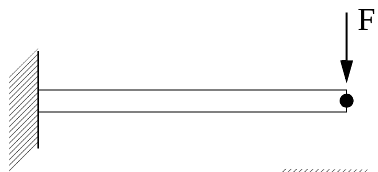
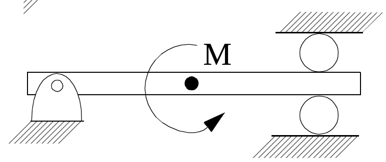
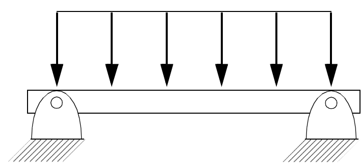
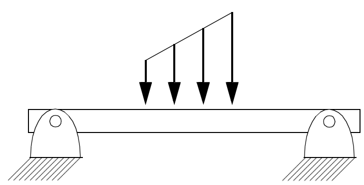
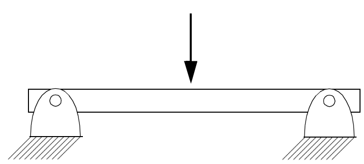
Description	Examples	See	Bulk Data Entry
Concentrated loads applied to grid points.	 	Section 9.2	FORCE MOMENT
Distributed or concentrated loads on line elements.	  	Section 9.3	PLOAD1

Table 9-1 Basic Static Loads (continued)

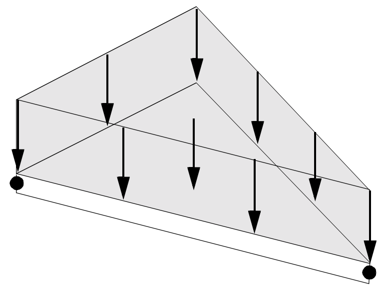
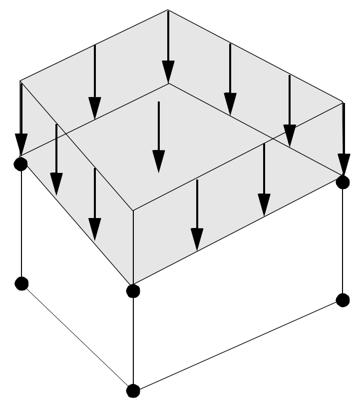
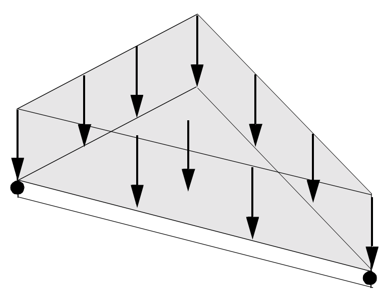
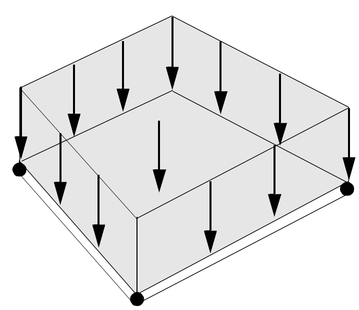
Description	Examples	See	Bulk Data Entry
Normal uniform pressure load on a triangular or quadrilateral surface (by defining grid points). This is for either 2D or 3D elements.	 	Section 9.4	PLOAD
Normal uniform pressure load on a 2-D element (by defining element IDs).	 	Section 9.4	PLOAD2

Table 9-1 Basic Static Loads (continued)

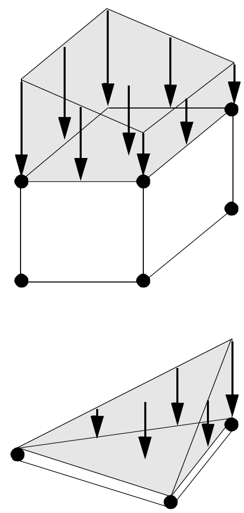
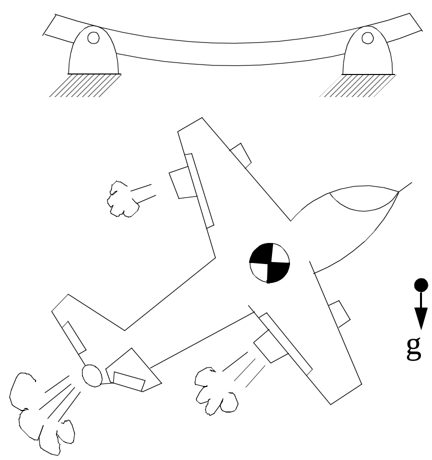
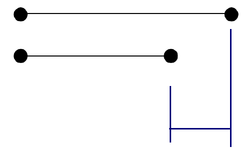
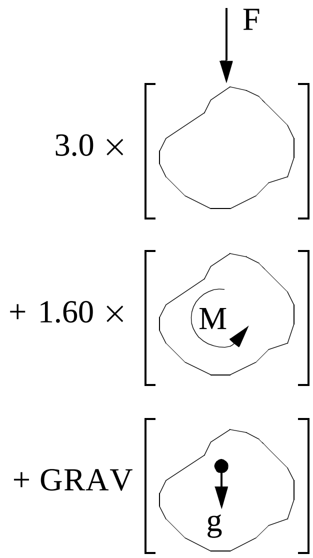
Description	Examples	See	Bulk Data Entry
Normal or traction pressure load on the face of a 2-D or 3-D element (by specifying element IDs). Pressure can be uniform or linearly varying.		Section 9.4	PLOAD4
Gravity or acceleration loads		Section 9.5	GRAV

Table 9-1 Basic Static Loads (continued)

Description	Examples	See	Bulk Data Entry
Axial deformation of one-dimensional elements		Section 9.6	DEFORM
Linear combinations of loads		Section 9.7	LOAD

Load Sets

The concept of sets-collections of entities of a particular type-is pervasive throughout MSC.Nastran. Load sets in linear statics problems are collections, or lists, of loads selected by Case Control commands. The Case Control command used depends on the type of load or loads being applied. For the loads described in this chapter, the general form of the Case Control command is either

```
LOAD=n
```

or

```
DEFORM=n
```

where n is an arbitrary user-defined set identification number (SID) in field 2 of the load Bulk Data entry. All loads with the same SID are combined into a load set.

The applicability of each Case Control command is shown in the following table.

Case Control Command	Bulk Data Entry
LOAD	FORCE, FORCE1, FORCE2 MOMENT, MOMENT1, MOMENT2 PLOAD, PLOAD1, PLOAD2, PLOAD4 GRAV* LOAD* SPCD*
DEFORM	DEFORM

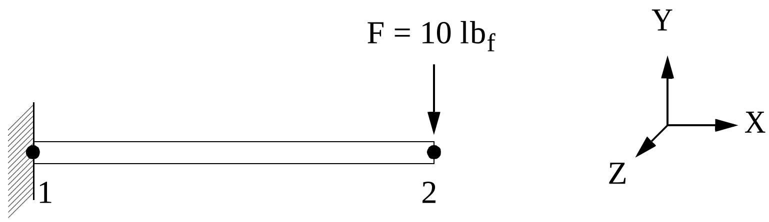
* Special rules apply-see individual load descriptions for details.

9.2 Concentrated Loads

This section shows how to apply concentrated forces and concentrated moments at grid points.

Forces (FORCE)

Consider a force F acting on a cantilever beam:



A concentrated force F is applied to a CBAR element connecting grid points 1 and 2. The Bulk Data entry required to specify this load is called FORCE. Its format is as follows:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one $\text{Ni} \neq 0.0$)

In our example, the FORCE entry may be written as:

FORCE	2	2		10.	0.	-1.	0.		

or, in free field format,

```
FORCE,2,2,,10.,0.,-1.,0.
```

The load set identification number (SID in column 2) refers to a command defined in the Case Control Section (LOAD=2 in this example; the integer value 2 is arbitrarily chosen). Leaving column 4 blank means that the basic (default) coordinate system is used to specify the orientation of the load. The (0., -1., 0.) entries in columns 6, 7, and 8 refer to a vector in the -Y direction, defining the direction of application of the load. The force applied to the grid point is $\vec{f}$, given by

$$\vec{f} = F\vec{N} \qquad \text{Eq. 9-1}$$

where $\vec{N} = (N1,N2,N3)$

Thus, the value of F in column 5 is the full value of the applied load of 10 lb_f because vector $\vec{N}$ (in this example) is of unit length.

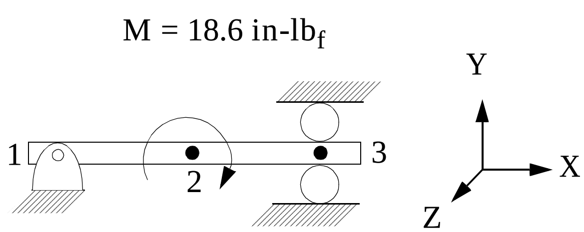
MSC.Nastran provides two other methods to apply a concentrated force:

FORCE1	Uses two grid points, not necessarily the same as the loaded grid point, to define load direction.
FORCE2	Defines the direction of the force as parallel to the cross product of two vectors.

The FORCE1 and FORCE2 entries are described in detail in the *MSC.Nastran Quick Reference Guide*. The type of FORCE entry to use is simply a matter of preference.

Moments (MOMENT)

The application of concentrated moments is analogous to forces. Consider moment M acting about the basic Z axis of the simply supported beam shown below:



The Bulk Data entry used to apply this load is called MOMENT and has the following format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one $Ni \neq 0.0$)

In our case, the MOMENT entry could be written as

1	2	3	4	5	6	7	8	9	10
MOMENT	6	2		-18.6	0.0	0.0	1.		

or, in free field format,

```
MOMENT,6,2,,-18.6,0.,0.,1.
```

The applied moment $\vec{m}$ is given by

$$\vec{m} = M\vec{N} \qquad \text{Eq. 9-2}$$

where $\vec{N}$ is the vector (N1, N2, N3). As was the case with FORCE entry, M is the full magnitude of the moment since $\vec{N}$ is a vector of unit length. The direction of the applied moment is given by the sign of M according to the right-hand rule because $\vec{N} = (0., 0., 1.)$ is a vector in the direction of the positive Z axis direction. Note that specifying $M = 18.6$ and $\vec{N} = (0., 0., -1.)$ would produce an equivalent result.

MSC.Nastran provides two other methods to apply a concentrated moment:

MOMENT1	Uses two grid points to determine direction ($\vec{m} = M\vec{n}$, where $\vec{n}$ is a unit vector parallel to the vector from grid 1 to grid 2).
MOMENT2	Uses four grid points to determine direction ($\vec{m} = M\vec{n}$, where $\vec{n}$ is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4).

Again, the choice of which entry to use is a matter of preference.

9.3 Distributed Load on a 1-D Element (PLOAD1)

The PLOAD1 Bulk Data entry is used to apply a uniformly or linearly varying distributed load to CBAR and CBEAM elements.

The load can be applied along the entire element length, a segment of the length, or at a point along the length. The form of the PLOAD1 entry is shown below. The meanings of X1, X2, P1, and P2 are shown in **Figure 9-1**.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number. (Integer > 0)
TYPE	Load type. (Character: “FX”, “FY”, “FZ”, “FXE”, “FYE”, “FZE”, “MX”, “MY”, “MZ”, “MXE”, “MYE”, “MZE”)
SCALE	Determines scale factor for X1, X2. (Character: “LE”, “FR”, “LEPR”, “FRPR”)
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank, $0 \leq X1 \leq X2$)
P1, P2	Load factors at positions X1, X2. (Real or blank)

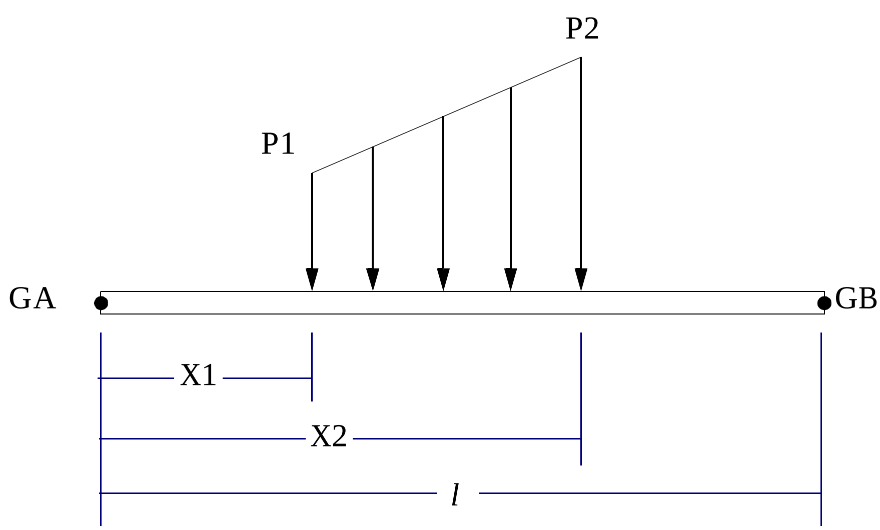


Figure 9-1 PLOAD1 Convention in CBAR or CBEAM Elements

The load set ID (SID) is selected by the Case Control command LOAD=SID. The available load TYPEs are defined as follows:

FX	Force in the X direction of the basic coordinate system.
FY	Force in the Y direction of the basic coordinate system.
FZ	Force in the Z direction of the basic coordinate system.
FXE	Force in the x direction of the element coordinate system.
FYE	Force in the y direction of the element coordinate system.
FZE	Force in the z direction of the element coordinate system.
MX	Moment in the X direction of the basic coordinate system.
MY	Moment in the Y direction of the basic coordinate system.
MZ	Moment in the Z direction of the basic coordinate system.
MXE	Moment in the x direction of the element coordinate system.
MYE	Moment in the y direction of the element coordinate system.
MZE	Moment in the z direction of the element coordinate system.

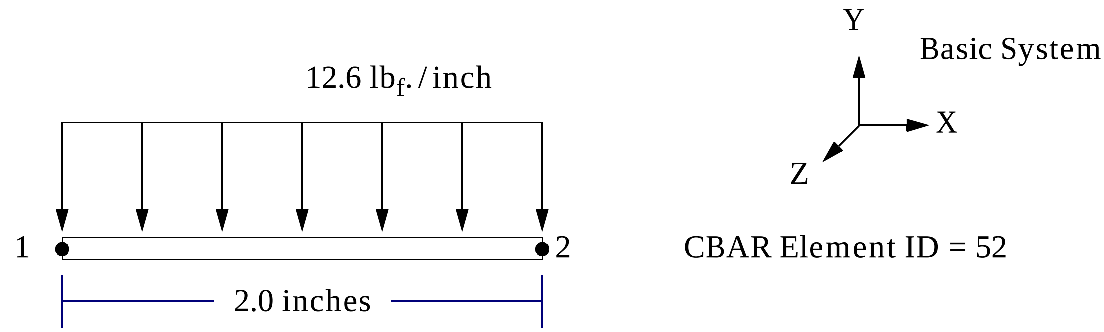
The SCALE factors mean the following:

LE	LENGTH - the values of Xi are the actual length of the element.
FR	FRACTIONAL - the values of Xi are fractional distances (fraction of length of element) along the length of the element starting from end A.
LEPR	LENGTH PROJECTED - the Xi values are actual distances along the element axis, and the distributed load is input in terms of the projected length of the element.
FRPR	FRACTIONAL PROJECTED - the Xi values are ratios of the actual distance to the length of the element, and the distributed load is specified in terms of the projected length of the element.

Additional details concerning projected loads can be found in the ***MSC.Nastran Quick Reference Guide*** under Bulk Data, PLOAD1 entry. We will concentrate on Length and Fractional loads.

All of this may look rather confusing, but it is really quite simple. The best way to demonstrate the use of PLOAD1 is with a few examples.

Example 1 Use a PLOAD1 entry to apply a uniformly distributed load over the full length of a CBAR element using fractional (normalized) scaling.



Note that P1 = P2 = 12.6 lb_f / inch

X1 = 0.0
X2 = 0.0

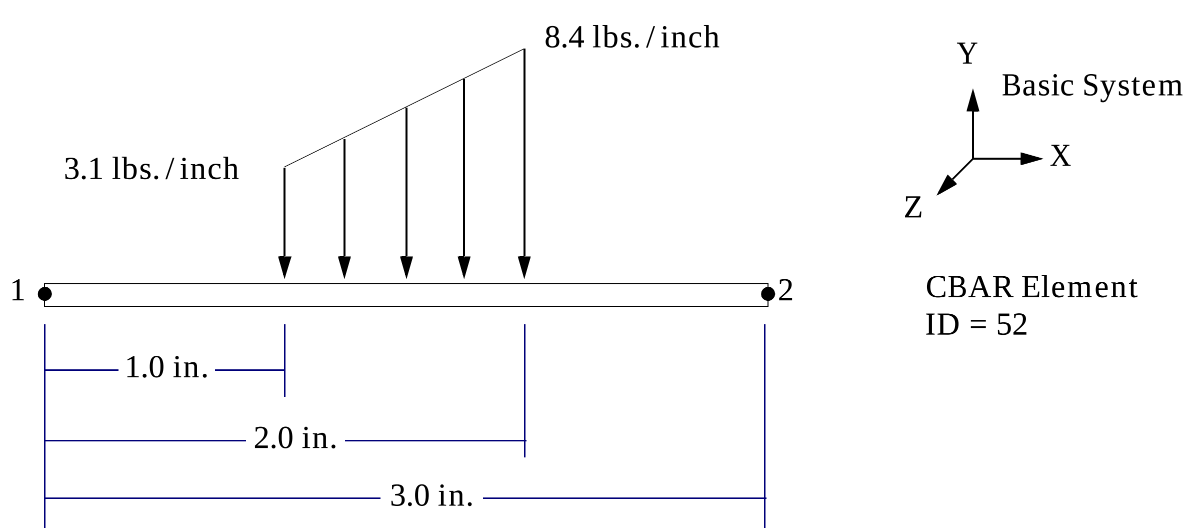
fractional scaling

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	FR	0.0	-12.6	1.0	-12.6	

Example 2 Same as Example 1, but with length scaling.

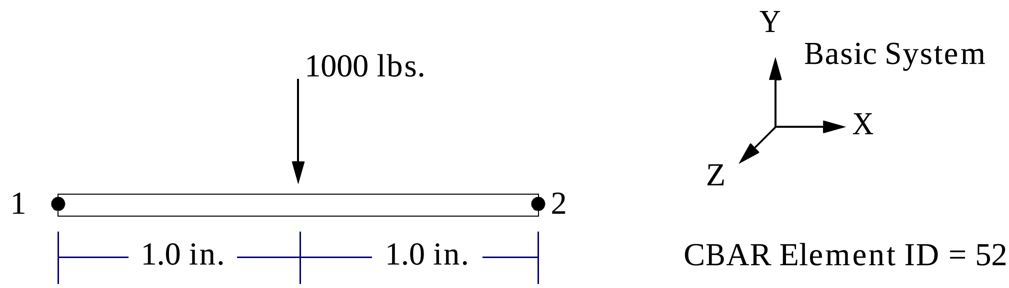
1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	LE	0.0	-12.6	2.0	-12.6	

Example 3 Use a PLOAD1 entry to apply a linearly varying distributed load to the interior of a CBAR element using length scaling.



1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	LE	1.0	-3.1	2.0	-8.4	

Example 4 Use a PLOAD1 entry to apply a concentrated load at an interior point of a CBAR element using fractional scaling.



1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	FR	0.5	-1000.			

9.4 Pressure Loads

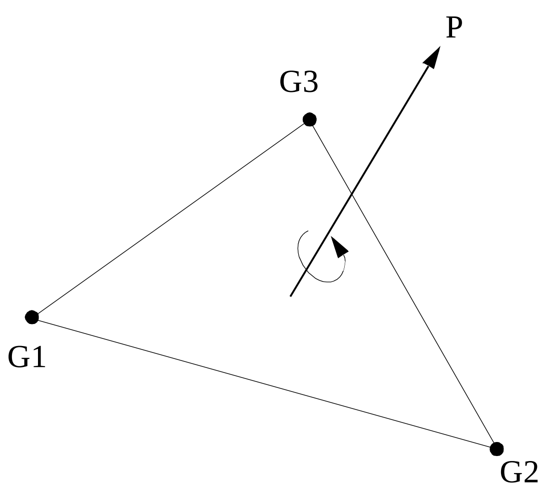
Uniform Normal Pressure Load on a Triangular or Quadrilateral Surface (PLOAD)

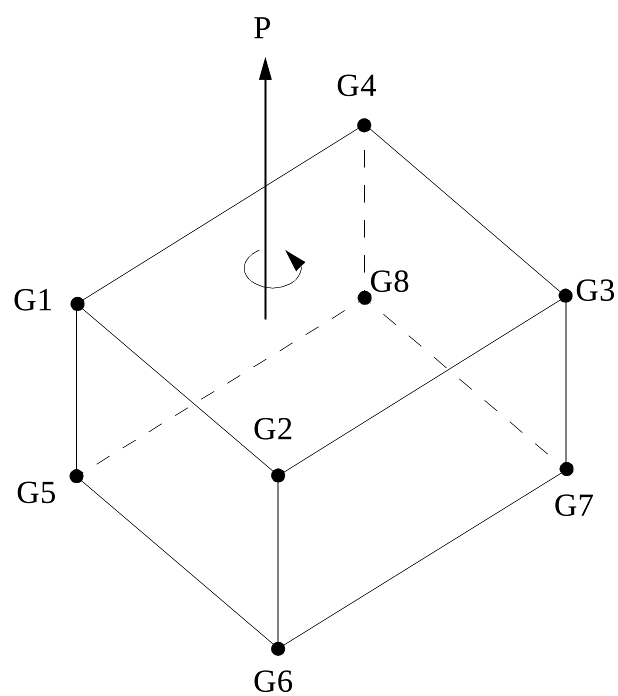
The PLOAD Bulk Data entry is used to define a uniform normal static pressure load on triangular or quadrilateral surfaces by using grid points. The PLOAD entry may be applied to 2-D (surface) or 3-D (solid) elements, and has the following form:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure. (Real)
Gi	Grid point identification numbers. (Integer > 0; G4 may be zero or blank)

Grid Points G1, G2, G3, and G4 define either a triangular or quadrilateral surface; if G4 is zero or blank, the surface is triangular. The direction of the pressure load is determined by applying the right-hand rule to the grid point ordering sequence of the surface.





Pressure is applied in the opposite direction by making the value of P negative.

Uniform Normal Pressure Load on a 2-D Element (PLOAD2)

The PLOAD2 Bulk Data entry is used to apply a normal uniform pressure load to CQUAD4 or CTRIA3 2-D (surface) elements using element IDs.

The PLOAD2 entry has two forms:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Alternate form:

PLOAD2	SID	P	EID1	"THRU"	EID2				

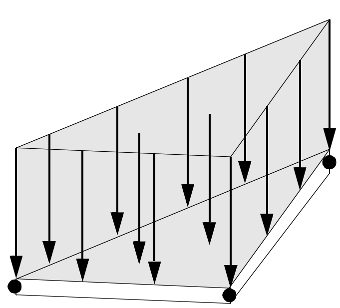
Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
EIDi	Element identification number. (Integer $\geq$ 0 or blank; for the "THRU" option, EID1 < EID2)

The load set ID (SID) is selected by the Case Control command LOAD=SID. The direction of the pressure is determined using the connected GRID points in the same right-hand rule sense as the PLOAD entry (i.e., with respect to the positive element z axis). In addition, when using the "THRU" option, all referenced elements must actually exist.

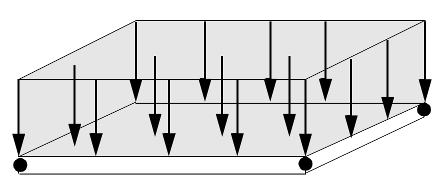
Normal or Traction Pressure Load on the Face of a 2-D or 3-D Element (PLOAD4)

The PLOAD4 Bulk Data entry defines a pressure load on the face of a variety of surface or solid elements. The pressure load can either be normal to the surface, or contain a traction (not normal to the surface) component. In addition, a different value of pressure can be entered at each corner. The PLOAD4 entry applies to the following elements described in this book:

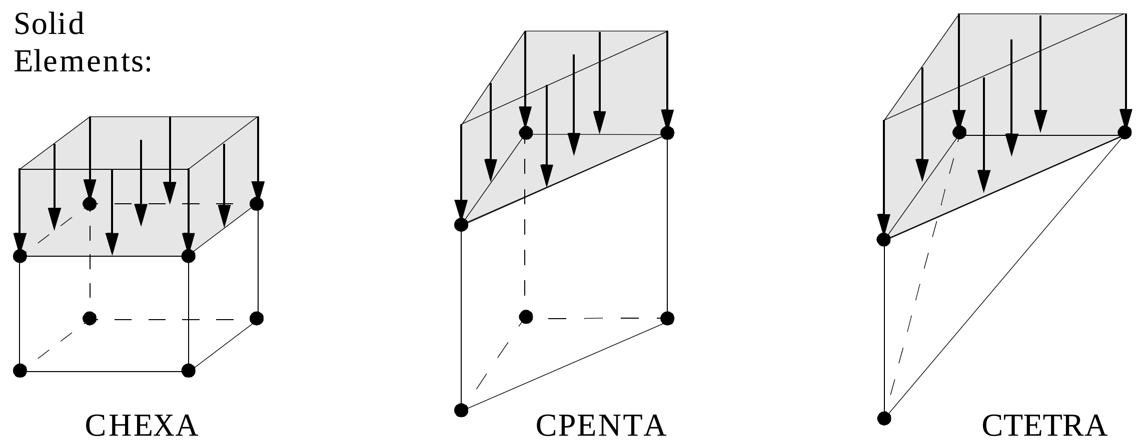
Surface Elements:



CTRIA3



CQUAD4



The format of the PLOAD4 entry is shown below:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

Alternate Format (available only for surface elements):

PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3					

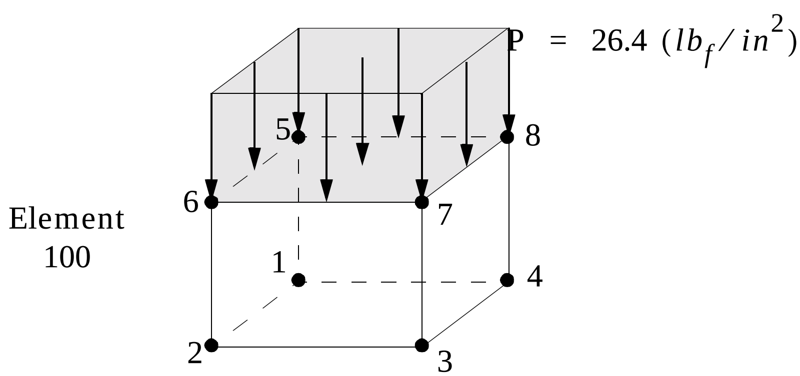
Field	Contents
SID	Load set identification number. (Integer > 0)
EID EID1 EID2	Element identification number. (Integer > 0; for the “THRU” option, EID1 < EID2)
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1)
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA or CPENTA element. Required data for quadrilateral faces of CHEXA and CPENTA elements only. G3 must be omitted for a triangular surface on a CPENTA element.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. (Real)

The load set ID (SID) is selected by the Case Control command LOAD = SID. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1; P4 is left blank for a triangular face. In addition, for pressure that acts normal to the face, the continuation entry is not used. For details about how to apply pressure loads in directions other than normal to the surface (traction loads) see the description of “**PLOAD4**” on page 1871 of the .

The direction of positive pressure for surface elements is the positive normal, determined by applying the right-hand rule to the sequence of grid points on the loaded face. Load intensities P1, P2, P3, (and P4) act at corner points G1, G2, G3, (and G4) for triangular (and quadrilateral) elements. The default direction of positive pressure for faces of solid elements is inward. In addition, when using the “THRU” option, all referenced elements must actually exist.

Example

Specify the PLOAD4 entry for a uniform normal pressure load applied to the CHEXA solid element shown in the following figure:



1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

PLOAD4	12	100	26.4				5	7	

This load is selected in the Case Control Section with the command LOAD = 12. Leaving P2, P3, and P4 blank assigns a uniform pressure value of $6.4\ lb/in^2$.

9.5 Acceleration Loads (GRAV)

This section shows how to apply "static" acceleration loads to your model. Examples of common static acceleration loads are gravity (the response of a structure to its own weight) and vehicle maneuver loads (perhaps a pilot or an equipment package pulling g's in a rapid turn). The acceleration causes a static load, and not a dynamic one - the structural response is steady state (static) with the transient part being zero. In all cases, both the magnitude of the acceleration and the direction in which it acts must be specified.

Acceleration is applied using the GRAV Bulk Data entry, which has the following format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one $\text{Ni} \neq 0.0$)
MB	Used only in superelement analysis.

The direction and magnitude of acceleration are given by $\vec{a} = A\vec{N}$ where the vector $\vec{N}$ = (N1, N2, N3) gives the direction. The magnitude of $\vec{a}$ is equal to A times the magnitude of $\vec{N}$. Note, for example, that entering the value of A in $n/\sec^2$ indicates that other specifications of the model that involve length units (element length, moments of inertia, modulus of elasticity, etc.) must also be in inches to preserve unit consistency.

For models that have only GRAV loads, the load set ID (SID) is selected in the Case Control Section with a LOAD = SID Case Control command. If other types of loads (such as FORCE or PLOAD) are also present in the model, they must be combined with the GRAV load using a LOAD Bulk Data entry (see "**Combining Loads (LOAD)**" on page 28 for more information on the LOAD Bulk Data entry).

Finally, the model must include mass density information to use acceleration loads. Mass density is entered on the material property Bulk Data entry-MAT1 in our case. For example, for a model of typical structural steel using English units, the mass density might be

$$\rho_m = 7.0 \times 10^{-4} lb_f \cdot \sec^2/\text{in}^4$$

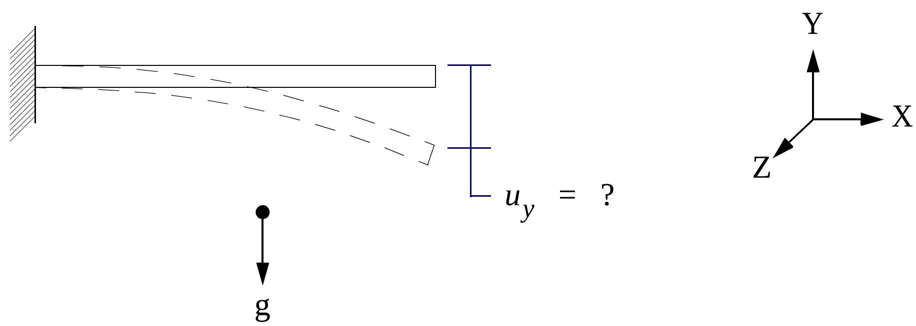
Thus, the MAT1 Bulk Data entry might look like the following:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	12	30.E6		0.3	7.0E-4				

See “**Material Properties**” on page 7, for further discussion of the MAT1 Bulk Data entry.

Example

What is the tip deflection of the cantilever beam due to its own weight?



First, observe that the force of gravity (g) acts in the -Y direction. We must tell MSC.Nastran which way is down (or, more precisely, where the center of the earth is). Thus, the vector $\vec{N}$ can be written as:

$$\vec{N} = (0., -1., 0.) \tag{Eq. 9-3}$$

The acceleration due to gravity on the Earth’s surface is approximately $2.2\ \mathrm{ft/sec}^2$ $(86.4\ \mathrm{in/sec}^2)$ or $.8\ \mathrm{m/sec}^2$. Recall that length units must be consistent throughout the model (this example uses inches).

Thus, the GRAV entry is as follows:

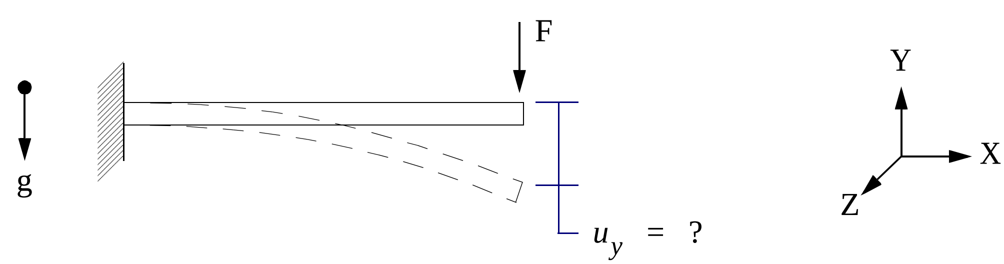
1	2	3	4	5	6	7	8	9	10
GRAV	15		386.4	0.	-1.	0.			

or, in free field format,

```
GRAV,15,,386.4,0.,-1.,0.
```

The Case Control command required to apply this load is LOAD = 15.

Now assume that a concentrated force is added to the beam as shown:



What is the tip deflection of the cantilever beam due to its own weight and the concentrated force?

To combine gravity loading with the concentrated force, the following approach must be used:

In Case Control: `LOAD = 15`

In Bulk Data:

LOAD	15	1.0	1.0	12	1.0	13			

and

GRAV	12
FORCE	13

The essential idea is that the set ID on a GRAV entry may not be the same as the Set ID on any other load entry. This restriction has no engineering significance-it is simply the way MSC.Nastran was designed.

As a final example, suppose you are analyzing an instrument package subjected to inertial loads in specific directions, say:

$$1.3 \text{ g in the x-direction } (1.3g = 1.3 \cdot 9.8 \text{ m/sec}^2 = 12.7 \text{ m/sec}^2).$$

$$2.8 \text{ g in the y-direction } (2.8g = 2.8 \cdot 9.8 \text{ m/sec}^2 = 27.4 \text{ m/sec}^2).$$

$$0.3 \text{ g in the z-direction } (0.3g = 0.3 \cdot 9.8 \text{ m/sec}^2 = 2.9 \text{ m/sec}^2).$$

Three separate GRAV entries can easily be written as:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

GRAV	15		12.7	1.	0.	0.			
GRAV	15		27.4	0.	1.	0.			
GRAV	15		2.9	0.	0.	1.			

9.6 Enforced Displacements

Static Element Deformation (DEFORM)

The DEFORM Bulk Data entry is used to apply an axial deformation to line elements in statics problems. The format of the DEFORM entry is shown below:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

Field	Contents
SID	Deformation set identification number. (Integer > 0)
EIDi	Element number. (Integer > 0)
Di	Deformation. (Real; positive value represents elongation)

The DEFORM entry is selected in the Case Control Section with the command DEFORM=SID. From one to three enforced element deformations may be defined on a single entry. Note that a positive value of Di represents elongation and a negative value represents contraction.

Enforced Displacement Value (SPCD)

Enforced grid point displacements in static analysis may be applied using the SPCD Bulk Data entry. SPCD has the following format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Field	Contents
SID	Identification number of a static load set. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. ($0 \le$ Integer ≤ 6; up to six unique Integers may be placed in the field with no embedded blanks)
Di	Value of enforced displacement at Gi and Ci. (Real)

For static solutions, the load set ID (SID) is selected by the Case Control command LOAD = SID.

9.7 Combining Loads (LOAD)

The LOAD entry defines a static load as a linear combination (superposition) of load sets defined using the FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD3, PLOAD4, PLOADX, SLOAD, SPCD, RFORCE, or GRAV entries.

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Overall scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)

The resulting combined load is determined by:

$$\text{LOAD} = S \sum_i Si \cdot \{Li\} \qquad \text{Eq. 9-4}$$

Where $\{Li\}$ represents the applied load vector corresponding to load set ID Li.

All load set IDs (Li in the equation above) must be unique. The LOAD entry must be used if acceleration loads (the GRAV entry) are to be used with any other type of load (e.g. FORCE). Up to 300 (Si, Li) pairs may be used with one LOAD entry.

Example

Assume that your model has one concentrated force of 15.2 lbs in the y direction applied to grid point 12, and one concentrated moment of 6.4 inch-lbs about the x-axis applied to grid point 127. It is required to double the value of force and triple the value of moment for your next analysis.

The LOAD Bulk Data entry may be written with an overall scale factor (S) of 1.0 and loadset scale factors (Si) of 2.0 for force and 3.0 for moment. Thus,

$$\begin{aligned} \text{LOAD} &= S \sum_i Si \cdot \{Li\} \\ &= 1.0[2.0\{L1\} + 3.0\{L2\}] \end{aligned}$$

In Case Control:

```
LOAD = 22
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
LOAD	22	1.0	2.0	30	3.0	40			
FORCE	30	12		15.2	0.	1.	0.		
MOMENT	40	127		6.4	1.	0.	0.		

9.8 Using Subcases (SUBCASE, SUBCOM, SUBSEQ)

MSC.Nastran allows you to efficiently analyze multiple load cases in one run using the SUBCASE Case Control command (each subcase defines a unique loading condition). In addition, linear combinations of subcases can be combined to create additional subcases using the SUBCOM command. Coefficients defining the linear combination of a SUBCOM are specified on a SUBSEQ command. A SUBSEQ command is required for each SUBCOM subcase.

The use of subcases is best illustrated with an example. The Case Control Section for an equipment rack analysis is shown in **Listing 9-1**.

Listing 9-1 SUBCASE and SUBCOM Case Control Section

Case Control requests above the first subcase level apply to all subcases unless overridden by requests within a subcase.

```
CEND
TITLE=Equipment Rack Analysis
ECHO=BOTH
SPC=20
SET 1=1 THRU 50
DISP=1
SUBCASE 1
     SUBTITLE=Dead Load
     LOAD=10
     DISP=All
SUBCASE 2
     SUBTITLE=NW Wind Load
     LOAD=20
     SET 10=2,4,6
     DISP=10
SUBCASE 3
     SUBTITLE=SW Wind Load
     LOAD=30
SUBCOM 10
     SUBTITLE=Load Combination 1
     LABEL=Dead Load + NW Wind
     SUBSEQ=1.0,1.0,0.0
SUBCOM 20
     SUBTITLE=Load Combination 2
     LABEL=Dead Load + (-) 1.5 SW Wind
     SUBSEQ=1.0,0.0,-1.5
     STRESS=ALL
     BEGIN BULK
```

Loads or sets defined within a subcase are limited to that subcase only.

Above the first SUBCASE delimiter (known as “above the subcase level”), we have the following:

TITLE means that	“EQUIPMENT RACK ANALYSIS” will appear on each page of output in all subcases.
ECHO = BOTH	will cause both sorted and unsorted Bulk Data listings to be printed for each subcase.

SPC = 20	defines the model's constraint set, which remains the same for each subcase.
SET 1 = 1 THRU 50	defines a set of entities with IDs from 1 to 50.
DISP = 1	refers to set 1, and will cause the displacements of grid points 1 through 50 to be printed for all subcases unless overridden within a subcase (as occurs in subcases 1 and 2).

The subtitles or labels in each subcase or subcom are added to the title "EQUIPMENT RACK ANALYSIS." The SUBSEQ entries have the following meaning:

In SUBCOM 10:

SUBSEQ = 1.0 (SUBCASE 1 Load) + 1.0 (SUBCASE 2 Load) + 0.0 (SUBCASE 3 Load)
= 1.0 (LOAD SET 10) + 1.0 (LOAD SET 20)
= 1.0 · Dead Load + 1.0 · NW Wind Load

Note that when using SUBSEQ, a coefficient must be given for each of the preceding subcases. A coefficient of 0.0 indicates that a subcase does not contribute to the SUBCOM. The 0.0 placeholder must still be present, however.

In SUBCOM 20:

SUBSEQ = 1.0 (SUBCASE 1 Load) + 0.0 (SUBCASE 2 Load) - 1.5 (SUBCASE 3 Load)
= 1.0 (LOAD SET 10) - 1.5 (LOAD SET 30)
= 1.0 · Dead Load - 1.5 · SW Wind Load

CHAPTER
10

Controlling the Analysis Output

10.1 Printing the Input File (ECHO)

An MSC.Nastran model resides in a .DAT text file. The .DAT file is submitted to MSC.Nastran and an .f06 results file is produced. The following Case Control command will produce sorted and unsorted model file listings, called echoes, at the beginning of the .f06 file:

```
ECHO=BOTH
```

The unsorted input file is an exact copy of the Executive Control, Case Control, and Bulk Data Sections of the input (.DAT) file, including comment ($) entries. The sorted input file is a listing of the Bulk Data Section with entries rearranged in alphabetical order and with comments removed. In addition, the sorted Bulk Data is expanded to ten fields, each eight columns wide. Therefore, if the .DAT file is entered in free field format, it will appear in small field format in the sorted Bulk Data listing. This small field format listing is especially helpful when reviewing the model's Bulk Data Section.

Examples of unsorted and sorted input listings are shown in **Listing 10-1** and **Listing 10-2**.

Listing 10-1 Unsorted Listings

```
                                                      SEPTEMBER  10, 2001  MSC.NASTRAN  9/10/01   PAGE     1

         N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O

      ID MPM,EXAMPLE1
      SOL 101
      TIME 100
      CEND
```

Listing 10-1 Unsorted Listings (continued)

```
SIMPLY SUPPORTED BEAM                                  SEPTEMBER  10, 2001  MSC.NASTRAN  9/10/01   PAGE     2
     WITH CONCENTRATED FORCE
                                        C A S E    C O N T R O L   D E C K   E C H O
          CARD
          COUNT
           1      ECHO=BOTH
           2      DISP=ALL
           3      STRESS=ALL
           4      FORCE=ALL
           5      SPCF=ALL
           6      SPC=100
           7      LOAD=10
           8      TITLE=SIMPLY SUPPORTED BEAM
           9      SUBTITLE=WITH CONCENTRATED FORCE
          10      $
          11      BEGIN BULK

SIMPLY SUPPORTED BEAM                                  SEPTEMBER  10, 2001  MSC.NASTRAN  9/10/01   PAGE     3
     WITH CONCENTRATED FORCE
                                             I N P U T   B U L K   D A T A   D E C K   E C H O
                            .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                          $     DEFINE GRID POINTS
                          GRID,1,,0.,0.,0.
                          GRID,2,,10.,0.,0.
                          GRID,3,,20.,0.,0.
                          GRID,4,,30.,0.,0.
                          $
                          $     DEFINE CBAR ELEMENTS
                          CBAR,1,101,1,2,0.,1.,0.
                          CBAR,2,101,2,3,0.,1.,0.
                          CBAR,3,101,3,4,0.,1.,0.
                          $
                          $     DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES
                          PBAR,101,201,2.,.667,.1667,.458,,,+PB1
                          +PB1,1.,.5,-1.,.5
                          $
                          $     DEFINE MATERIAL PROPERTIES
                          MAT1,201,30.E6,,.3
                          $
                          $     DEFINE  SPC CONSTRAINT SET
                          SPC1,100,12345,1
                          SPC1,100,12345,4
                          $
                          $     DEFINE CONCENTRATED FORCE
                          FORCE,10,3,,-100.,0.,1.,0.
                          $
                          ENDDATA

                          INPUT BULK DATA CARD COUNT =      26
```

Listing 10-2 Sorted Listing

```
SIMPLY SUPPORTED BEAM                                        SEPTEMBER  10, 2001  MSC.NASTRAN  9/10/01   PAGE     4
     WITH CONCENTRATED FORCE
                                                      S O R T E D   B U L K   D A T A   E C H O
             CARD
             COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                1-        CBAR    1       101     1       2       0.      1.      0.
                2-        CBAR    2       101     2       3       0.      1.      0.
                3-        CBAR    3       101     3       4       0.      1.      0.
                4-        FORCE   10      3               -100.   0.      1.      0.
                5-        GRID    1               0.      0.      0.
                6-        GRID    2               10.     0.      0.
                7-        GRID    3               20.     0.      0.
                8-        GRID    4               30.     0.      0.
                9-        MAT1    201     30.E6           .3
               10-        PBAR    101     201     2.      .667    .1667   .458                    +PB1
               11-        +PB1    1.      .5      -1.     .5
               12-        SPC1    100     12345   1
               13-        SPC1    100     12345   4
                          ENDDATA

                    TOTAL COUNT=    14
```

Other options for the ECHO command include:

ECHO = SORT	Prints only sorted Bulk Data (this is the default)
ECHO = UNSORT	Prints only unsorted Bulk Data
ECHO = NONE	Turns off the Bulk Data listing
ECHO = PUNCH	Prints a sorted echo of the Bulk Data to a separate file

While learning to use MSC.Nastran by using small models it is useful to set ECHO to BOTH. The resulting listings in the .f06 file will not occupy very much space and having the model information available can be useful. However, when the models become very large it may be best to use ECHO = PUNCH or even ECHO = NONE.

10.2 Output Titles (TITLE, SUBTITLE, LABEL)

Up to three title lines per page of MSC.Nastran output are available by using the TITLE, SUBTITLE, and LABEL Case Control commands. Any character string can be used. Titles are optional but highly recommended-without them, one MSC.Nastran job looks pretty much like another.

Example

These Case Control commands

```
TITLE=SIMPLY SUPPORTED BEAM
SUBTITLE=WITH CONCENTRATED FORCE
LABEL=MSC.NASTRAN GETTING STARTED USER'S GUIDE
```

produce the following headings on each page of output:

```
SIMPLY SUPPORTED BEAM                                  SEPTEMBER   10,  2001  MSC.NASTRAN  9/10/01   PAGE    12
     WITH CONCENTRATED FORCE
    MSC.NASTRAN GETTING STARTED USER'S GUIDE

                                         D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0            0.0            0.0            0.0            0.0           -2.221112E-04
             2      G      0.0           -1.943473E-03   0.0            0.0            0.0           -1.388195E-04
             3      G      0.0           -2.221112E-03   0.0            0.0            0.0            1.110556E-04
             4      G      0.0            0.0            0.0            0.0            0.0            2.776390E-04
```

10.3 Case Control Sets (SET)

A Case Control set is a collection of grid point IDs or element IDs for use in output requests. Case Control sets are used to obtain output for a selected portion of the model. Case Control sets are defined with the SET command according to the following formats:

$$\text{SET n} = \text{ALL}$$

$$\text{SET n} = i_1, i_2, i_3, ..., i_{12} \text{ THRU } i_{28}, i_{35}, ..$$

where n is the set identification number and i_1, i_2, i_3,...etc., are entity identification numbers, e.g. grid point numbers.

Example

Consider the Case Control Section shown below:

```
CEND
TITLE=OUTPUT SELECTION EXAMPLE
SUBTITLE=ILLUSTRATES USE OF SETS
LOAD=15
SET 1=3,4,7,9,11
SET 5=2,9,15 THRU 21,23
DISP=1
FORCE=1
STRESS=5
BEGIN BULK
```

DISP is a grid point output quantity, so displacements for grid points 3, 4, 7, 9, and 11 will be printed. FORCE is an element output quantity, so forces for elements 3, 4, 7, 9, and 11 will be printed. Note that grid quantities and element quantities can share the same set, since a set is simply a list of numbers. STRESS is an element quantity, so the stresses in elements 2, 9, 15 through 21 (inclusive), and 23 will be calculated and printed.

10.4 Requesting Analysis Results

MSC.Nastran output is requested in the Case Control Section. In a small or medium-sized analysis, "asking for everything" is probably not an issue. In a large problem, output can become overwhelming and it may be desirable to be selective. In either case, you have complete control and considerable flexibility in your choice of output. Two basic classes of output are available: grid point quantities or element quantities.

Grid Point Output (DISP, SPCF, OLOAD, GPFORCE)

Requests for output quantities that occur at grid points include the following (n is a SET ID, and ALL requests that all quantities be printed):

`DISP=ALL`
or
`DISP=n`
	Requests displacements for grid points. It is recommended that DISP = ALL be used unless you have a good reason for doing otherwise- displacement output takes up very little space, and you don't always know a priori what you will be interested in reviewing.

`SPCF=ALL`
or
`SPCF=n`
	Requests forces of single point constraint (SPCFORCES). SPCF = ALL should always be used so that reaction forces can be examined during the model validation process.

`OLOAD=ALL`
or
`OLOAD=n`
	Requests the set of applied loads in static analysis.

`GPFORCE=ALL`
or
`GPFORCE=n`
	The GPFORCE request generates a table containing a grid point force balance at the selected grid points. This is useful for determining load paths, contributions of applied loads to element response, and the effects of initial thermal strain. Contributors to the grid point force balance table include applied loads, SPC forces, and element elastic forces.

Element Output (STRESS, FORCE, STRAIN, ESE)

Requests for element-based output quantities include the following:

`STRESS=ALL`
or
`STRESS=n`
	Requests element stresses for a set of structural elements.

`FORCE=ALL`
or
`FORCE=n`
	Requests element forces to be calculated for a set of structural elements.

`STRAIN=ALL`	Requests the strains for a set of plate elements.
or	
`STRAIN=n`	
`ESE=ALL`	Requests the strain energy for a set of elements.
or	
`ESE=n`	

CQUAD4 stresses, strains, and forces are available at corner grid points, with output at the element center the default. CQUAD4 center and corner output is obtained using the STRESS, STRAIN, and FORCE Case Control commands as follows:

```
STRESS(CORNER) = {ALL or n}
STRAIN(CORNER) = {ALL or n}
FORCE(CORNER) = {ALL or n}
```

Where { } indicates that a choice of ALL or n is mandatory, but the braces are not included.

Currently, only one type of element output (center or corner) is supported per run. For further information regarding CQUAD4 corner output, see the ***MSC.Nastran Quick Reference Guide***.

CHAPTER
11

Additional Considerations

11.1 MSC.Nastran User and System Messages

MSC.Nastran provides a variety of messages regarding the status of your finite element model and your computer system. There are six basic types of messages:

UIM	User Information Message
UWM	User Warning Message
UFM	User Fatal Message
SIM	System Information Message
SWM	System Warning Message
SFM	System Fatal Message

Message Format. The general format of a message is

$$* * * \begin{pmatrix} \text{SYSTEM} \\ \text{USER} \end{pmatrix} \begin{pmatrix} \text{FATAL} \\ \text{WARNING} \\ \text{INFORMATION} \end{pmatrix} \text{MESSAGE id, text.}$$

where “id” is a unique message identification number and “text” is the message as indicated in capital letters for each of the diagnostic messages. A series of asterisks (****) in the text indicates information that will be filled in by MSC.Nastran specifically for the message, such as a grid point ID or the name of a Bulk Data entry. Many of the messages are followed by additional explanatory material, including suggestions for remedial action.

User Messages. In general, User Messages have something to do with the finite element model (e.g., an incorrect input file). User Information Messages provide general information that is not necessarily indicative of a problem. User Warning Messages indicate that an atypical situation has been detected-you must review the warning to determine whether or not a problem actually exists. UIMs and UWMs can appear throughout the output file, and the execution of the program continues in a normal manner following the printing of the message. User Fatal Messages describe errors that are severe enough to cause MSC.Nastran to terminate and usually appear at the end of the MSC.Nastran .f06 output file.

System Messages. System Messages refer to diagnostics associated with program errors and are otherwise analogous to User Messages.

Additional explanations of some errors are provided. A short excerpt from the message list is shown in **Figure 11-1**.

MSC.Nastran SYSTEM AND USER MESSAGES

4697***USER FATAL MESSAGE 4697, THE FOLLOWING FREE FIELD CARD HAS MORE THAN TEN FIELDS SPECIFIED (CARD IGNORED).

A free field Bulk Data entry had more than ten fields specified. Only ten fields are allowed on a free field Bulk Data entry.

4698***USER WARNING MESSAGE 4698, STATISTICS FOR DECOMPOSITION OF MATRIX ****. THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN ****, OR HAVE NEGATIVE TERMS ON THE FACTOR DIAGONAL.

Module DCMP generates this message. During decomposition, the degrees of freedom listed have pivot ratios greater than maxratio or are negative. Verify that the degrees of freedom are not part of a mechanism and that elements do not have excessive stiffness. In superelement analysis, this will cause run termination. PARAM,BAILOUT may be used to continue the run.

4699***USER WARNING MESSAGE 4699, INPUT FIELD TO REPLICATOR HAS MORE THAN 8 COLUMNS. SOME DATA MAY BE DISCARDED.

The Bulk Data entry replicator reads the first eight fields of an entry, then discards any that may remain. This may lead to unintended results. Check all entries generated by the replicator if this message appears.

Figure 11-1 Excerpt from System and User Message List

11.2 Epsilon-A Measure of Numerical Behavior

In each MSC.Nastran linear static analysis run, a number called epsilon is automatically printed in the .F06 output listing. Epsilon is based on a strain energy error ratio, and provides an important measure of roundoff error and numerical ill-conditioning.

A system of linear equations is said to be ill-conditioned if small perturbations in the system lead to large changes in the solution. MSC.Nastran checks for evidence of ill-conditioning in the system of equations representing the structural model. A high value of epsilon indicates a potential ill-conditioning problem. Ill-conditioning does not necessarily result in a fatal error, but it can result in inaccurate answers. Possible causes of ill-conditioning include a high difference in stiffness between adjacent elements in the model, unconnected degrees of freedom, rigid body motion, or the presence of mecahnisms.

Epsilon should be one of the first items you examine when reviewing your output file. A small value of epsilon does not guarantee a correct solution, but it is certainly a necessary starting point. A large value of epsilon (greater than about 10^{-3}) is an indication of numerical ill-conditioning and requires further investigation. Possible causes of a large epsilon include rigid body motion, the presence of a mechanism, unreasonably stiff elements, or very large differences in stiffness between adjacent elements. It does not matter whether epsilon is positive or negative, as long as it is small. In addition, a value of, say 10^{-12} is not a "better" value than 10^{-10}; both are "small enough".

A sample epsilon printout is shown in **Figure 11-2**.

```
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

    LOAD SEQ. NO.           EPSILON              EXTERNAL WORK     EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
                1       -9.6854459E-16          4.7980525E-02
```

Figure 11-2 EPSILON Printout

11.3 Element Distortion and Accuracy

In general, the accuracy of a finite element degrades as its shape is distorted. We will examine distortions associated with CQUAD4 plate and CHEXA solid elements.

CQUAD4 Plate Element Distortion

There are four basic types of CQUAD4 element distortion: aspect ratio, warp, skew, and taper.

CQUAD4 Aspect Ratio. Aspect ratio is the ratio of the element's longest side to its adjacent side.

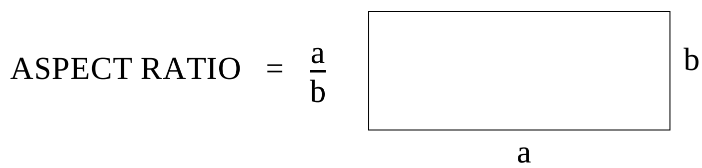


Figure 11-3 CQUAD4 Aspect Ratio

An element's aspect ratio $\frac{a}{b}$ should be less than about 4:1, and much less in regions where stress levels change rapidly. In regions of nearly uniaxial stress fields, larger aspects ratios are acceptable. MSC.Nastran does not perform an aspect ratio check on CQUAD4 elements.

CQUAD4 Warp. Warp is the extent to which an element deviates from being planar.

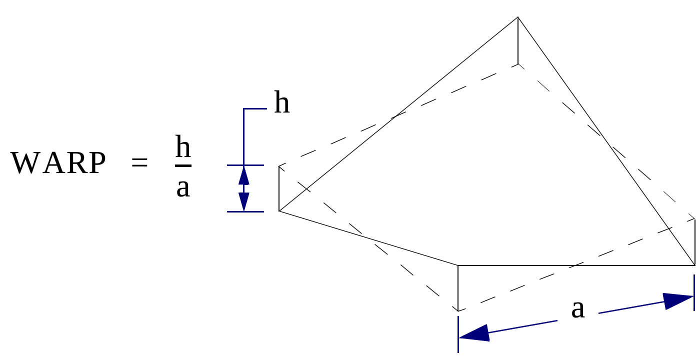


Figure 11-4 CQUAD4 Warp

Warp of up to about 5% is normally acceptable. MSC.Nastran does not check element warp.

CQUAD4 Skew. Skew is the angle between the lines that join opposite midsides.

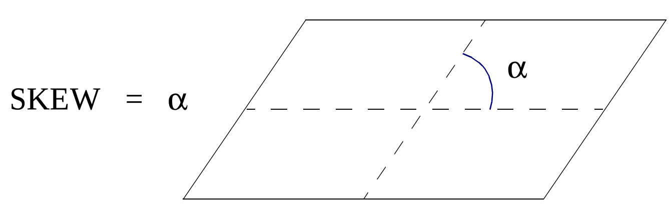


Figure 11-5 CQUAD4 Skew

For an element with no skew, $\alpha = 90$. When the skew angle is less than 30 degrees, User Information Message 5491 is issued. As a general rule, the CQUAD4 element should be kept as square as possible.

CQUAD4 Taper. Taper is the ratio of the areas on the two sides of a diagonal; if the ratio is greater than three, then the taper test performed by MSC.Nastran fails and User Information Message 5419 is issued. As a practical rule, taper angles should not exceed 30 degrees in most applications.

$$\text{TAPER} = \gamma$$

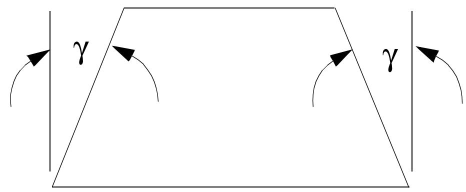


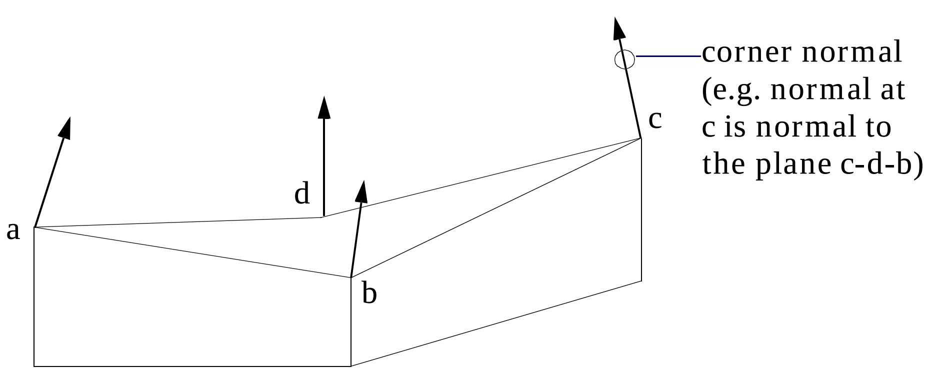
Figure 11-6 CQUAD4 Taper

CHEXA Solid Element Distortion

MSC.Nastran makes two element geometry checks for CHEXA elements: aspect ratio and face warping. These checks are made because the solution accuracy in regions of nonconstaint stress can degrade if the element geometry distorts significantly from its ideal cubic shape.

Aspect ratio is the ratio of the length of any two sides. If this ratio is greater than 100, indicating a very elongated CHEXA element, then User Information Message 4655 is issued for that element.

Warping indicates that a face of a CHEXA is not planar. This is indicated by User Information Message 4656, which indicates that a face is considerably out of plane. This message is issued if the corner normals at the opposite corners of a face deviate from each other by more than 45 degrees as shown in **Figure 11-7** (midside nodes are neglected in the calculation).



"Out of Plane" message is issued if the angle between normals at b and d > 45 degrees

"Out of Plane" message is issued if the angle between normals at a and c > 45 degrees

Figure 11-7 CHEXA Warping

11.4 The Current Error List

MSC Software maintains a comprehensive Current Error List (CEL) for recent versions of MSC.Nastran and recommends that all users review each edition of the Current Error List. The list is divided into two sections: general limitations and current errors. General limitations acknowledge and describe a lack of functionality in various areas of the program. An excerpt from the General Limitation List is shown in **Figure 11-8**.

CONEAX - Wrong Output

Conical shell problems do not prepare SORT2 output properly. In addition, the SORT2 requests generate unwanted and meaningless punched output. (903)

HEXA and QUAD8 - Stress Discontinuities

Reducing mesh sizes by connecting two elements to one with midside nodes causes local stress perturbations because the assumed displacement shapes at the interface are not compatible.

An avoidance is to use PENTA or TRIA3 elements to change the mesh size. (1405)

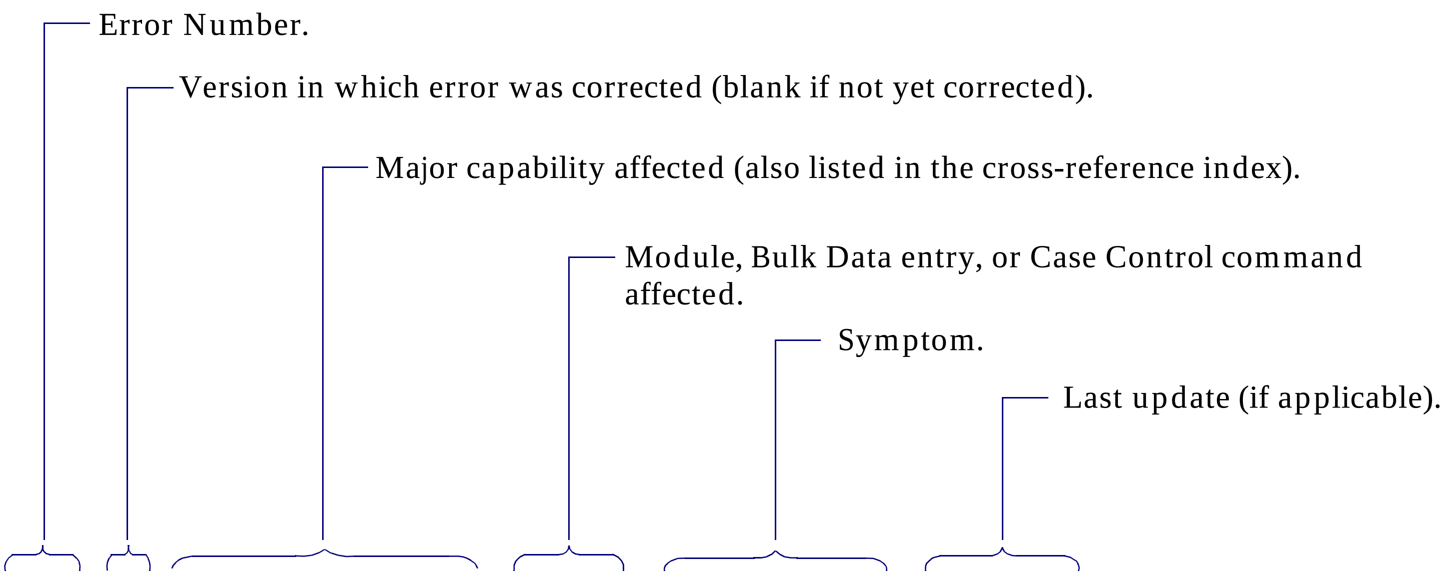
SOL 64 - Various Symptoms of Numerical Instability

If flat plate models have out-of-plane singularities when unloaded but have enough angle between adjacent plates to constrain singularities in the deformed state, various errors may occur such as overflow or machine-dependent messages from trigonometric functions.

An avoidance is to use PARAM,K6ROT,1.0 to provide stiffness for the out-of-plane singularity. (1581)

Figure 11-8 General Limitation List Excerpt

The Current Error List describes errors in the program and suggests possible avoidances. An excerpt from the CEL is shown in **Figure 11-9**.



3041;67Material Nonlinearity - CLOAD - Wrong Answers - [17MAY91]

When applying loads other than thermal loads to superelements with the CLOAD input in SOL 66, wrong answers will occur. The load effects on the residual will not be factored correctly.

Avoidance: In addition to the CLOAD request in the residual subcase, a dummy load request must be included which references LSEQ Bulk Data input which in turn references the same DAREA input as the upstream loads. There must also be LSEQ and dummy load requests included for unloaded superelements if data recovery on the unloaded superelement is desired. This required procedure is demonstrated under Error 2549 and in an application note dated June 1990. The avoidance will no longer be necessary in Version 67.

3042Documentation - DBMGR, HSEA - UFM 4566

Section 5.2 of the *MSC.Nastran Handbook for Superelement Analysis* indicates the procedure for setting up multiple databases. In the examples, the DBMGR step that defines the second database is ALTERed after statement 1. This alteration may lead to UFM 4566, indicating that a data block cannot be found on the database. It is better practice to place this step immediately following the DBMGR step in the solution sequence that defines DB01. Depending on the solution sequence, this placement will result in the alter statement being placed at statement 7 or 8.

Figure 11-9 Current Error List Excerpt

The Current Error List also includes a cross-reference index and a keyword index with critical errors highlighted in bold type. MSC Software defines a critical error as one that usually results in a wrong answer and/or a significant waste of resources. Examples of the keyword index and cross-reference index are shown in **Figure 11-10** and **Figure 11-11**.

6.3.2.3 Keyword Index

<u>Keyword</u> <u>Error Numbers</u>

5411 FLOATING EXCEPTION 2262

ABNORMAL EXIT 3888
ACCE 1811,2206
ACCESS 2956,3615,3628,3799,3869
ACMODL 3874
ACOUSTICS 3565,3616,**3636**,**3649**,**3672**,**3831**
ADD5 3808,3900
ADG **2784**
AERO 2503
AEROELASTIC ANALYSIS **2784**

Figure 11-10 Excerpt from CEL Keyword Index

6.3.2.3 Cross-Reference Index, Version 67

Note: The error numbers appearing in **boldface** type are the critical errors.

Acoustic Cavity Analysis

(0),3565, 3616, 3617, **3649**, **3672**, 3752, 3757, **3831**, **3843**, 3874, 3882, 3889, **3916**, 4151, **4262**, **4317**

Aeroelastic Analysis

(7),2009, 2409, 2437, 2501, 2503, **2784**, **2949**, 2997, 3008, 3028, 3049, **3072**, **3107**, 3254, 3256, **3266**, **3275**, 3296, **3317**, **3363**, 3366, 3377, **3384**, **3437**, **3443**, 3540, 3609, **3658**, **3659**, **3660**, 3684, 3739, 3792, 3793, 3850, 3890, 3917, **4037**, **4251**

Case Control

(0),2414, 2645, 3079, **3188**, **3215**, 3544, **3573**, 3576, **3779**, 3829, 3918

Note that the number in parentheses is the number of general limitations associated with that category.

Figure 11-11 Excerpt from CEL Cross-Reference Index

11.5 The Comment ($) Entry

The input (.DAT) file listing of a typical commercial structure can be dozens or hundreds of pages long. Comment entries can be used to help organize your input data. Comment entries begin with a dollar ($) sign in column one followed by any characters out to column 80. Comment entries are ignored by MSC.Nastran and may appear anywhere within the input file. Comments appear only in the unsorted echo of the input file listing. Generous use of comments is highly recommended.

An example of typical comment entries in a Bulk Data Section is shown in **Listing 11-1**.

Listing 11-1 Example of the Comment Entry

```
CQUAD4,11,101,13,18,19,14
CQUAD4,12,101,14,19,20,15
$
$     DEFINE PRESSURE LOAD ON PLATES
PLOAD2,5,0.25,1,THRU,12
$     DEFINE PROPERTIES OF PLATE ELEMENTS
PSHELL,101,105,.05,105,,105
MAT1,105,30.E6,,0.3
$
$     DEFINE FIXED EDGE
SPC1,100,123456,16,THRU,20
$
$     DEFINE HINGED EDGES
SPC1,100,1234,1,6,11,5,10,15
$
$     CONSTRAIN OUT-OF-PLANE ROTATION FOR ALL GRIDS
SPC1,100,6,1,THRU,20
ENDDATA
```

CHAPTER
12

Performing an MSC.Nastran Analysis Step-by-Step

12.1 Defining the Problem

In this chapter, we perform a complete MSC.Nastran analysis step-by-step. Consider the hinged steel beam shown in **Figure 12-1**. It has a rectangular cross section and is subjected to a 100 lb concentrated force. Determine the deflection and stresses in the beam at the point of application of the load, with and without the effects of transverse shear.

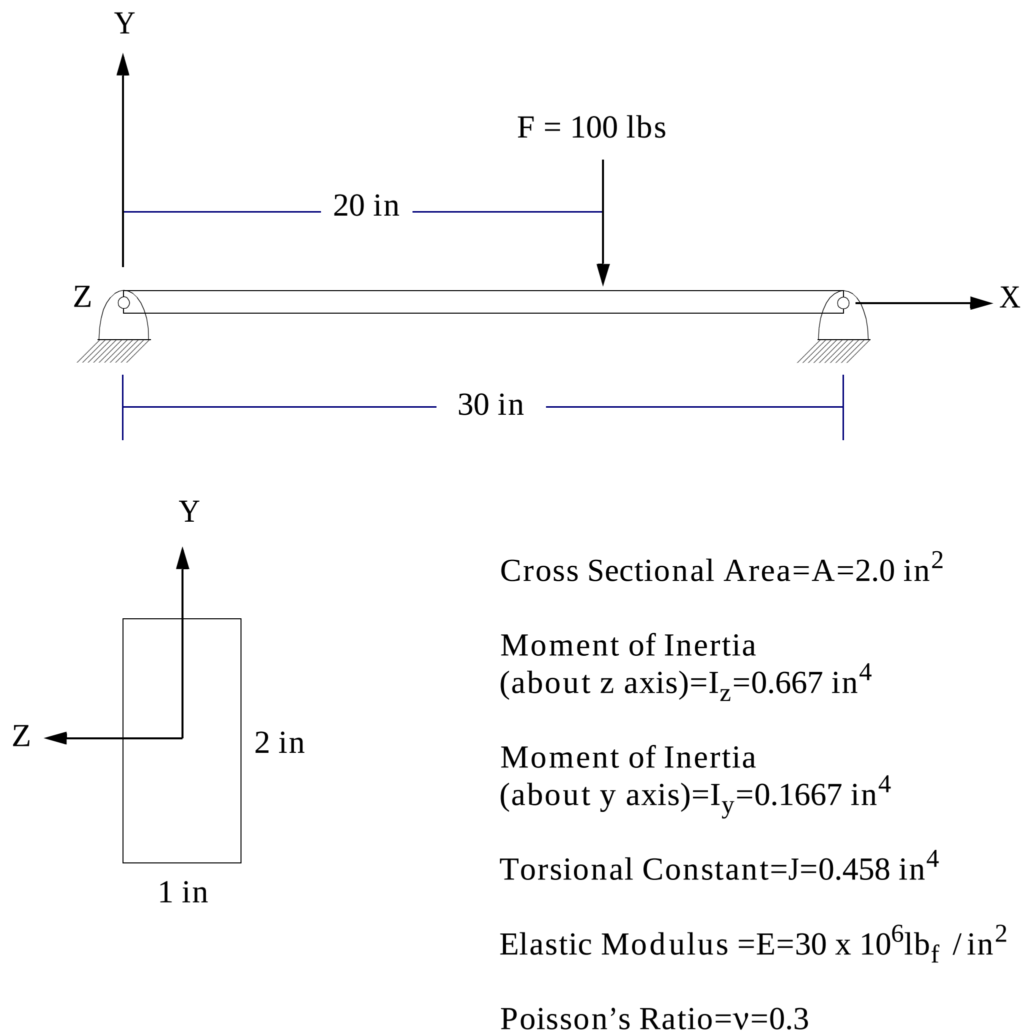


Figure 12-1 Beam Geometry and Load

12.2 Specifying the Type of Analysis

The type of analysis to be performed is specified in the Executive Control Section of the input file using the SOL (SOLution) statement. In this problem, we choose Solution 101, which is the linear static analysis solution sequence. The statement required is:

```
SOL 101
```

We will also identify the job with an ID statement and set the CPU time limit with a TIME statement as follows:

```
ID MPM,CH 12 EXAMPLE
TIME 100
```

The end of the Executive Control Section is indicated by the CEND delimiter. Thus, the complete Executive Control Section is written as follows:

```
ID MPM,CH 12 EXAMPLE
SOL 101
TIME 100
CEND
```

Note: Recall from "**Specifying the Type of Analysis**" on page 7 that both the TIME and ID statements are optional. The default value of TIME, however, is too small for all but the most trivial problems.

Also, recall that the format of the ID entry (ID i1,i2) must be adheared to or a fatal error will result. See "**Executive Control Section**" on page 8 for further information.

12.3 Designing the Model

The structure is a classical hinged slender beam subjected to bending behavior from a concentrated load. The CROD element will not work since it supports only extension and torsion. The CBEAM element would work, but its special capabilities are not required for this problem and its property entry is more difficult to work with. Thus, the CBAR element is a good choice. The number of elements to use is always a crucial decision; in our case the simplicity of the structure and its expected behavior allows the use of very few elements. We will choose three CBAR elements and four evenly spaced grid points as shown in **Figure 12-2**.

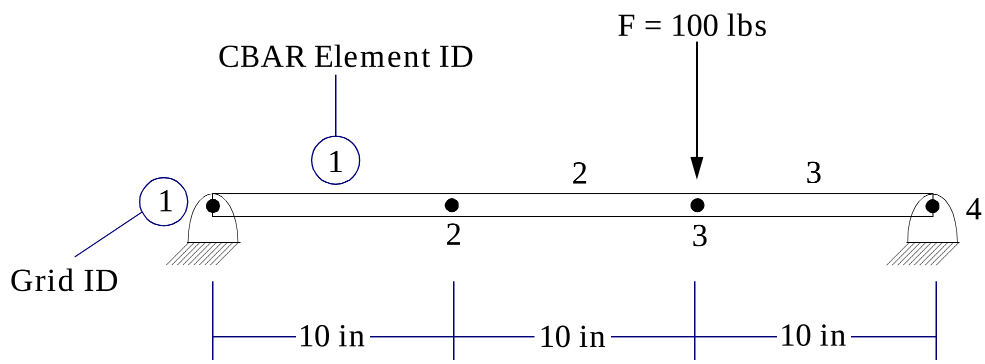


Figure 12-2 The Finite Element Model

Note that GRID points were located at the point of application of the load and at each reaction point.

12.4 Creating the Model Geometry

Coordinate System

Recall that MSC.Nastran has a default rectangular coordinate system called the basic system. Therefore, no special effort is required to orient our model. We will choose to define the model's coordinate system as shown in **Figure 12-3**. The beam's element x-axis will be parallel to the basic system's x-axis by our choice of X1, X2, and X3 (x, y, and z) on the GRID entries.

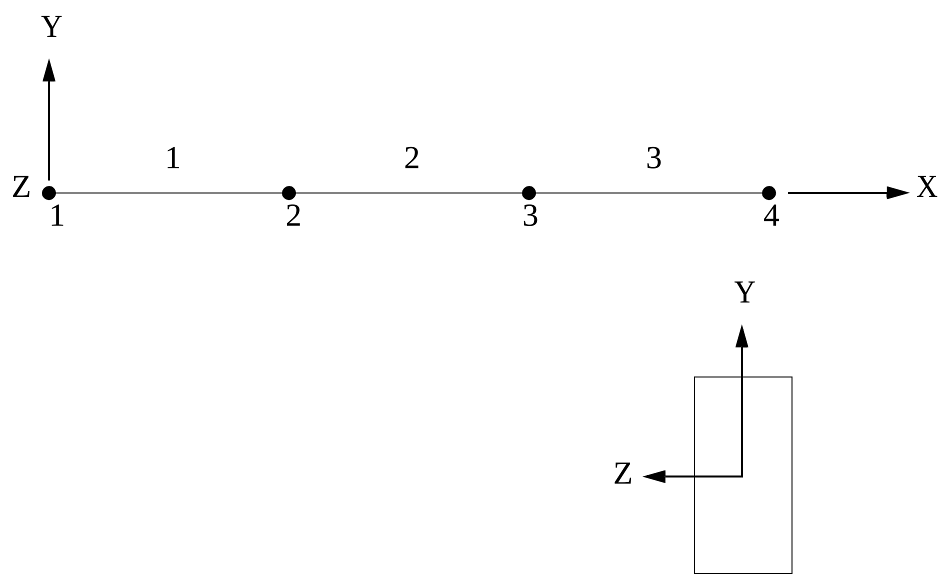


Figure 12-3 Model Coordinate System

GRID Points

GRID points are defined in the Bulk Data Section of the input file. Recall that the format of the GRID entry is:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Field	**Contents**
ID	Grid point identification number. (0 < Integer < 1000000)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ –1 or blank)

Field	Contents
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank)
SEID	Superelement identification number. (Integer ≥ 0; Default = 0)

The default basic coordinate system is defined by leaving field 3 (CP) blank (the basic coordinate system's ID number is zero).

The values of X1, X2, and X3 (in our rectangular system these mean x, y, and z) in fields 4, 5, and 6 are as follows:

GRID	X	Y	Z
1	0.	0.	0.
2	10.0	0.	0.
3	20.0	0.	0.
4	30.0	0.	0.

Field 7 (CD) is left blank since we want grid point displacements and constraints to be defined in the basic coordinate system. The constraints for this problem could be defined on field 8 (PS) of grid points 1 and 4. Instead, we will use SPC1 entries and leave field 8 blank.

Finally, field 9 is left blank since superelements are not part of this problem.

The completed GRID entries are written as follows:

1	2	3	4	5	6	7	8	9	10
GRID	1		0.	0.	0.				
GRID	2		10.0	0.	0.				
GRID	3		20.0	0.	0.				
GRID	4		30.0	0.	0.				

Or, in free field format, the GRID entries are written

```
GRID,1,,0.,0.,0.
GRID,2,,10.,0.,0.
GRID,3,,20.,0.,0.
GRID,4,,30.,0.,0.
```

12.5 Defining the Finite Elements

The CBAR Entry

Elements are defined in the Bulk Data Section of the input file. The format of the CBAR simple beam element is as follows:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PBAR entry. (Integer > 0 or blank; Default is EID unless BAROR entry has nonzero entry in field 3)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; $GA \neq GB$)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. (Integer > 0)
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees of freedom of the bar. The degrees of freedom are defined in the element's coordinate system. The bar must have stiffness associated with the PA and PB degrees of freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0)
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$, respectively, in displacement coordinate systems at points GA and GB, respectively. (Real or blank)

The property identification number (PID) is arbitrarily chosen to be 101-this label points to a PBAR beam property entry. The same PID is used for each of the three CBAR elements.

GA and GB are entered for each beam element, starting with GA (end A) of CBAR element 1 at (0., 0., 0.). Recall that the direction of the X-element axis is defined as the direction from GA to GB.

The beam orientation vector $\vec{v}$, described by GA and the components X1, X2, and X3, is arbitrarily chosen by setting X1 = 0.0, X2 = 1.0, and X3 = 0.0. Orientation vector $\vec{v}$ is shown in **Figure 12-4**.

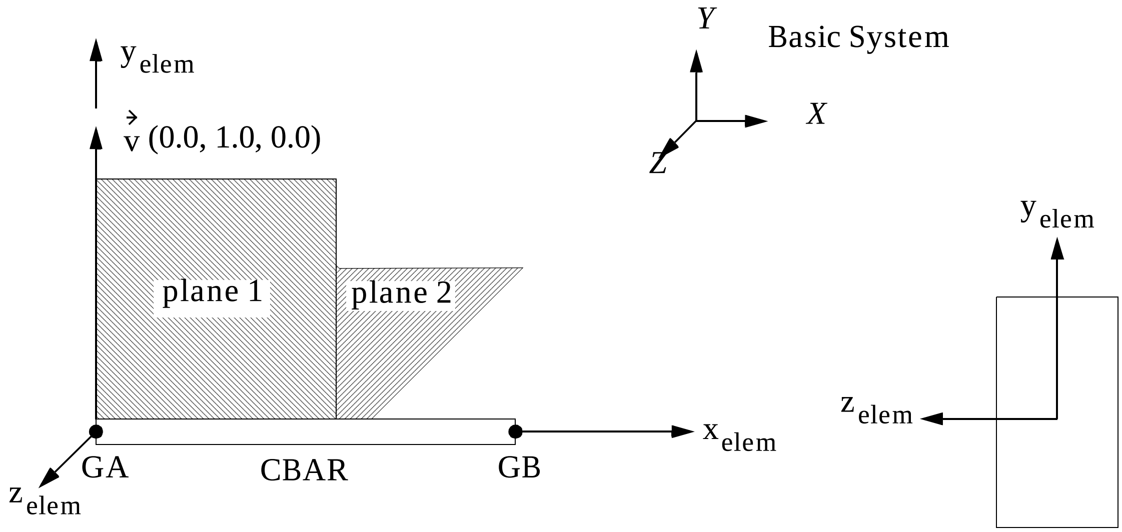


Figure 12-4 $\vec{v}$ and x_{elem} Defines Plane 1 and the y_{elem} Axis

Plane 1 is thus formed by $\vec{v}$ and the x-element axis. The y-element axis (y_{elem}) is perpendicular to the x-element axis and lies in plane 1.

Plane 2 is perpendicular to plane 1, and the z-element axis (z_{elem}) is formed by the cross product of the x-element and y-element axes.

The completed CBAR entries are written as follows:

1	2	3	4	5	6	7	8	9	10
CBAR	1	101	1	2	0.	1.	0.		
CBAR	2	101	2	3	0.	1.	0.		
CBAR	3	101	3	4	0.	1.	0.		

Or, in free field format, the CBAR entries appear as:

```
CBAR,1,101,1,2,0.,1.,0.
CBAR,2,101,2,3,0.,1.,0.
CBAR,3,101,3,4,0.,1.,0.
```

Continuations of the CBAR entries are not required since pin flags and offset vectors are not used in this model.

The PBAR Entry

The format of the PBAR entry is as follows:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		

1	2	3	4	5	6	7	8	9	10
	C1	C2	D1	D2	El	E2	F1	F2	
	K1	K2	I12						

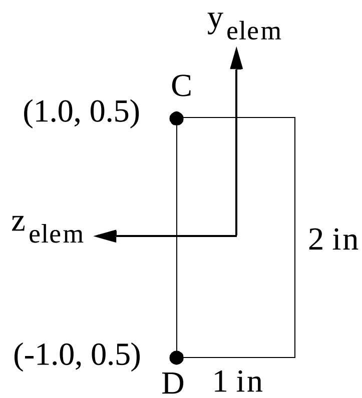
Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of bar cross section. (Real)
I1, I2, I12	Area moments of inertia. (Real; $I1 \geq 0.0$, $I2 \geq 0.0$, $I1 \cdot I2 > I12^2$)
J	Torsional constant. (Real)
NSM	Nonstructural mass per unit length. (Real)
K1, K2	Area factor for shear. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)

For our model, the property ID (PID) is 101, as called out on the CBAR entry. The material ID (MID) is arbitrarily chosen to be 201-this label points to a MAT1 entry. The beam's cross sectional area A is entered in field 4, and the torsional constant J is entered in field 7. The beam has no nonstructural mass (NSM), so column 8 is left blank.

Now for the tricky part; specifying I1 and I2 in fields 5 and 6. Recall that the choice of orientation vector $\vec{v}$ is arbitrary. What is not arbitrary is getting each value of I to match its correct plane. I1 is the moment of inertia for bending in plane 1 (which is the same as bending about the z axis, as it was probably called in your strength of materials class). Similarly, I2 is the moment of inertia for bending in plane 2 (about the y axis). Thus, $I1 = I_z = 0.667\text{ in}^4$, and $I2 = I_y = 0.1667\text{ in}^4$.

As a check for this model, think of plane 1 in this problem as the "stiff plane" (larger value of I) and plane 2 as the "not-as-stiff" plane (smaller value of I). This visualization should help you keep the bookkeeping straight.

Stress recovery coefficients are user-selected coordinates located on the bar's element y-z plane at which stresses are calculated by MSC.Nastran. We will choose the following two points (there is no requirement that all four available points must be used):



Finally, the problem statement requires that we investigate the effect of shear deflection. To add shear deflection to the bar, we include appropriate values of K1 and K2 on the second continuation of the PBAR entry. For a rectangular cross section, K1 = K2 = 5/6.

Leaving K1 and K2 blank results in default values of infinity (i.e., transverse shear flexibility is set equal to zero). This means that no deflection due to shear will occur.

The completed PBAR entry is written as follows (no shear deflection):

1	2	3	4	5	6	7	8	9	10
PBAR	101	201	2.	.667	.1667	.458			
	1.	.5	-1.	.5					

To add shear deflection, a second continuation is added:

PBAR	101	201	2.	.667	.1667	.458			
	1.	.5	-1.	.5					
	.8333	.8333							

In free field format, the PBAR entry is written as follows:

```
PBAR,101,201,2.,.667,.1667,.458
    ,1.,.5,-1.,.5
    ,.8333,.8333
```

12.6 Representing Boundary Conditions

The beam is hinged, so we must constrain GRID points 1 and 4 to represent this behavior. We will use one SPC1 Bulk Data entry for both grid points since the constraints at each end are the same.

The format of the SPC1 entry is as follows:

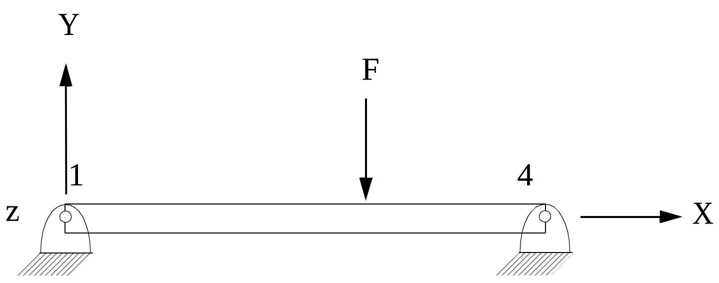
1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Field	Contents
SID	Identification number of single-point constraint set. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; for “THRU” option, G1 < G2. MSC.Nastran allows missing grid points in the sequence G1 through G2)

An SPC set identification number (SID) of 100 is arbitrarily chosen and entered in field 2. To select the SPC, the following Case Control command must be added to the Case Control Section:

```
SPC=100
```

Recall that constraints are applied in the GRID point’s displacement coordinate system-in our problem this is the basic coordinate system. The required components of constraint are shown below:



Grids 1 and 4 cannot translate in the x, y, or z directions (constrain DOFs 1, 2, and 3).
Grids 1 and 4 cannot rotate about the x-axis or y-axis (constrain DOFs 4 and 5).
Grids 1 and 4 can rotate about the z-axis (leave DOF 6 unconstrained).

Therefore, the required SPC1 entry is written as follows:

SPC1	100	12345	1	4					

Or in free field format we enter:

```
SPC1,100,12345,1,4
```

12.7 Specifying Material Properties

The beam's material is steel, with an elastic modulus of 0×10^6 lb / in^2. Poisson's ratio is 0.3. The format of the MAT1 entry is shown below (we will not use the optional stress limit / margin of safety capability on the MAT1 continuation line).

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	

Field	Contents
MID	Material identification number. (Integer > 0)
E	Young's modulus. (Real ≥ 0.0 or blank)
G	Shear modulus. (Real ≥ 0.0 or blank)
NU	Poisson's ratio. ($-1.0 <$ Real ≤ 0.5 or blank)
RHO	Mass density. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. (Real; Default = 0.0 if A is specified)
GE	Structural element damping coefficient. (Real)

The material identification number called out on the PBAR entry is 201; this goes in field 2 of the MAT1 entry. Values for RHO, A, TREF, and GE are irrelevant to this problem and are therefore left blank. Thus, the MAT1 entry is written as follows:

MAT1	201	30.E6		.3					

In free field format,

```
MAT1,201,30.E6,,.3
```

12.8 Applying the Loads

The beam is subjected to a single concentrated force of 100 lb_f acting on GRID 3 in the negative Y direction. The FORCE Bulk Data entry is used to apply this load. Its format is described below:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one $Ni \neq 0.0$)

A load set identification number (SID) of 10 is arbitrarily chosen and entered in field 2 of the FORCE entry. To select the load set, the following Case Control command must be added to the Case Control Section:

```
LOAD=10
```

The FORCE entry is written as follows:

FORCE	10	3		-100	0.	1.	0.		

where (0., 1., 0.) is a unit vector in the positive Y direction of the displacement coordinate system.

In free field format, the entry is written as follows.

```
FORCE,10,3,,-100.,0.,1.,0.
```

12.9 Controlling the Analysis Output

The types of analysis quantities to be printed are specified in the Case Control Section. This problem requires displacements and element stresses, so the following commands are needed:

`DISP=ALL`	(prints all GRID point displacements)
`STRESS=ALL`	(prints all element stresses)

In order to help verify the model results, we will also ask for the following output quantities:

`FORCE=ALL`	(prints all element forces)
`SPCF=ALL`	(prints all forces of single point constraint; i.e., reaction forces)

The following command will yield both unsorted and sorted input file listings:

```
ECHO=BOTH
```

TITLE and SUBTITLE headings will appear on each page of the output, and are chosen as follows:

```
TITLE=HINGED BEAM
SUBTITLE=WITH CONCENTRATED FORCE
```

Finally, we select constraint and load sets as follows:

```
SPC=100
LOAD=10
```

The complete Case Control Section is shown below. The commands can be entered in any order after the CEND delimiter.

```
CEND
ECHO=BOTH
DISP=ALL
STRESS=ALL
FORCE=ALL
SPCF=ALL
SPC=100
LOAD=10
TITLE=HINGED BEAM
SUBTITLE=WITH CONCENTRATED FORCE
```

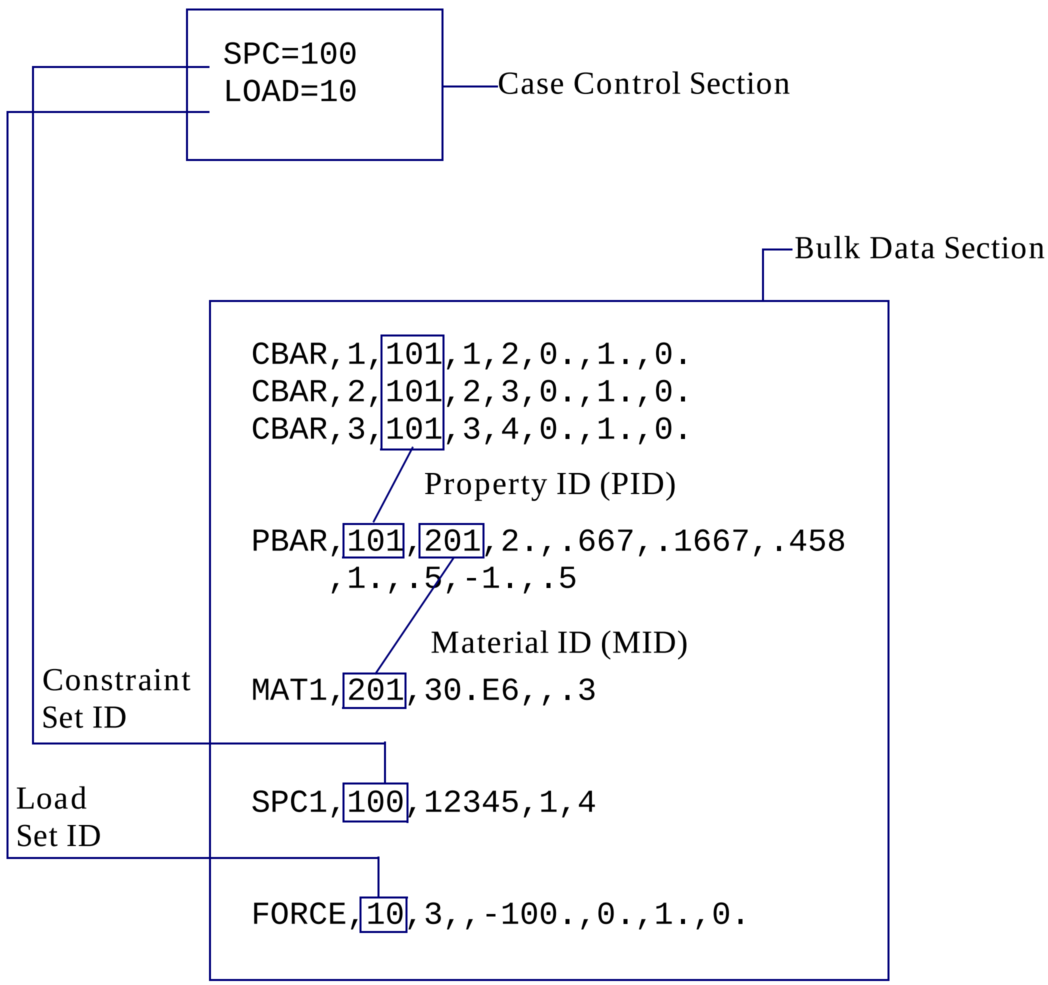
12.10 Completing the Input File and Running the Model

The completed input file (model without shear deflection) is called BASICEX1.DAT, and is shown in **Listing 12-1**.

Listing 12-1

```
ID MPM,EXAMPLE1
SOL 101
TIME 100
CEND
ECHO=BOTH
DISP=ALL
STRESS=ALL
FORCE=ALL
SPCF=ALL
SPC=100
LOAD=10
TITLE=HINGED BEAM
SUBTITLE=WITH CONCENTRATED FORCE
$
BEGIN BULK
$     DEFINE GRID POINTS
GRID,1,,0.,0.,0.
GRID,2,,10.,0.,0.
GRID,3,,20.,0.,0.
GRID,4,,30.,0.,0.
$
$     DEFINE CBAR ELEMENTS
CBAR,1,101,1,2,0.,1.,0.
CBAR,2,101,2,3,0.,1.,0.
CBAR,3,101,3,4,0.,1.,0.
$
$     DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES
PBAR,101,201,2.,.667,.1667,.458
    ,1.,.5,-1.,.5
$
$     DEFINE MATERIAL PROPERTIES
MAT1,201,30.E6,,.3
$
$     DEFINE  SPC CONSTRAINT SET
SPC1,100,12345,1,4
$
$     DEFINE CONCENTRATED FORCE
FORCE,10,3,,-100.,0.,1.,0.
$
ENDDATA
```

It is useful at this point to review “what points to what” in the model. Set and property relationships are summarized in the diagram below:



The job is submitted to MSC.Nastran with a system command similar to the following:

```
NASTRAN BASICEX1 SCR=YES
```

The details of the command are unique to your system; contact your computer staff or refer to your *MSC.Nastran Configuration and Operations Guide* for more information.

12.11 MSC.Nastran Output

The results of an MSC.Nastran job are contained in the .f06 file. The complete .f06 file for this problem (no shear deflection) is shown in Listing 12-2. Blank pages are shown for completeness.

Listing 12-2 Complete .f06 Results File

```
                     THIS PROGRAM IS CONFIDENTIAL AND A TRADE SECRET OF MSC.SOFTWARE CORP.  THE RECEIPT OR
                    POSSESSION OF THIS PROGRAM DOES NOT CONVEY ANY RIGHTS TO REPRODUCE OR DISCLOSE ITS CONTENTS, OR TO
                   MANUFACTURE, USE, OR SELL ANYTHING HEREIN, IN WHOLE OR IN PART, WITHOUT THE SPECIFIC WRITTEN CONSENT OF
MSC.SOFTWARE CORPORATION.

* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
* *                                   * *
* *           MSC.SOFTWARE            * *
* *               CORP                * *
* *                                   * *
* *      M S C . N a s t r a n        * *
* *                                   * *
* *        VERSION - 2001.0.1         * *
* *                                   * *
* *           SEP 10, 2001            * *
* *                                   * *
* *                                   * *
* *              Intel                * *
* *                                   * *
* *MODEL PentiumIII/995 (MERCED.scm)* *
* *                                   * *
* *  Windows 2000 5.0 (Build 2195)    * *
* *                                   * *
* *                                   * *
* *                                   * *
* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
```

MSC.Nastran version number

Installation date

Your computer and operating system

```
                                                                               SEPTEMBER  10, 2001  MSC.NASTRAN    9/10/01    PAGE     1

        N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O
     ID MPM,CH 12 EXAMPLE
     SOL 101
     TIME 100
     CEND
```

```
  HINGED BEAM                                                                   SEPTEMBER  10, 2001  MSC.NASTRAN    9/10/01    PAGE     2
     WITH CONCENTRATED FORCE
                                           C A S E    C O N T R O L    D E C K    E C H O
                 CARD
                 COUNT

                  1        ECHO=BOTH
                  2        DISP=ALL
                  3        STRESS=ALL
                  4        FORCE=ALL
                  5        SPCF=ALL
                  6        SPC=100
                  7        LOAD=10
                  8        TITLE=HINGED BEAM
                  9        SUBTITLE=WITH CONCENTRATED FORCE
                 10        $
                 11        BEGIN BULK
```

Listing 12-2 Complete .f06 Results File (continued)

```
 HINGED BEAM                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3
    WITH CONCENTRATED FORCE
                                            I N P U T   B U L K   D A T A   D E C K   E C H O
                                  .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                                  $     DEFINE GRID POINTS
                                  GRID,1,,0.,0.,0.
                                  GRID,2,,10.,0.,0.
                                  GRID,3,,20.,0.,0.
                                  GRID,4,,30.,0.,0.
                                  $
                                  $     DEFINE CBAR ELEMENTS
                                  CBAR,1,101,1,2,0.,1.,0.
                                  CBAR,2,101,2,3,0.,1.,0.
                                  CBAR,3,101,3,4,0.,1.,0.
                                  $
                                  $     DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES
                                  PBAR,101,201,2.,.667,.1667,.458,,,+PB1
                                  +PB1,1.,.5,-1.,.5
                                  $
                                  $     DEFINE MATERIAL PROPERTIES
                                  MAT1,201,30.E6,,.3
                                  $
                                  $     DEFINE SPC CONSTRAINT SET
                                  SPC1,100,12345,1,4
                                  $
                                  $     DEFINE CONCENTRATED FORCE
                                  FORCE,10,3,,-100.,0.,1.,0.
                                  $
                                  ENDDATA
                                  INPUT BULK DATA CARD COUNT =      25
```

```
 HINGED BEAM                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4
    WITH CONCENTRATED FORCE
                                               S O R T E D   B U L K   D A T A   E C H O
                 CARD
                 COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                    1-        CBAR    1       101     1       2       0.      1.      0.
                    2-        CBAR    2       101     2       3       0.      1.      0.
                    3-        CBAR    3       101     3       4       0.      1.      0.
                    4-        FORCE   10      3               -100.   0.      1.      0.
                    5-        GRID    1               0.      0.      0.
                    6-        GRID    2               10.     0.      0.
                    7-        GRID    3               20.     0.      0.
                    8-        GRID    4               30.     0.      0.
                    9-        MAT1    201     30.E6           .3
                   10-        PBAR    101     201     2.      .667    .1667   .458                    +PB1
                   11-        +PB1    1.      .5      -1.     .5
                   12-        SPC1    100     12345   1       4
                              ENDDATA
                        TOTAL COUNT=        13
```

```
 HINGED BEAM                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     5
    WITH CONCENTRATED FORCE
 USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
                                                    OLOAD    RESULTANT
                T1             T2             T3             R1             R2             R3
       1   0.0000000E+00 -1.0000000E+02  0.0000000E+00  0.0000000E+00  0.0000000E+00 -2.0000000E+03
```

```
 HINGED BEAM                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     6
    WITH CONCENTRATED FORCE
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

   LOAD SEQ. NO.             EPSILON              EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
                1        -4.0856207E-17         1.1105558E-01
```

Listing 12-2 Complete .f06 Results File (continued)

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     7
     WITH CONCENTRATED FORCE
  USER INFORMATION MESSAGE
 ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
  RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.

                                                 SPCFORCE RESULTANT

                T1             T2             T3             R1             R2             R3
        1   0.0000000E+00  1.0000000E+02  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.0000000E+03
```

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     8
     WITH CONCENTRATED FORCE
                                         D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0            0.0            0.0            0.0            0.0           -2.221112E-04
             2      G      0.0           -1.943473E-03   0.0            0.0            0.0           -1.388195E-04
             3      G      0.0           -2.221112E-03   0.0            0.0            0.0            1.110556E-04
             4      G      0.0            0.0            0.0            0.0            0.0            2.776390E-04
```

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     9
     WITH CONCENTRATED FORCE
                           F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G      0.0            3.333333E+01   0.0            0.0            0.0            0.0
             4      G      0.0            6.666666E+01   0.0            0.0            0.0            0.0
```

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    10
     WITH CONCENTRATED FORCE
                             F O R C E S   I N   B A R   E L E M E N T S         ( C B A R )

     ELEMENT         BEND-MOMENT END-A            BEND-MOMENT END-B                - SHEAR -                AXIAL
       ID.         PLANE 1       PLANE 2        PLANE 1       PLANE 2        PLANE 1       PLANE 2        FORCE          TORQUE
            1     0.0           0.0            3.333333E+02  0.0           -3.333333E+01  0.0            0.0            0.0
            2     3.333333E+02  0.0            6.666667E+02  0.0           -3.333333E+01  0.0            0.0            0.0
            3     6.666667E+02  0.0            2.034505E-05  0.0            6.666666E+01  0.0            0.0            0.0
```

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    11
     WITH CONCENTRATED FORCE
```

```
  HINGED BEAM                                                SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    12
     WITH CONCENTRATED FORCE
                             S T R E S S E S   I N   B A R   E L E M E N T S          ( C B A R )

  ELEMENT        SA1            SA2            SA3            SA4           AXIAL          SA-MAX         SA-MIN     M.S.-T
    ID.          SB1            SB2            SB3            SB4           STRESS         SB-MAX         SB-MIN     M.S.-C

        1    0.0            0.0            0.0            0.0            0.0            0.0            0.0
            -4.997502E+02   4.997502E+02   0.0            0.0                           4.997502E+02  -4.997502E+02

        2   -4.997502E+02   4.997502E+02   0.0            0.0            0.0            4.997502E+02  -4.997502E+02
            -9.995003E+02   9.995003E+02   0.0            0.0                           9.995003E+02  -9.995003E+02

        3   -9.995003E+02   9.995003E+02   0.0            0.0            0.0            9.995003E+02  -9.995003E+02
            -3.050233E-05   3.050233E-05   0.0            0.0                           3.050233E-05  -3.050233E-05
```

Listing 12-2 Complete .f06 Results File (continued)

```
 HINGED BEAM                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    13
    WITH CONCENTRATED FORCE

 HINGED BEAM                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    14
    WITH CONCENTRATED FORCE

* * * *  D B D I C T   P R I N T  * * * *      SUBDMAP = PRTSUM  ,  DMAP STATEMENT NO.     13

                                   * * * *  A N A L Y S I S  S U M M A R Y  T A B L E  * * * *

 SEID  PEID PROJ VERS APRCH      SEMG SEMR SEKR SELG SELR MODES DYNRED SOLLIN PVALID SOLNL LOOPID DESIGN CYCLE SENSITIVITY
-------------------------------------------------------------------------------------------------------------------------
    0     0    1    1 '        '    T    T    T    T    T     F      F      T      0     F     -1           0           F

SEID = SUPERELEMENT ID.
PEID = PRIMARY SUPERELEMENT ID OF IMAGE SUPERELEMENT.
PROJ = PROJECT ID NUMBER.
VERS = VERSION ID.
APRCH = BLANK FOR STRUCTURAL ANALYSIS.  HEAT FOR HEAT TRANSFER ANALYSIS.
SEMG = STIFFNESS AND MASS MATRIX GENERATION STEP.
SEMR = MASS MATRIX REDUCTION STEP (INCLUDES EIGENVALUE SOLUTION FOR MODES).
SEKR = STIFFNESS MATRIX REDUCTION STEP.
SELG = LOAD MATRIX GENERATION STEP.
SELR = LOAD MATRIX REDUCTION STEP.
MODES = T (TRUE) IF NORMAL MODES OR BUCKLING MODES CALCULATED.
DYNRED = T (TRUE) MEANS GENERALIZED DYNAMIC AND/OR COMPONENT MODE REDUCTION PERFORMED.
SOLLIN = T (TRUE) IF LINEAR SOLUTION EXISTS IN DATABASE.
PVALID = P-DISTRIBUTION ID OF P-VALUE FOR P-ELEMENTS
LOOPID = THE LAST LOOPID VALUE USED IN THE NONLINEAR ANALYSIS.  USEFUL FOR RESTARTS.
SOLNL = T (TRUE) IF NONLINEAR SOLUTION EXISTS IN DATABASE.
DESIGN CYCLE = THE LAST DESIGN CYCLE (ONLY VALID IN OPTIMIZATION).
SENSITIVITY = SENSITIVITY MATRIX GENERATION FLAG.


                                       * * * END OF JOB * * *
```

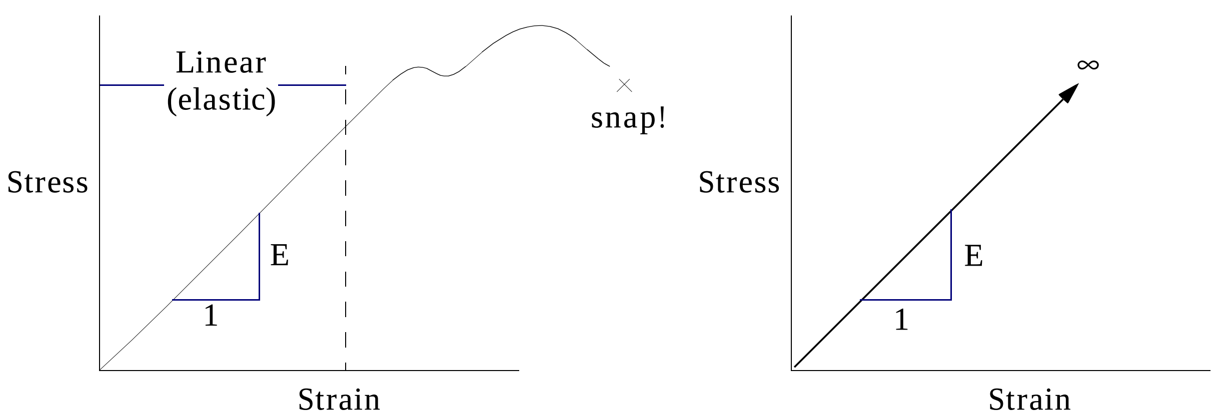
12.12 Reviewing the Results

You cannot simply move directly to the displacement and stress results and accept the answers. You are responsible for verifying the correctness of the model. Some common checks are described in this section.

Check for Error Messages, Epsilon, and Reasonable Displacements

No error or warning messages are present in the .f06 (results) file-this is certainly no guarantee of a correct run, but it's a good first step. Also, examine the value of epsilon on page 6 of the output. It is very small ($\sim 10^{-16}$), showing stable numerical behavior. Next, it is a good policy to check the displacement values, just to verify that they are not absurdly out of line with the physical problem or that a geometric nonlinear analysis is not required. For example, this beam displacing several inches might indicate that a load is orders of magnitude too high, or that a cross sectional property or an elastic modulus has been incorrectly specified. In our case, the lateral displacements (page 8 of the output) are on the order of 10^{-3} inches, which seems reasonable for this problem.

Aside. Suppose you *did* obtain displacements of several inches-or perhaps into the next city. Shouldn't MSC.Nastran give some sort of engineering sanity warning? The answer is no, because the program is doing precisely what it was told to do and has no ability to judge what a reasonable displacement is. Recall that our analysis is linear and that the MAT1 material property entry thinks that the elastic modulus E *is* the material curve. This distinction is shown in **Figure 12-5**.



(a) What We Think (b) What MSC.Nastran Uses (MAT1)

Figure 12-5 Reality versus Modeling

The MAT1 entry states that our material is always elastic and infinitely strong. In reality, we will violate restrictions on small displacements and material linearity given sufficient loading.

Once again, we remind ourselves that finite element analysis is a *tool*, that a complete understanding of its inherent assumptions and limitations is crucial to obtaining good results, and that treating the program as a black box reduces the practice of engineering to a matter of luck.

Check Reactions

To check static equilibrium, we calculate the reaction forces at the constraints and obtain 33.3 lbs. in the +y direction at grid point 1 and 66.6 lbs. in the +y direction at grid point 4 (**Figure 12-7**(a)). These values match the forces of single point constraint reported on page 9 of the output (T2 in this table means forces in the Y direction). Thus, the load and resulting reactions make sense.

Check Shear Along the Beam

The shear diagram for the beam is shown in **Figure 12-7**(b). Page 10 of the output lists the shear forces across each element as -33.3 lbs. for elements 1 and 2 and +66.6 lbs. for element 3.

Note that shear occurs only in plane 1 (the plane of the applied force). The sign convention for CBAR element internal shear forces in Plane 1 ($_{elem} - y_{elem}$plane) is shown in **Figure 12-6**:

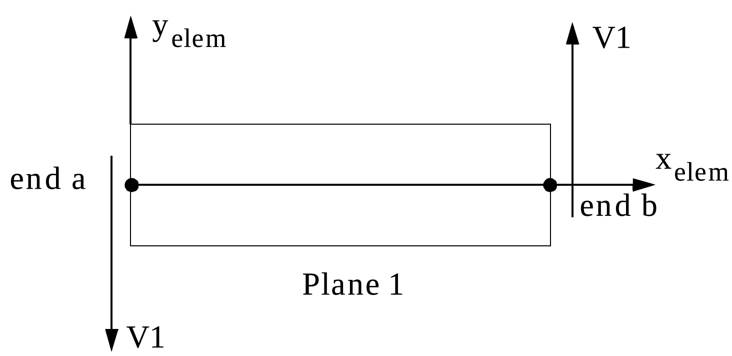


Figure 12-6 CBAR Element Shear Convention (Plane 1)

Thus, the signs make sense with respect to the applied load.

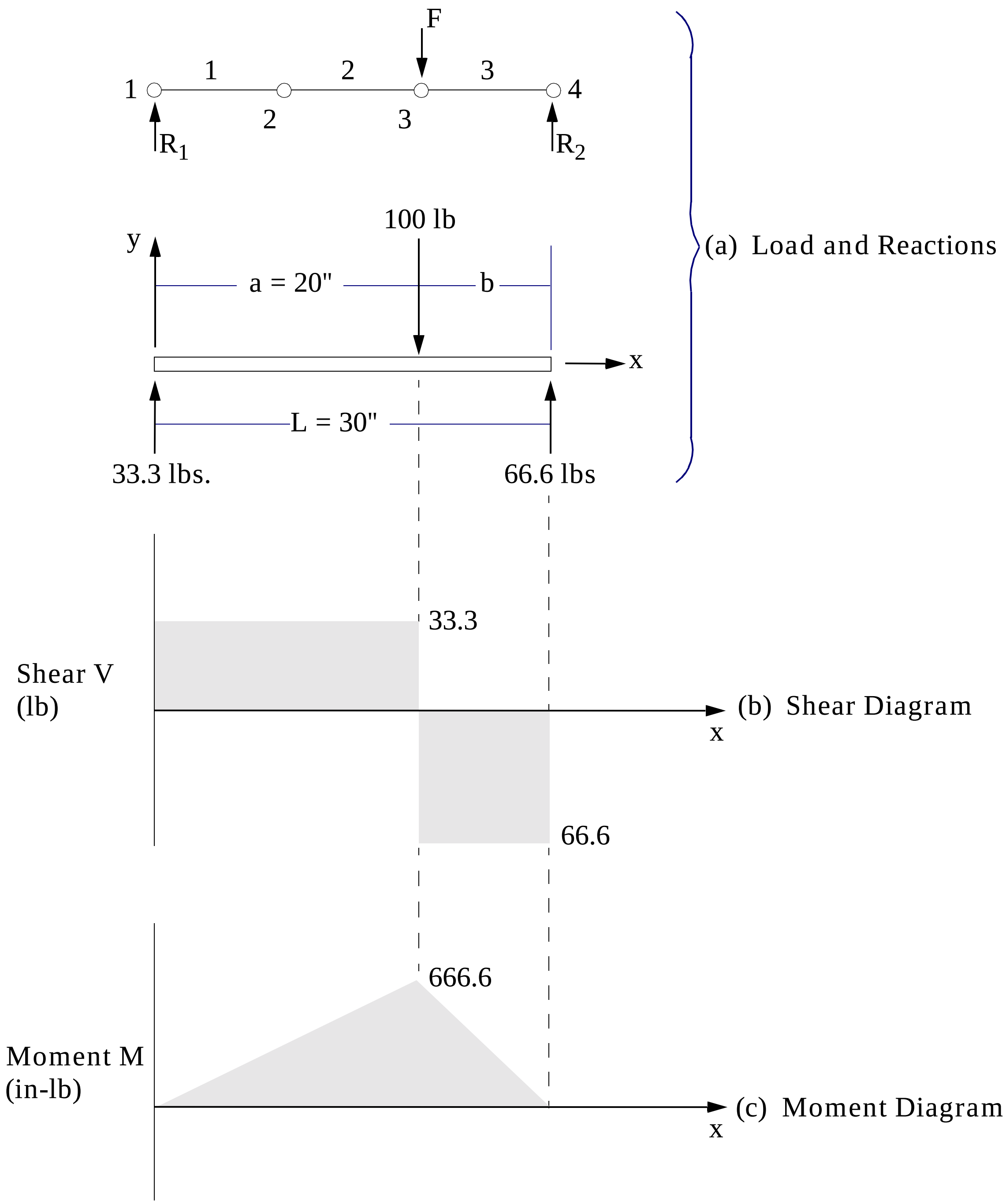


Figure 12-7 Beam Reaction Forces, Shear Diagram, and Moment Diagram

Displacement and Stress Results

The displacement at the point of application of the load (GRID 3) is shown on page 8 of the results:

$$u_y^3 = -2.221112\text{E-}3 \text{ inch}$$

The deflection is in the -y direction as expected.

The CBAR element stresses at the point of application of the load (GRID 3) are reported by end b of CBAR 2 and end a of CBAR 3. Positive stress values indicate tension and negative values indicate compression. The top of the beam is in compression and the bottom of the beam is in tension. Stress recovery point 1 is located on the top of the beam and point 2 is located at the bottom of the beam, as shown in **Figure 12-8**:

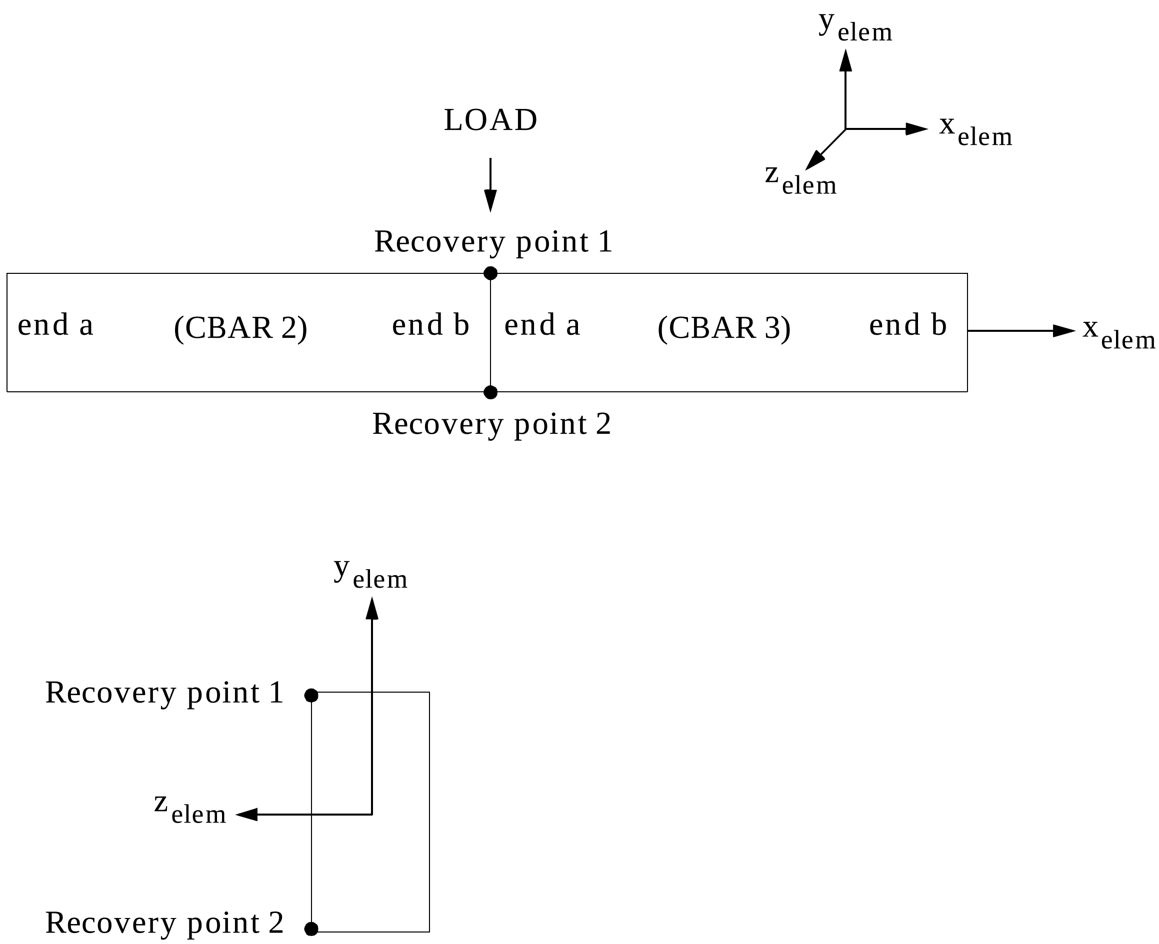


Figure 12-8 Bar Element Output Nomenclature

The MSC.Nastran CBAR element stress output (Figures 6-7) is interpreted as shown in **Figure 12-9**:

CBAR 2
End b, point 1
-999.5 lb/in^2

CBAR 2
End b, point 2
999.5 lb/in^2

```
HINGED BEAM
     WITH CONCENTRATED FORCE
                                  S T R E S S E S

ELEMENT      SA1          SA2          SA3
ID.          SB1          SB2          SB3

   1     0.0           0.0             0.0
        -4.997502E+02  4.997502E+02    0.0
   2    -4.997502E+02  4.997502E+02    0.0
        -9.995003E+02  9.995003E+02    0.0
   3    -9.995003E+02  9.995003E+02    0.0
        -3.050233E-05  3.050233E-05    0.0
```

CBAR 3
End a, point 1
-999.5 lb/in^2

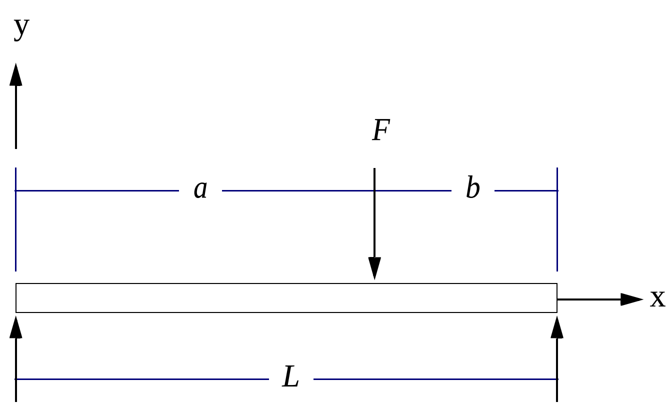
CBAR 3
End a, point 2
999.5 lb/in^2

Figure 12-9 Bar Element Stress Output

Therefore, the top surface of the beam (point 1) sees $999.5\ lb/in^2$(compression) and the bottom surface sees $99.5\ lb/in^2$(tension).

12.13 Comparing the Results with Theory

First, the deflection at the point of application of the load will be determined by hand. This calculation does not include shear effects, so it can be directly compared with the MSC.Nastran results shown in "**MSC.Nastran Output**" on page 24. The deflection due to bending only is calculated as follows:



$a = 20$ in
$b = 10$ in
$F = 100$ lb
$L = 30$ in

$$\text{For} \quad x = a, u_y = -\frac{Fa^2b^2}{3EIL} = -\frac{(100\ \text{lb})(20\ \text{in})^2(10\ \text{in})^2}{3(30 \times 10^6\ \text{lb/in}^2)(.667\ \text{in}^4)(30\text{in})}$$

$$u_y = -2.22111\text{E-3 inch}$$

This value is in exact agreement with the T2 value for GRID 3 on page 8 of the MSC.Nastran output.

The effect of shear deflection is determined by adding the second continuation of the PBAR entry and rerunning the job. The new Bulk Data Section is shown in **Figure 12-10**.

```
HINGED BEAM                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4
     WITH CONCENTRATED FORCE
                                                        S O R T E D   B U L K   D A T A   E C H O
            CARD
            COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                1-        CBAR    1       101     1       2       0.      1.      0.
                2-        CBAR    2       101     2       3       0.      1.      0.
                3-        CBAR    3       101     3       4       0.      1.      0.
                4-        FORCE   10      3               -100.   0.      1.      0.
                5-        GRID    1               0.      0.      0.
                6-        GRID    2               10.     0.      0.
                7-        GRID    3               20.     0.      0.
                8-        GRID    4               30.     0.      0.
                9-        MAT1    201     30.E6           .3
               10-        PBAR    101     201     2.      .667    .1667   .458                    +000001
               11-        ++0000011.      .5      -1.     .5                                      +000002
               12-        ++000002.8333   .8333
               13-        SPC1    100     12345   1       4
                          ENDDATA
                   TOTAL COUNT=    14
```

Shear Factor K:
K1 = K2 = 5/6 = .8333 for rectangular sections

Figure 12-10 Shear Factor K on PBAR Entry

The deflection results are given on page 8 of the output:

```
HINGED BEAM                                               SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    8
     WITH CONCENTRATED FORCE
                                                     D I S P L A C E M E N T   V E C T O R

POINT ID.    TYPE          T1             T2             T3             R1             R2             R3
       1       G      0.0            0.0            0.0            0.0            0.0           -2.221112E-04
       2       G      0.0           -1.960807E-03   0.0            0.0            0.0           -1.388195E-04
       3       G      0.0           -2.255780E-03   0.0            0.0            0.0            1.110556E-04
       4       G      0.0            0.0            0.0            0.0            0.0            2.776390E-04
```

Comparing deflection at GRID 3 with and without shear, we have:

$$u_y^3 \text{ (without shear)} = -2.221112\text{E-}3 \text{ inch}$$

$$u_y^3 \text{ (with shear)} = -2.255780\text{E-}3 \text{ inch}$$

Thus, adding shear to the model results in about 1.6% greater deflection of GRID 3.

The stresses on the top and bottom surfaces of the beam at the point of application of the load are given by

$$\sigma = \text{bending stress} = \pm\frac{Mc}{I}$$

where:

M = moment at GRID point 3

c = distance from neutral axis to outer fiber

I = bending moment of inertia in plane 1

From **Figure 12-7**(c), the moment at GRID 3 is 666.6 in-lb. Thus,

$$\sigma = \pm\frac{(666.6 \text{ in-lb})(1.0 \text{ in})}{(.667 \text{ in}^4)} = \pm 999.4 \text{ lb/in}^2$$

which is in agreement with the MSC.Nastran results.

CHAPTER

13 Example Problems

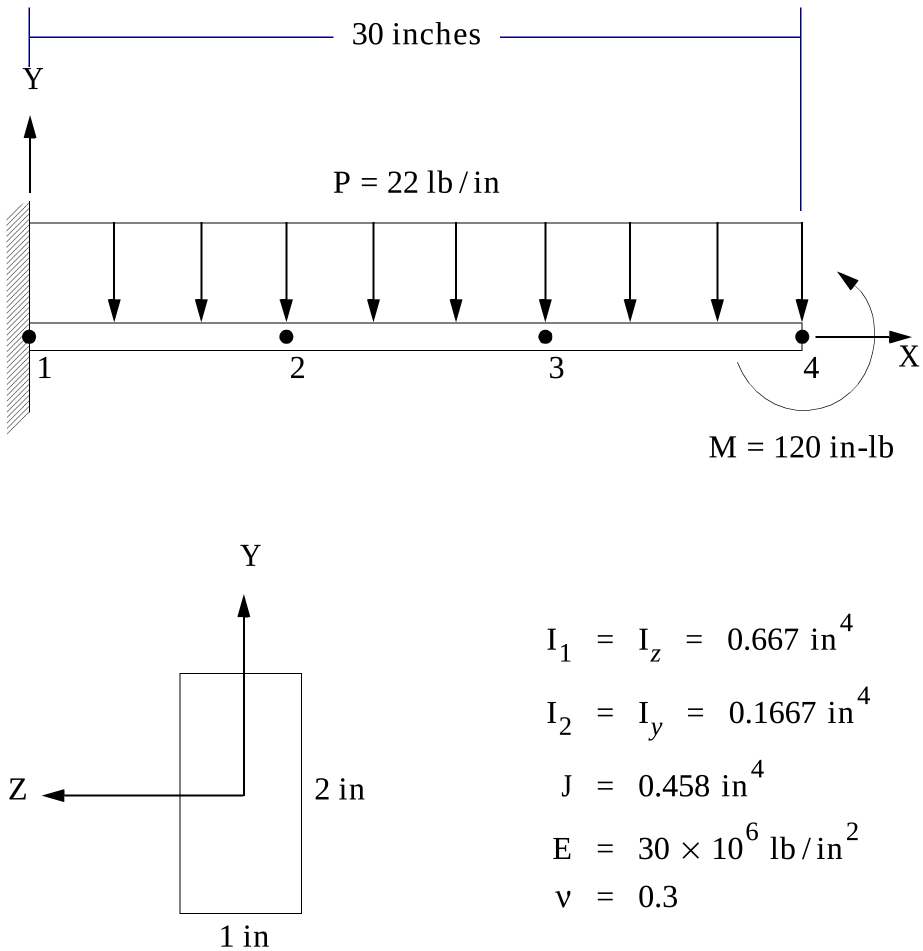
- Cantilever Beam with a Distributed Load and a Concentrated Moment
- Rectangular Plate (fixed-hinged-hinged-free) with a Uniform Lateral Pressure Load
- Gear Tooth with Solid Elements

13.1 Cantilever Beam with a Distributed Load and a Concentrated Moment

This problem uses the same beam as the problem in "**Performing an MSC.Nastran Analysis Step-by-Step**" on page 7 (i.e., the GRIDs, CBAR elements, and element properties are identical). The loads and constraints have been changed.

Problem Statement

Find the free end deflection of a rectangular cantilever beam subject to a uniform distributed load and a concentrated moment at the free end. The beam's geometry, properties, and loading are shown in **Figure 13-1**.



$$I_1 = I_z = 0.667 \text{ in}^4$$
$$I_2 = I_y = 0.1667 \text{ in}^4$$
$$J = 0.458 \text{ in}^4$$
$$E = 30 \times 10^6 \text{ lb / in}^2$$
$$\nu = 0.3$$

Figure 13-1 Beam Geometry, Properties, and Loads

The Finite Element Model

Applying the Loads

The uniform distributed load is applied to the three CBAR elements using a PLOAD1 entry. One PLOAD1 entry is required for each element. We have chosen fractional scaling, which means that the physical length of the element is normalized to a length of 1.0. Since the distributed load runs the entire length of each element, each PLOAD1 entry will be applied from 0.0 to 1.0. Since the load is uniform, P1 = P2 = 22.0 lb / in.

The concentrated end moment is applied using a MOMENT entry. The direction of the moment (by the right hand rule) is about the +z axis. Thus,

$$\vec{m} = M\vec{N}$$

where M is the magnitude of 120.0 in-lb, and $\vec{N}$ is the vector (0., 0., 1.).

The load set ID is 10, and the loads are selected in the Case Control Section with the Command LOAD = 10.

Applying the Constraints

Grid 1 is fixed in a wall, so all six DOFs (123456) are constrained to zero. This can be done directly on the GRID entry using Field 8 (PS-permanent single point constraints associated with the grid point). No other constraints are required in this model.

Output Requests

The Case Control Command DISP = ALL is required to report displacements. In addition, it is a good idea to look at constraint forces at the wall as part of checking out the model. Thus, we will add the Case Control Command SPCF = ALL.

The Input File

The complete input file is shown in **Listing 13-1**.

Listing 13-1

```
ID MPM,EXAMPLE2
SOL 101
TIME 100
CEND
ECHO=BOTH
DISP=ALL
SPCF=ALL
LOAD=10
TITLE=EXAMPLE 2
SUBTITLE=CANTILEVER BEAM
LABEL=DISTRIBUTED LOAD AND END MOMENT
$
BEGIN BULK
$      DEFINE GRID POINTS
GRID,1,,0.,0.,0.,,123456
GRID,2,,10.,0.,0.
GRID,3,,20.,0.,0.
GRID,4,,30.,0.,0.
$
$      DEFINE CBAR ELEMENTS
CBAR,1,101,1,2,0.,1.,0.
CBAR,2,101,2,3,0.,1.,0.
CBAR,3,101,3,4,0.,1.,0.
$
$      DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES
PBAR,101,201,2.,.667,.1667,.458,,,+PB1
+PB1,1.,.5
$
```

```
$     DEFINE MATERIAL PROPERTIES
MAT1,201,30.E6,,.3
$
$     DEFINE UNIFORM DISTRIBUTED LOAD
PLOAD1,10,1,FY,FR,0.,-22.,1.,-22.
PLOAD1,10,2,FY,FR,0.,-22.,1.,-22.
PLOAD1,10,3,FY,FR,0.,-22.,1.,-22.
$
$     DEFINE CONCENTRATED MOMENT AT FREE END
MOMENT,10,4,,120.,0.,0.,1.
ENDDATA
```

MSC.Nastran Results

The .f06 Results File

The MSC.Nastran results are shown in **Listing 13-2**.

Listing 13-2

```
                   THIS PROGRAM IS CONFIDENTIAL AND A TRADE SECRET OF MSC.SOFTWARE CORP.  THE RECEIPT OR
                  POSSESSION OF THIS PROGRAM DOES NOT CONVEY ANY RIGHTS TO REPRODUCE OR DISCLOSE ITS CONTENTS, OR TO
                 MANUFACTURE, USE, OR SELL ANYTHING HEREIN, IN WHOLE OR IN PART, WITHOUT THE SPECIFIC WRITTEN CONSENT
         OF MSC.SOFTWARE CORPORATION.


* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
* *                                  * *
* *           MSC.SOFTWARE           * *
* *               CORP               * *
* *                                  * *
* *      M S C . N a s t r a n       * *
* *                                  * *
* *       VERSION - 2001.0.1         * *
* *                                  * *
* *          SEP 10, 2001            * *
* *                                  * *
* *                                  * *
* *              Intel               * *
* *                                  * *
* *MODEL PentiumIII/995 (MERCED.scm)* *
* *                                  * *
* *  Windows 2000 5.0 (Build 2195)   * *
* *                                  * *
* *                                  * *
* *                                  * *
* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *

                                                                         SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     1

         N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O



     ID MPM,EXAMPLE2
     SOL 101
     TIME 100
     CEND
```

Listing 13-2 (continued)

```
 EXAMPLE 2                                                          SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     2
  CANTILEVER BEAM
  DISTRIBUTED LOAD AND END MOMENT
                                      C A S E    C O N T R O L   D E C K   E C H O
              CARD
              COUNT
               1      ECHO=BOTH
               2      DISP=ALL
               3      SPCF=ALL
               4      LOAD=10
               5      TITLE=EXAMPLE 2
               6      SUBTITLE=CANTILEVER BEAM
               7      LABEL=DISTRIBUTED LOAD AND END MOMENT
               8      $
               9      BEGIN BULK
```

```
 EXAMPLE 2                                                          SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3
  CANTILEVER BEAM
  DISTRIBUTED LOAD AND END MOMENT
                                           I N P U T   B U L K   D A T A   D E C K   E C H O

                           .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                            $     DEFINE GRID POINTS
                            GRID,1,,0.,0.,0.,,123456
                            GRID,2,,10.,0.,0.
                            GRID,3,,20.,0.,0.
                               GRID,4,,30.,0.,0.
                            $
                            $     DEFINE CBAR ELEMENTS
                            CBAR,1,101,1,2,0.,1.,0.
                            CBAR,2,101,2,3,0.,1.,0.
                            CBAR,3,101,3,4,0.,1.,0.
                            $
                            $     DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES
                            PBAR,101,201,2.,.667,.1667,.458,,,+PB1
                            +PB1,1.,.5
                            $
                            $     DEFINE MATERIAL PROPERTIES
                            MAT1,201,30.E6,,.3
                            $
                            $     DEFINE UNIFORM DISTRIBUTED LOAD
                            PLOAD1,10,1,FY,FR,0.,-22.,1.,-22.
                            PLOAD1,10,2,FY,FR,0.,-22.,1.,-22.
                            PLOAD1,10,3,FY,FR,0.,-22.,1.,-22.
                            $
                            $     DEFINE CONCENTRATED MOMENT AT FREE END
                            MOMENT,10,4,,120.,0.,0.,1.
                            ENDDATA
                            INPUT BULK DATA CARD COUNT =      26
```

```
 EXAMPLE 2                                                          SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4
  CANTILEVER BEAM
  DISTRIBUTED LOAD AND END MOMENT
                                               S O R T E D   B U L K   D A T A   E C H O
              CARD
              COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                 1-        CBAR    1       101     1       2       0.      1.      0.
                 2-        CBAR    2       101     2       3       0.      1.      0.
                 3-        CBAR    3       101     3       4       0.      1.      0.
                 4-        GRID    1               0.      0.      0.              123456
                 5-        GRID    2               10.     0.      0.
                 6-        GRID    3               20.     0.      0.
                 7-        GRID    4               30.     0.      0.
                 8-        MAT1    201     30.E6           .3
                 9-        MOMENT  10      4               120.    0.      0.      1.
                10-        PBAR    101     201     2.      .667    .1667   .458                    +PB1
                11-        +PB1    1.      .5
                12-        PLOAD1  10      1       FY      FR      0.      -22.    1.      -22.
                13-        PLOAD1  10      2       FY      FR      0.      -22.    1.      -22.
                14-        PLOAD1  10      3       FY      FR      0.      -22.    1.      -22.
                           ENDDATA
                     TOTAL COUNT=       15
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     5
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT
 USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
                                                  OLOAD    RESULTANT
              T1             T2             T3             R1             R2             R3
        1   0.0000000E+00 -6.6000000E+02  0.0000000E+00  0.0000000E+00  0.0000000E+00 -9.7800000E+03
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     6
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
   LOAD SEQ. NO.             EPSILON              EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
               1         -7.7706584E-17          1.4106205E+01
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     7
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT
 USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
                                                  SPCFORCE RESULTANT
              T1             T2             T3             R1             R2             R3
        1   0.0000000E+00  6.6000000E+02  0.0000000E+00  0.0000000E+00  0.0000000E+00  9.7800000E+03
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     8
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT

                                             D I S P L A C E M E N T   V E C T O R

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G      0.0            0.0            0.0            0.0            0.0            0.0
            2      G      0.0           -1.939863E-02   0.0            0.0            0.0           -3.421623E-03
            3      G      0.0           -6.110278E-02   0.0            0.0            0.0           -4.644345E-03
            4      G      0.0           -1.086207E-01   0.0            0.0            0.0           -4.767616E-03
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     9
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT

                              F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G      0.0            6.600000E+02   0.0            0.0            0.0            9.780000E+03
```

```
   EXAMPLE 2                                                                    SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    10
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT
```

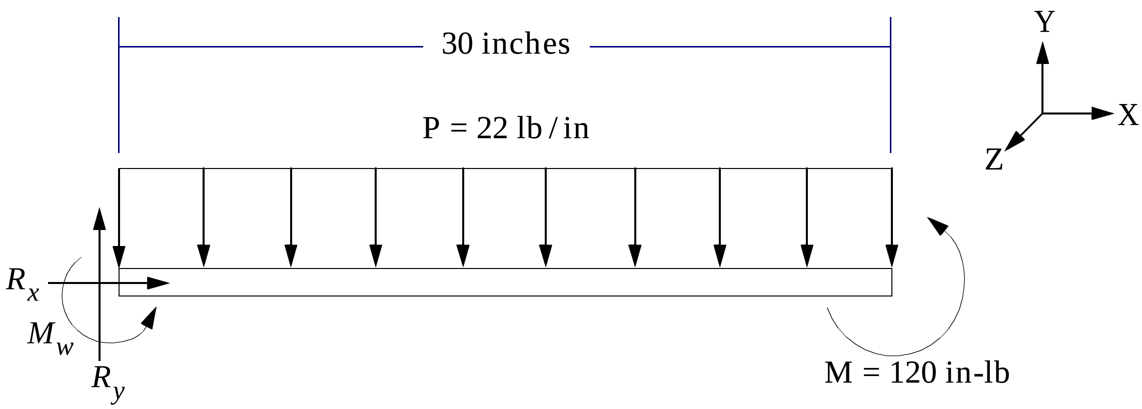
Listing 13-2 (continued)

```
   EXAMPLE 2                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    11
    CANTILEVER BEAM
    DISTRIBUTED LOAD AND END MOMENT
* * * *  D B D I C T   P R I N T  * * * *      SUBDMAP = PRTSUM  ,  DMAP STATEMENT NO.     13
                                   * * * *  A N A L Y S I S  S U M M A R Y  T A B L E  * * * *
 SEID  PEID PROJ VERS APRCH      SEMG SEMR SEKR SELG SELR MODES DYNRED SOLLIN PVALID SOLNL LOOPID DESIGN CYCLE SENSITIVITY
-------------------------------------------------------------------------------------------------------------------------
    0     0    1    1 '        '    T    T    T    T    T     F      F      T      0     F     -1            0           F
SEID = SUPERELEMENT ID.
PEID = PRIMARY SUPERELEMENT ID OF IMAGE SUPERELEMENT.
PROJ = PROJECT ID NUMBER.
VERS = VERSION ID.
APRCH = BLANK FOR STRUCTURAL ANALYSIS.  HEAT FOR HEAT TRANSFER ANALYSIS.
SEMG = STIFFNESS AND MASS MATRIX GENERATION STEP.
SEMR = MASS MATRIX REDUCTION STEP (INCLUDES EIGENVALUE SOLUTION FOR MODES).
SEKR = STIFFNESS MATRIX REDUCTION STEP.
SELG = LOAD MATRIX GENERATION STEP.
SELR = LOAD MATRIX REDUCTION STEP.
MODES = T (TRUE) IF NORMAL MODES OR BUCKLING MODES CALCULATED.
DYNRED = T (TRUE) MEANS GENERALIZED DYNAMIC AND/OR COMPONENT MODE REDUCTION PERFORMED.
SOLLIN = T (TRUE) IF LINEAR SOLUTION EXISTS IN DATABASE.
PVALID = P-DISTRIBUTION ID OF P-VALUE FOR P-ELEMENTS
LOOPID = THE LAST LOOPID VALUE USED IN THE NONLINEAR ANALYSIS.  USEFUL FOR RESTARTS.
SOLNL = T (TRUE) IF NONLINEAR SOLUTION EXISTS IN DATABASE.
DESIGN CYCLE = THE LAST DESIGN CYCLE (ONLY VALID IN OPTIMIZATION).
SENSITIVITY = SENSITIVITY MATRIX GENERATION FLAG.

                                        * * * END OF JOB * * *
```

Reviewing the Results

First, we review the .f06 output file for any warning or error messages. None are present in this file. Next, look at epsilon on page 6 of the output. Its value of -7.77E-17 is indeed very small, showing no evidence of numerical difficulties. Finally, we review the reaction forces (forces of single point constraint, or SPC forces) at the wall. As a check, a free body diagram of the structure is used to solve for reaction forces as follows:



Solving for the reactions at the wall, we obtain:

Forces in x: $\overset{+}{\rightarrow} \sum F_x = 0 = R_x$

$$\boxed{R_x = 0}$$

Forces in y: $+\uparrow \sum F_y = 0 = R_y - (22 \text{ lb/in})(30 \text{ in})$

$$\boxed{R_y = 660 \text{ lbs}}$$

Moment at wall: $\circlearrowright^{+} \sum M_{wall} = M_w + 120 \text{ in-lb} - (660 \text{lbs})(15 \text{ in})$

$$\boxed{M_w = 9780 \text{ in-lb}}$$

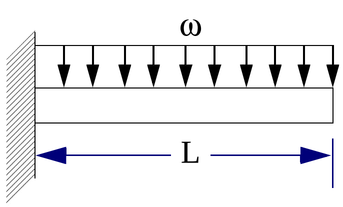
The SPC forces are listed on page 9 of the MSC.Nastran results. The T2 reaction (force at grid point 1 in the y direction) is +660 lbs. The R3 reaction (moment about the z axis) is +9780 lb. Thus, we can be confident that the loads were applied correctly, and at least the static equilibrium of the problem makes sense.

The displacement results are shown on page 8 of the .f06 file. Note that all displacements at the wall (GRID 1) are exactly zero, as they should be. The free end deflection in the y direction (T2 of GRID 4) is -1.086207E-1 in.

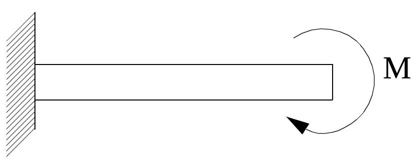
As a final observation, note that there is no axial shortening of the beam as it deflects downward (all T1's are exactly zero). This is a consequence of the simplifying small displacement assumptions built into slender beam theory, and MSC's beam elements when used in linear analysis. If the load on the beam is such that large displacement occurs, nonlinear analysis must be used to update the element matrices as the structure deforms. The shortening terms will then be part of the solution.

Comparison with Theory

The theory solution to this problem is as follows:



$$\text{Maximum deflection} = \frac{\omega L^4}{8EI}$$



$$\text{Maximum deflection} = \frac{ML^2}{2EI}$$

Using superposition, the net deflection at free end is given by:

$$-\left(\frac{ML^2}{2EI} + \frac{\omega L^4}{8EI}\right) = -\frac{L^2}{2EI}\left(M + \frac{\omega L^2}{4}\right) = -0.10862 \text{ inches}$$

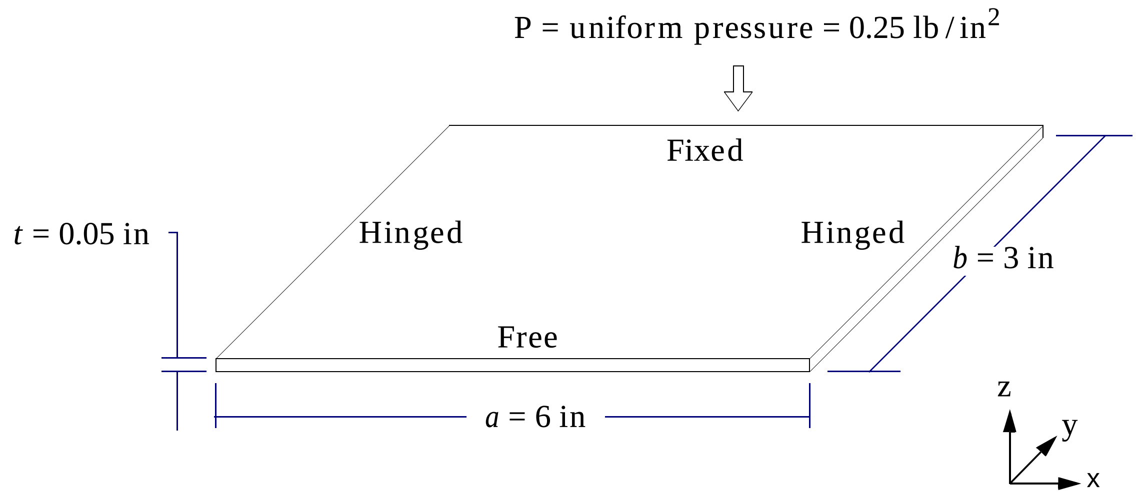
Thus, we are in exact agreement with the MSC.Nastran result.

It should be noted that simple beam bending problems such as this give exact answers, even with one element. This is a very special case (as was the extensional rod example in "**Overview of the MSC.Nastran Finite Element Model**" on page 7), and is by no means typical of real world problems.

13.2 Rectangular Plate (fixed-hinged-hinged-free) with a Uniform Lateral Pressure Load

Problem Statement

Create an MSC.Nastran model to analyze the thin rectangular plate shown in **Figure 13-2**. The plate is subject to a uniform pressure load of $.25\ lb/in^2$ the -z direction. Find the maximum deflection of the plate.



$$E = 30 \times 10^6\ lb/in^2$$
$$\nu = 0.3$$

Figure 13-2 Plate Geometry, Boundary Conditions, and Load

The Finite Element Model

Designing the Model

First, we need to examine the structure to verify that it can reasonably qualify as a thin plate. The thickness is 1/60 of the next largest dimension (3 inches), which is satisfactory.

Next, we observe by inspection that the maximum deflection, regardless of the actual value, should occur at the center of the free edge. Thus, it will be helpful to locate a grid point there to recover the maximum displacement.

As a matter of good practice, we wish to design a model with the fewest elements that will do the job. In our case, doing the job means good displacement accuracy. The model shown in **Figure 13-3** contains 20 GRID points and 12 CQUAD4 elements, which we hope will yield reasonable displacement results. If we have reason to question the accuracy of the solution, we can always rerun the model with a finer mesh.

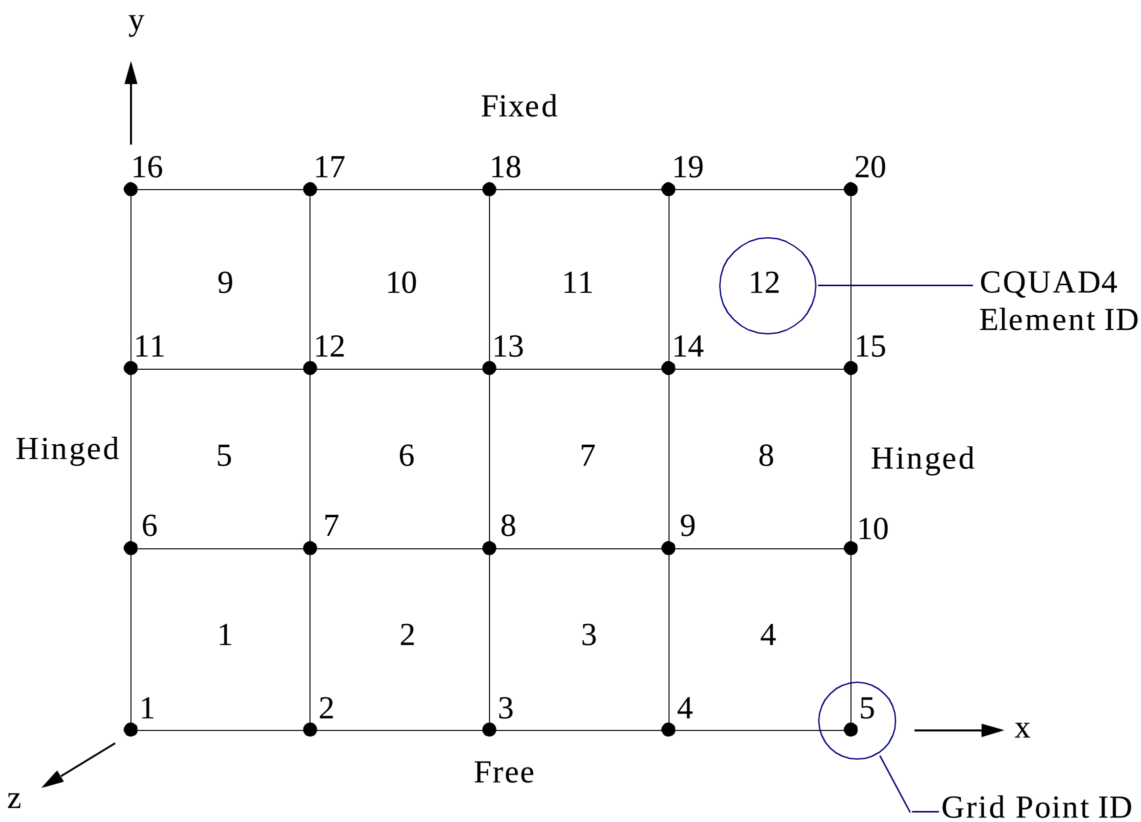


Figure 13-3 Plate Finite Element Model

Applying the Load

The uniform pressure load is applied to all plate elements using the PLOAD2 entry. Only one PLOAD2 entry is required by using the "THRU" feature (elements 1 THRU 12). The positive normal to each plate element (as dictated by the GRID point ordering sequence) is in the negative z axis direction, which is the same direction as the pressure load. Therefore, the value of pressure in Field 3 of the PLOAD2 entry is positive.

Applying the Constraints

SPC1 entries are used to model the structure's constraints. The SPC1 entries have a set ID of 10, which is selected by the Case Control command SPC = 100. The constraints on the structure are shown in **Figure 13-4**.

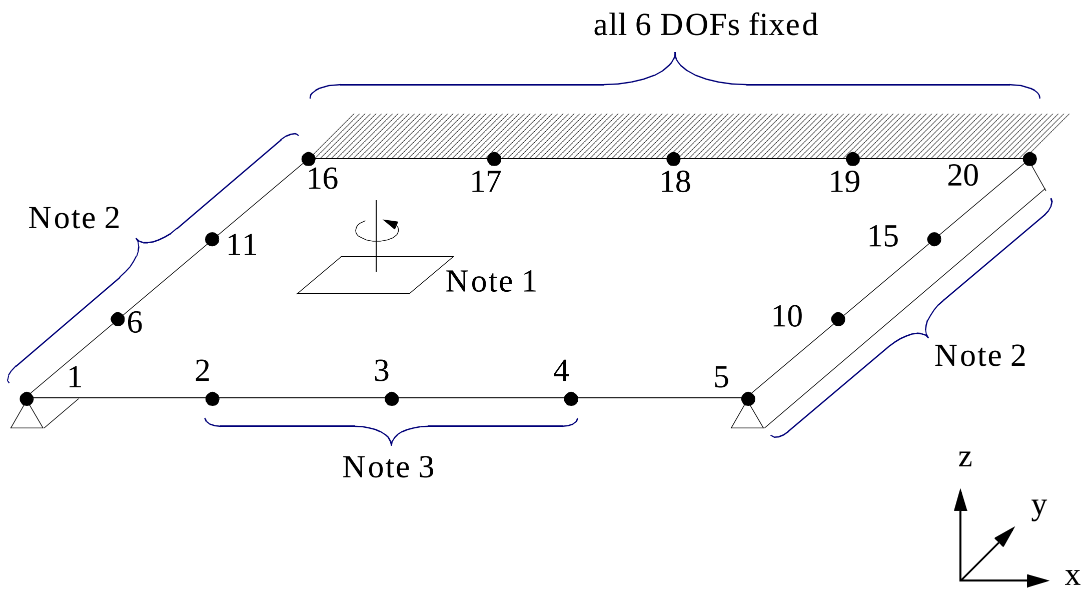


Figure 13-4 Constraints on the Plate Structure

Note:

1. The out-of-plane rotational DOF (degree of freedom 6) is constrained for all grids in the model. This is a requirement of a CQUAD4 flat plate element, and has nothing to do with this specific problem.
2. Grids 16 and 20, shared with the fixed edge, are fixed-the greater constraint governs. For the remaining grids:

   Displacements Allowed: Rotation about y-axis (DOF 5)

   Displacements Not Allowed:Rotation about x-axis (DOF 4)
   Translation in x, y, or z (DOFs 1, 2, 3)
3. The non-corner grids of the free edge have no additional constraints.

The SPC1 entries are written as follows:

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Alternate Format:

SPC1	SID	C	G1	"THRU"	G2				

Out-of-plane Rotations:

SPC1	100	6	1	THRU	20				

Hinged Edges:

spc1	100	1234	1	6	11	5	10	15	

Fixed Edge:

SPC1	100	123456	16	THRU	20				

Note that some constraints are redundantly specified. For example, GRID 17 is constrained in all 6 DOFs with the fixed edge SPC1, and again in DOF 6 with the out-of-plane rotational constraint. This is perfectly acceptable, and keeps the constraint bookkeeping a little tidier.

Output Requests

The problem statement requires displacements. As a matter of good practice, we will also request SPC forces to check the model's reactions. Thus, the following output requests are included in the Case Control Section:

```
DISP=ALL
SPCF=ALL
```

The Input File

The complete input file is shown in **Listing 13-3**.

Listing 13-3

```
ID MPM,EXAMPLE3
SOL 101
TIME 100
CEND
SPCF=ALL
DISP=ALL
TITLE=PLATE EXAMPLE
SUBTITLE=FIXED-HINGED-HINGED-FREE
LABEL=UNIFORM LATERAL PRESSURE LOAD (0.25 lb/in**2)
SPC=100
ECHO=BOTH
LOAD=5
$
BEGIN BULK
$     DEFINE GRID POINTS
GRID,1,,0.,0.,0.
GRID,2,,1.5,0.,0.
GRID,3,,3.0,0.,0.
GRID,4,,4.5,0.,0.
GRID,5,,6.0,0.,0.
GRID,6,,0.,1.,0.
GRID,7,,1.5,1.,0.
GRID,8,,3.0,1.,0.
GRID,9,,4.5,1.,0.
GRID,10,,6.0,1.,0.
GRID,11,,0.,2.,0.
GRID,12,,1.5,2.,0.
GRID,13,,3.0,2.,0.
GRID,14,,4.5,2.,0.
GRID,15,,6.0,2.,0.
GRID,16,,0.,3.,0.
GRID,17,,1.5,3.,0.
GRID,18,,3.0,3.,0.
GRID,19,,4.5,3.,0.
```

```
GRID,20,,6.0,3.,0.
$
$     DEFINE PLATE ELEMENTS
CQUAD4,1,101,1,6,7,2
CQUAD4,2,101,2,7,8,3
CQUAD4,3,101,3,8,9,4
CQUAD4,4,101,4,9,10,5
CQUAD4,5,101,6,11,12,7
CQUAD4,6,101,7,12,13,8
CQUAD4,7,101,8,13,14,9
CQUAD4,8,101,9,14,15,10
CQUAD4,9,101,11,16,17,12
CQUAD4,10,101,12,17,18,13
CQUAD4,11,101,13,18,19,14
CQUAD4,12,101,14,19,20,15
$
$     DEFINE PRESSURE LOAD ON PLATES
PLOAD2,5,0.25,1,THRU,12
$     DEFINE PROPERTIES OF PLATE ELEMENTS
PSHELL,101,105,.05,105,,105
MAT1,105,30.E6,,0.3
$
$     DEFINE FIXED EDGE
SPC1,100,123456,16,THRU,20
$
$     DEFINE HINGED EDGES
SPC1,100,1234,1,6,11,5,10,15
$
$     CONSTRAIN OUT-OF-PLANE ROTATION FOR ALL GRIDS
SPC1,100,6,1,THRU,20
ENDDATA
```

MSC.Nastran Results

The .f06 Results File

The MSC.Nastran results are shown in **Listing 13-4**.

Listing 13-4

```
                     THIS PROGRAM IS CONFIDENTIAL AND A TRADE SECRET OF MSC.SOFTWARE CORP.  THE RECEIPT OR
                    POSSESSION OF THIS PROGRAM DOES NOT CONVEY ANY RIGHTS TO REPRODUCE OR DISCLOSE ITS CONTENTS, OR TO
                   MANUFACTURE, USE, OR SELL ANYTHING HEREIN, IN WHOLE OR IN PART, WITHOUT THE SPECIFIC WRITTEN CONSENT
        OF MSC.SOFTWARE CORPORATION.

* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
* *                                  * *
* *           MSC.SOFTWARE           * *
* *               CORP               * *
* *                                  * *
* *      M S C . N a s t r a n       * *
* *                                  * *
* *       VERSION - 2001.0.1         * *
* *                                  * *
* *          SEP 10, 2001            * *
* *                                  * *
* *                                  * *
* *              Intel               * *
* *                                  * *
* *MODEL PentiumIII/995 (MERCED.scm)* *
* *                                  * *
* *  Windows 2000 5.0 (Build 2195)   * *
* *                                  * *
* *                                  * *
* *                                  * *
* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
```

```
                                                                       SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    1

        N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O
     ID MPM,EXAMPLE3
     SOL 101
     TIME 100
     CEND
```

```
  PLATE EXAMPLE                                                         SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     2
   FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                                     C A S E    C O N T R O L    D E C K    E C H O
                 CARD
                 COUNT
                  1      SPCF=ALL
                  2      DISP=ALL
                  3      TITLE=PLATE EXAMPLE
                  4      SUBTITLE=FIXED-HINGED-HINGED-FREE
                  5      LABEL=UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                  6      SPC=100
                  7      ECHO=BOTH
                  8      LOAD=5
                  9      $
                 10      BEGIN BULK
```

Listing 13-4 (continued)

```
 PLATE EXAMPLE                                                  SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                                               I N P U T   B U L K   D A T A   D E C K   E C H O
                              .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                              $     DEFINE GRID POINTS
                              GRID,1,,0.,0.,0.
                              GRID,2,,1.5,0.,0.
                              GRID,3,,3.0,0.,0.
                              GRID,4,,4.5,0.,0.
                              GRID,5,,6.0,0.,0.
                              GRID,6,,0.,1.,0.
                              GRID,7,,1.5,1.,0.
                              GRID,8,,3.0,1.,0.
                              GRID,9,,4.5,1.,0.
                              GRID,10,,6.0,1.,0.
                              GRID,11,,0.,2.,0.
                              GRID,12,,1.5,2.,0.
                              GRID,13,,3.0,2.,0.
                              GRID,14,,4.5,2.,0.
                              GRID,15,,6.0,2.,0.
                              GRID,16,,0.,3.,0.
                              GRID,17,,1.5,3.,0.
                              GRID,18,,3.0,3.,0.
                              GRID,19,,4.5,3.,0.
                              GRID,20,,6.0,3.,0.
                              $
                              $     DEFINE PLATE ELEMENTS
                              CQUAD4,1,101,1,6,7,2
                              CQUAD4,2,101,2,7,8,3
                              CQUAD4,3,101,3,8,9,4
                              CQUAD4,4,101,4,9,10,5
                              CQUAD4,5,101,6,11,12,7
                              CQUAD4,6,101,7,12,13,8
                              CQUAD4,7,101,8,13,14,9
                              CQUAD4,8,101,9,14,15,10
                              CQUAD4,9,101,11,16,17,12
                              CQUAD4,10,101,12,17,18,13
                              CQUAD4,11,101,13,18,19,14
                              CQUAD4,12,101,14,19,20,15
                              $
                              $     DEFINE PRESSURE LOAD ON PLATES
                              PLOAD2,5,0.25,1,THRU,12
                              $     DEFINE PROPERTIES OF PLATE ELEMENTS
                              PSHELL,101,105,.05,105,,105
                              MAT1,105,30.E6,,0.3
                              $
                              $     DEFINE FIXED EDGE
                              SPC1,100,123456,16,THRU,20
                              $
                              $     DEFINE HINGED EDGES
                              SPC1,100,1234,1,6,11,5,10,15
                              $
                              $     CONSTRAIN OUT-OF-PLANE ROTATION FOR ALL GRIDS
                              SPC1,100,6,1,THRU,20
                              ENDDATA
                              INPUT BULK DATA CARD COUNT =      51
```

Listing 13-4 (continued)

```
 PLATE EXAMPLE                                                               SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4
    FIXED-HINGED-HINGED-FREE
 UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                                                         S O R T E D   B U L K   D A T A   E C H O
                CARD
                COUNT        .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                   1-        CQUAD4  1       101     1       6       7       2
                   2-        CQUAD4  2       101     2       7       8       3
                   3-        CQUAD4  3       101     3       8       9       4
                   4-        CQUAD4  4       101     4       9       10      5
                   5-        CQUAD4  5       101     6       11      12      7
                   6-        CQUAD4  6       101     7       12      13      8
                   7-        CQUAD4  7       101     8       13      14      9
                   8-        CQUAD4  8       101     9       14      15      10
                   9-        CQUAD4  9       101     11      16      17      12
                  10-        CQUAD4  10      101     12      17      18      13
                  11-        CQUAD4  11      101     13      18      19      14
                  12-        CQUAD4  12      101     14      19      20      15
                  13-        GRID    1               0.      0.      0.
                  14-        GRID    2               1.5     0.      0.
                  15-        GRID    3               3.0     0.      0.
                  16-        GRID    4               4.5     0.      0.
                  17-        GRID    5               6.0     0.      0.
                  18-        GRID    6               0.      1.      0.
                  19-        GRID    7               1.5     1.      0.
                  20-        GRID    8               3.0     1.      0.
                  21-        GRID    9               4.5     1.      0.
                  22-        GRID    10              6.0     1.      0.
                  23-        GRID    11              0.      2.      0.
                  24-        GRID    12              1.5     2.      0.
                  25-        GRID    13              3.0     2.      0.
                  26-        GRID    14              4.5     2.      0.
                  27-        GRID    15              6.0     2.      0.
                  28-        GRID    16              0.      3.      0.
                  29-        GRID    17              1.5     3.      0.
                  30-        GRID    18              3.0     3.      0.
                  31-        GRID    19              4.5     3.      0.
                  32-        GRID    20              6.0     3.      0.
                  33-        MAT1    105     30.E6           0.3
                  34-        PLOAD2  5       0.25    1       THRU    12
                  35-        PSHELL  101     105     .05     105             105
                  36-        SPC1    100     6       1       THRU    20
                  37-        SPC1    100     1234    1       6       11      5       10      15
                  38-        SPC1    100     123456  16      THRU    20
                             ENDDATA
                       TOTAL COUNT=        39
```

```
 PLATE EXAMPLE                                                               SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     5
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
 USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.

                                                           OLOAD    RESULTANT

                         T1             T2             T3             R1             R2             R3

        1   0.0000000E+00  0.0000000E+00 -4.5000000E+00 -6.7500000E+00  1.3500000E+01  0.0000000E+00
```

```
 PLATE EXAMPLE                                                               SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     6
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

   LOAD SEQ. NO.             EPSILON              EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS

               1          1.8446709E-15         2.1564697E-03
```

Listing 13-4 (continued)

```
 PLATE EXAMPLE                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     7
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.

                                             SPCFORCE RESULTANT

               T1             T2             T3             R1             R2             R3

      1   0.0000000E+00  0.0000000E+00  4.5000000E+00  6.7500000E+00 -1.3500000E+01  0.0000000E+00
```

```
 PLATE EXAMPLE                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     8
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                                               D I S P L A C E M E N T   V E C T O R

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G      0.0            0.0            0.0            0.0            2.010781E-03   0.0
            2      G      0.0            0.0           -2.627660E-03   1.125413E-03   1.375545E-03   0.0
            3      G      0.0            0.0           -3.678445E-03   1.583532E-03  -1.514721E-21   0.0
            4      G      0.0            0.0           -2.627660E-03   1.125413E-03  -1.375545E-03   0.0
            5      G      0.0            0.0            0.0            0.0           -2.010781E-03   0.0
            6      G      0.0            0.0            0.0            0.0            1.188741E-03   0.0
            7      G      0.0            0.0           -1.544730E-03   1.037140E-03   7.948730E-04   0.0
            8      G      0.0            0.0           -2.149283E-03   1.469851E-03  -2.318725E-21   0.0
            9      G      0.0            0.0           -1.544730E-03   1.037140E-03  -7.948730E-04   0.0
           10      G      0.0            0.0            0.0            0.0           -1.188741E-03   0.0
           11      G      0.0            0.0            0.0            0.0            4.129650E-04   0.0
           12      G      0.0            0.0           -5.316482E-04   9.315911E-04   2.673020E-04   0.0
           13      G      0.0            0.0           -7.322498E-04   1.282421E-03   1.553187E-21   0.0
           14      G      0.0            0.0           -5.316482E-04   9.315911E-04  -2.673020E-04   0.0
           15      G      0.0            0.0            0.0            0.0           -4.129650E-04   0.0
           16      G      0.0            0.0            0.0            0.0            0.0            0.0
           17      G      0.0            0.0            0.0            0.0            0.0            0.0
           18      G      0.0            0.0            0.0            0.0            0.0            0.0
           19      G      0.0            0.0            0.0            0.0            0.0            0.0
           20      G      0.0            0.0            0.0            0.0            0.0            0.0
```

```
 PLATE EXAMPLE                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     9
    FIXED-HINGED-HINGED-FREE
   UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
                                   F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G      0.0            0.0            3.572939E-01  -9.891411E-02   0.0            0.0
            5      G      0.0            0.0            3.572939E-01  -9.891411E-02   0.0            0.0
            6      G      0.0            0.0            4.663838E-01  -4.931600E-02   0.0            0.0
           10      G      0.0            0.0            4.663838E-01  -4.931600E-02   0.0            0.0
           11      G      0.0            0.0            3.687456E-01   2.768068E-01   0.0            0.0
           15      G      0.0            0.0            3.687456E-01   2.768068E-01   0.0            0.0
           16      G      0.0            0.0           -4.810094E-01   1.029394E-01   2.657713E-02   0.0
           17      G      0.0            0.0            9.980871E-01  -7.319196E-01  -5.360183E-03   0.0
           18      G      0.0            0.0            1.080998E+00  -1.002403E+00   0.0            0.0
           19      G      0.0            0.0            9.980871E-01  -7.319196E-01   5.360183E-03   0.0
           20      G      0.0            0.0           -4.810094E-01   1.029394E-01  -2.657713E-02   0.0
```

Listing 13-4 (continued)

```
 PLATE EXAMPLE                                                        SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01    PAGE    11
    FIXED-HINGED-HINGED-FREE
  UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
* * * *  D B D I C T   P R I N T  * * * *      SUBDMAP = PRTSUM  ,  DMAP STATEMENT NO.     13

                                   * * * *  A N A L Y S I S  S U M M A R Y  T A B L E  * * * *

 SEID  PEID PROJ VERS APRCH      SEMG SEMR SEKR SELG SELR MODES DYNRED SOLLIN PVALID SOLNL LOOPID DESIGN CYCLE SENSITIVITY
 --------------------------------------------------------------------------------------------------------------------------
     0     0    1    1 '        '    T    T    T    T    T     F      F      T      0     F     -1            0           F

0SEID = SUPERELEMENT ID.
 PEID = PRIMARY SUPERELEMENT ID OF IMAGE SUPERELEMENT.
 PROJ = PROJECT ID NUMBER.
 VERS = VERSION ID.
 APRCH = BLANK FOR STRUCTURAL ANALYSIS.  HEAT FOR HEAT TRANSFER ANALYSIS.
 SEMG = STIFFNESS AND MASS MATRIX GENERATION STEP.
 SEMR = MASS MATRIX REDUCTION STEP (INCLUDES EIGENVALUE SOLUTION FOR MODES).
 SEKR = STIFFNESS MATRIX REDUCTION STEP.
 SELG = LOAD MATRIX GENERATION STEP.
 SELR = LOAD MATRIX REDUCTION STEP.
 MODES = T (TRUE) IF NORMAL MODES OR BUCKLING MODES CALCULATED.
 DYNRED = T (TRUE) MEANS GENERALIZED DYNAMIC AND/OR COMPONENT MODE REDUCTION PERFORMED.
 SOLLIN = T (TRUE) IF LINEAR SOLUTION EXISTS IN DATABASE.
 PVALID = P-DISTRIBUTION ID OF P-VALUE FOR P-ELEMENTS
 LOOPID = THE LAST LOOPID VALUE USED IN THE NONLINEAR ANALYSIS.  USEFUL FOR RESTARTS.
 SOLNL = T (TRUE) IF NONLINEAR SOLUTION EXISTS IN DATABASE.
 DESIGN CYCLE = THE LAST DESIGN CYCLE (ONLY VALID IN OPTIMIZATION).
 SENSITIVITY = SENSITIVITY MATRIX GENERATION FLAG.

                                              * * * END OF JOB * * *
```

Reviewing the Results

The value of epsilon, listed on page 6 of the output, is small, indicating a numerically well-behaved problem. A plot of the deformed plate is shown in **Figure 13-5**. As expected, the maximum displacement (-3.678445E-3 inches) occurs at grid point 3 in the -T3 direction. This deflection is approximately one-fourteenth the thickness of the plate, and is therefore a fairly reasonable “small” displacement.

It is also useful to check the applied loads against the reaction forces. We have

$$\text{Total Lateral Applied Force} = (0.25\ \text{lb/in}^2)(3\ \text{in})(6\ \text{in}) = 4.5\ \text{lbs}$$

which is in agreement with the T3 direction SPCFORCE resultant listed on page 7 of the output. Note that the SPCFORCE is positive, and the applied load is in the negative z (-T3) direction.

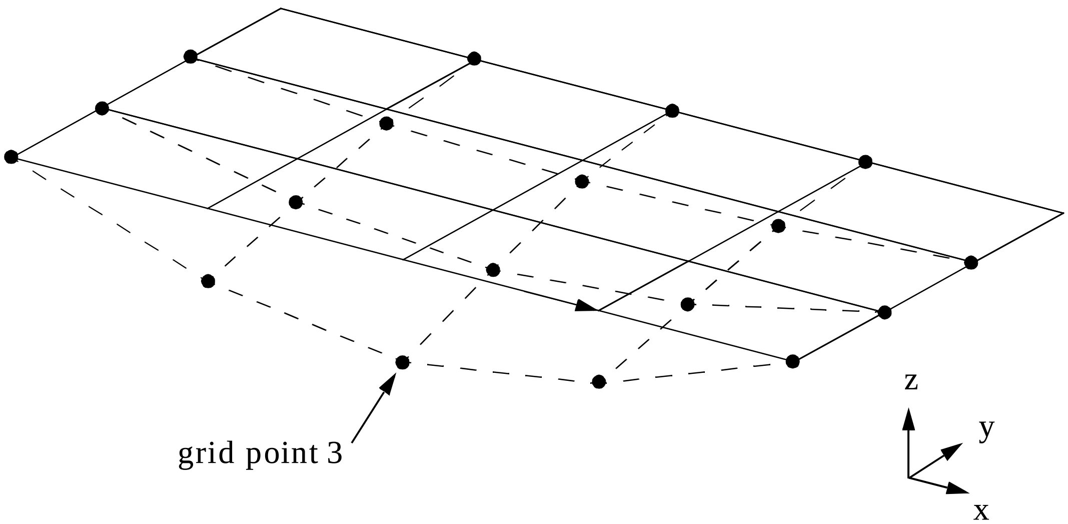


Figure 13-5 Deformed Shape

Comparison with Theory

Article 46 of Timoshenko, *Theory of Plates and Shells*, 2nd ed., gives the analytical solution for the maximum deflection of a fixed-hinged-hinged-free plate with a uniform lateral load as:

$$W_{max} = (0.0582\ qb^4/D) \qquad \left(\text{for } b/a = \frac{1}{2}\right)$$

where:

$$q = \text{lateral pressure} = 0.25\ \text{lb/in}^2$$

$$D = \frac{Et^3}{12(1-\nu^2)} = \frac{(30 \times 10^6\ \text{lb/in}^2)(0.5\ \text{in})^3}{12(1-0.3^2)} = 343.407$$

Therefore, the maximum deflection is:

$$W_{max} = \frac{.0582(0.25\ \text{lb/in}^2)(3\ \text{in})^4}{343.407\ \text{in-lb}} = 3.43193\text{E-3 in}$$

The MSC.Nastran solution at grid point 3 is:

$$W_{max} = 3.678445\text{E-3 in}$$

The MSC.Nastran result (which includes transverse shear) is 7.2% greater than the theory solution. The theory solution does not account for transverse shear deflection. Rerunning the model without shear (by eliminating MID3 in field 7of the PSHELL entry) gives a deflection of

$$W_{max}\ (\text{no shear}) = 3.664290\text{E-3 in}$$

Thus, for this thin plate, adding shear deflection results in less than half a percent difference in the total deflection.

13.3 Gear Tooth with Solid Elements

In this problem we create a very simple CHEXA solid element model of a gear tooth. In addition, MSC.Nastran's subcase feature is used to apply two load cases in a single run.

Problem Statement

Two spur gears are in contact as shown in **Figure 13-6**. The gears are either aligned or misaligned. In the aligned case, a distributed load of 600 N / mm exists across the line of contact between two teeth. The line of contact is located at a radius of 99.6 mm from the gear's center. In the misaligned case, a concentrated load of 6000 N acts at a single point of contact at the edge of a tooth. The gear teeth are 10 mm wide and 23.5 mm high (from base to tip). The gear's material properties are:

$$E = 2.0 \times 10^5 \text{ MPa}$$

$$\nu = 0.3$$

The goal is to obtain a rough estimate of a gear tooth's peak von Mises stress for each load case. von Mises stress, a commonly used quantity in finite element stress analysis, is given by:

$$\frac{1}{\sqrt{2}}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{yz})^2 + 6(\tau_{zx})^2 + 6(\tau_{xy} \qquad \text{Eq. 13-1}$$

Figure 13-6 Spur Gears

The Finite Element Model

A single gear tooth is modeled using two CHEXA solid elements with midside grid points as shown in **Figure 13-7**. Midside grid points are useful when the shape of a structure is complex or when bending effects are important.

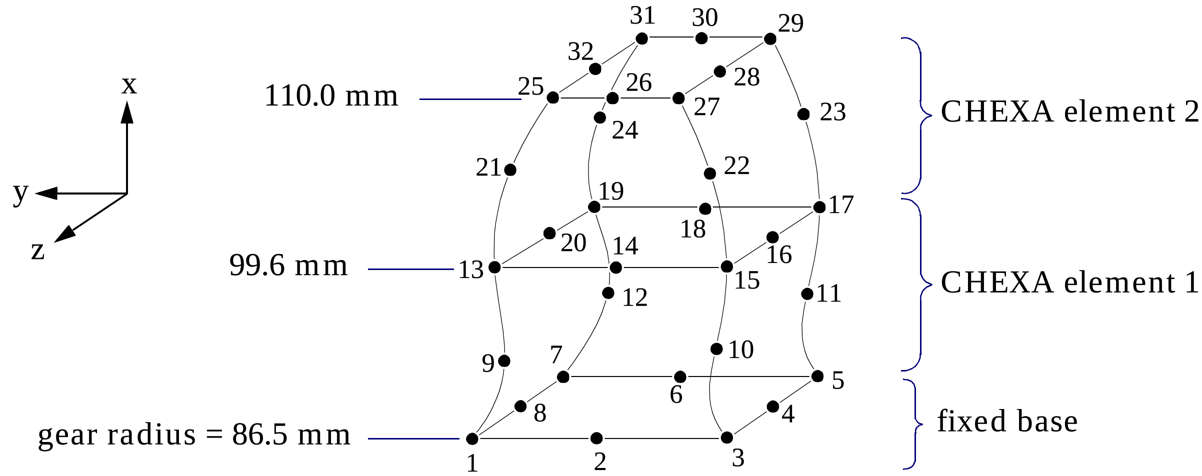


Figure 13-7 Finite Element Model of a Single Gear Tooth

Applying the Loads

Subcase 1 represents aligned gear teeth and uses the distributed load shown in **Figure 13-8**. The total applied load is given by:

$$\text{Total Load} = \text{Distributed Load} \cdot \text{Width of Gear Tooth}$$

$$= 600 \ \text{N/mm} \cdot 10 \ \text{mm} = 6000 \ \text{N}$$

In order to approximate the “contact patch” of mating gear teeth, we distribute the total force of 6000 N across the line of contact with 1000 N on each corner grid (grid points 15 and 17) and 4000 N on the center grid (grid point 16). A load set identification number of 41 (arbitrarily chosen) is given to the three FORCE Bulk Data entries of subcase 1.

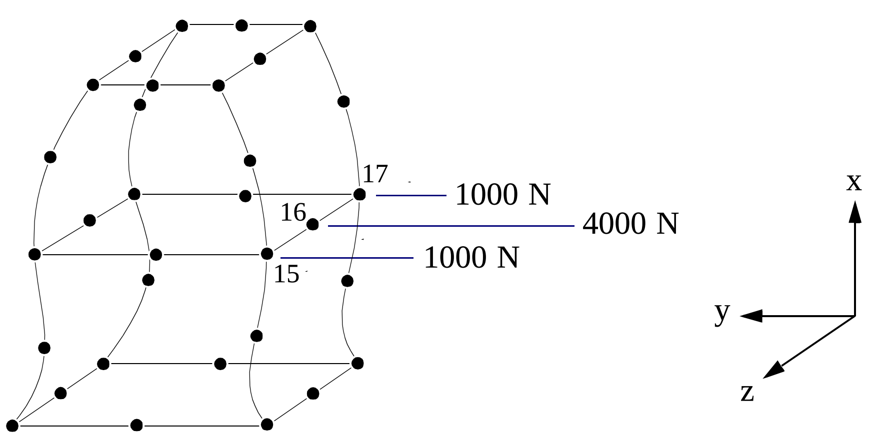


Figure 13-8 Subcase 1 - Gears in Alignment (Distributed Load)

Subcase 2 represents misaligned gear teeth and uses a single concentrated force of 6000 N as shown in **Figure 13-9**. A load set identification number of 42 is given to the single FORCE entry of subcase 2. Note that the total applied force (i.e., force transmitted from one tooth to the next) is the same in both subcases.

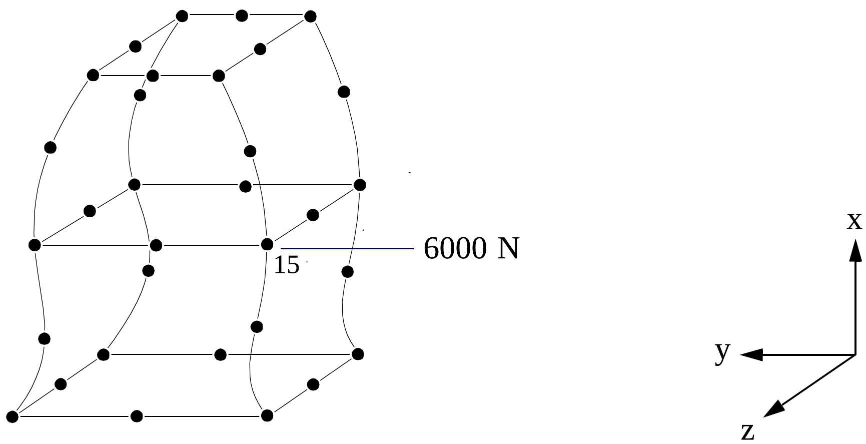


Figure 13-9 Subcase 2 - Gears Misaligned (Concentrated Load)

Applying the Constraints

The base of the tooth is assumed to be fixed as shown in **Figure 13-7**. Consequently, grid points 1 through 8 are constrained to zero displacement in their translational DOFs (1, 2, and 3). Recall that solid elements have only translational DOFs and no rotational DOFs. Since each grid point starts out with all six DOFs, the remaining “unattached” rotational DOFs must be constrained to prevent numerical singularities. Thus, all grid points in the model (1 through 32) are constrained in DOFs 4, 5, and 6. The constraints are applied using SPC1 Bulk Data entries.

Output Requests

Stress output is selected with the Case Control command STRESS = ALL. Note that this command appears above the subcase level and therefore applies to both subcases.

The Input File

The complete input file is shown in **Listing 13-5**.

Listing 13-5 Gear Tooth Input File

```
ID   SOLID, ELEMENT MODEL
SOL  101
TIME 100
CEND
TITLE         = GEAR TOOTH EXAMPLE
STRESS        = ALL
SPC           = 30
SUBCASE 1
   LOAD       = 41
   SUBTITLE   = GEAR TOOTH UNDER 600 N/mm LINE LOAD
SUBCASE 2
   LOAD       = 42
   SUBTITLE   = GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
BEGIN BULK
GRID,          1,         ,    86.5,    12.7,     5.0
GRID,          2,         ,    86.5,     0.0,     5.0
GRID,          3,         ,    86.5,   -12.7,     5.0
GRID,          4,         ,    86.5,   -12.7,     0.0
GRID,          5,         ,    86.5,   -12.7,    -5.0
GRID,          6,         ,    86.5,     0.0,    -5.0
GRID,          7,         ,    86.5,    12.7,    -5.0
```

```
GRID,         8,        ,   86.5,   12.7,    0.0
GRID,         9,        ,   93.0,    8.7,    5.0
GRID,        10,        ,   93.0,   -8.7,    5.0
GRID,        11,        ,   93.0,   -8.7,   -5.0
GRID,        12,        ,   93.0,    8.7,   -5.0
GRID,        13,        ,   99.6,    7.8,    5.0
GRID,        14,        ,  100.0,    0.0,    5.0
GRID,        15,        ,   99.6,   -7.8,    5.0
GRID,        16,        ,   99.6,   -7.8,    0.0
GRID,        17,        ,   99.6,   -7.8,   -5.0
GRID,        18,        ,  100.0,    0.0,   -5.0
GRID,        19,        ,   99.6,    7.8,   -5.0
GRID,        20,        ,   99.6,    7.8,    0.0
GRID,        21,        ,  105.0,    5.7,    5.0
GRID,        22,        ,  105.0,   -5.7,    5.0
GRID,        23,        ,  105.0,   -5.7,   -5.0
GRID,        24,        ,  105.0,    5.7,   -5.0
GRID,        25,        ,  110.0,    3.5,    5.0
GRID,        26,        ,  110.0,    0.0,    5.0
GRID,        27,        ,  110.0,   -3.5,    5.0
GRID,        28,        ,  110.0,   -3.5,    0.0
GRID,        29,        ,  110.0,   -3.5,   -5.0
GRID,        30,        ,  110.0,    0.0,   -5.0
GRID,        31,        ,  110.0,    3.5,   -5.0
GRID,        32,        ,  110.0,    3.5,    0.0
$
CHEXA,       1,      10,      3,      5,      7,      1,     15,     17,
     ,      19,      13,      4,      6,      8,      2,     10,     11,
     ,      12,       9,     16,     18,     20,     14
$
CHEXA,        2,      10,     15,     17,     19,     13,     27,     29,
     ,       31,      25,     16,     18,     20,     14,     22,     23,
     ,       24,      21,     28,     30,     32,     26
$
PSOLID,      10,      20
MAT1,        20,    2.+5,       ,     0.3
SPC1,        30,     456,      1,    THRU,     32
SPC1,        30,     123,      1,    THRU,      8
$    DISTRIBUTED LOAD FOR SUBCASE 1
FORCE,       41,      15,       ,  1000.,    0., 1., 0.
FORCE,       41,      16,       ,  4000.,    0., 1., 0.
FORCE,       41,      17,       ,  1000.,    0., 1., 0.
$    CONCENTRATED LOAD FOR SUBCASE 2
FORCE,       42,      15,       ,  6000.,    0., 1., 0.
ENDDATA
```

MSC.Nastran Results

The .f06 Results File

The MSC.Nastran results are shown in **Listing 13-6**.

Listing 13-6 Gear Tooth Results

```
                    THIS PROGRAM IS CONFIDENTIAL AND A TRADE SECRET OF MSC.SOFTWARE CORP.  THE RECEIPT OR
                   POSSESSION OF THIS PROGRAM DOES NOT CONVEY ANY RIGHTS TO REPRODUCE OR DISCLOSE ITS CONTENTS, OR TO
                  MANUFACTURE, USE, OR SELL ANYTHING HEREIN, IN WHOLE OR IN PART, WITHOUT THE SPECIFIC WRITTEN CONSENT
         OF MSC.SOFTWARE CORPORATION.

* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
* *                                  * *
* *           MSC.SOFTWARE           * *
* *               CORP               * *
* *                                  * *
* *      M S C . N a s t r a n       * *
* *                                  * *
* *       VERSION - 2001.0.1         * *
* *                                  * *
* *          SEP 10, 2001            * *
* *                                  * *
* *                                  * *
* *              Intel               * *
* *                                  * *
* *MODEL PentiumIII/995 (MERCED.scm)* *
* *                                  * *
* *  Windows 2000 5.0 (Build 2195)   * *
* *                                  * *
* *                                  * *
* *                                  * *
* * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * * * * * *
```

```
                                                                              SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     1

         N A S T R A N    E X E C U T I V E    C O N T R O L    D E C K    E C H O
      ID   SOLID, ELEMENT MODEL
      SOL  101
      TIME 100
      CEND
```

```
    GEAR TOOTH EXAMPLE                                                         SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     2

                                                   C A S E    C O N T R O L    D E C K    E C H O
                    CARD
                    COUNT
                   1      TITLE        = GEAR TOOTH EXAMPLE
                   2      STRESS       = ALL
                   3      SPC          = 30
                   4      SUBCASE 1
                   5         LOAD      = 41
                   6         SUBTITLE  = GEAR TOOTH UNDER 600 N/MM LINE LOAD
                   7      SUBCASE 2
                   8         LOAD      = 42
                   9         SUBTITLE  = GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
                  10      BEGIN BULK
                               INPUT BULK DATA CARD COUNT =      52
```

Listing 13-6 Gear Tooth Results (continued)

```
  GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     3

                                                        S O R T E D   B U L K   D A T A   E C H O
                CARD
                COUNT        .  1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
                   1-        CHEXA   1       10      3       5       7       1       15      17       +000001
                   2-        ++00000119      13      4       6       8       2       10      11       +000002
                   3-        ++00000212      9       16      18      20      14                       +000003
                   4-        CHEXA   2       10      15      17      19      13      27      29       +000004
                   5-        ++00000431      25      16      18      20      14      22      23       +000005
                   6-        ++00000524      21      28      30      32      26                       +000006
                   7-        FORCE   41      15              1000.   0.      1.      0.
                   8-        FORCE   41      16              4000.   0.      1.      0.
                   9-        FORCE   41      17              1000.   0.      1.      0.
                  10-        FORCE   42      15              6000.   0.      1.      0.
                  11-        GRID    1               86.5    12.7    5.0
                  12-        GRID    2               86.5    0.0     5.0
                  13-        GRID    3               86.5    -12.7   5.0
                  14-        GRID    4               86.5    -12.7   0.0
                  15-        GRID    5               86.5    -12.7   -5.0
                  16-        GRID    6               86.5    0.0     -5.0
                  17-        GRID    7               86.5    12.7    -5.0
                  18-        GRID    8               86.5    12.7    0.0
                  19-        GRID    9               93.0    8.7     5.0
                  20-        GRID    10              93.0    -8.7    5.0
                  21-        GRID    11              93.0    -8.7    -5.0
                  22-        GRID    12              93.0    8.7     -5.0
                  23-        GRID    13              99.6    7.8     5.0
                  24-        GRID    14              100.0   0.0     5.0
                  25-        GRID    15              99.6    -7.8    5.0
                  26-        GRID    16              99.6    -7.8    0.0
                  27-        GRID    17              99.6    -7.8    -5.0
                  28-        GRID    18              100.0   0.0     -5.0
                  29-        GRID    19              99.6    7.8     -5.0
                  30-        GRID    20              99.6    7.8     0.0
                  31-        GRID    21              105.0   5.7     5.0
                  32-        GRID    22              105.0   -5.7    5.0
                  33-        GRID    23              105.0   -5.7    -5.0
                  34-        GRID    24              105.0   5.7     -5.0
                  35-        GRID    25              110.0   3.5     5.0
                  36-        GRID    26              110.0   0.0     5.0
                  37-        GRID    27              110.0   -3.5    5.0
                  38-        GRID    28              110.0   -3.5    0.0
                  39-        GRID    29              110.0   -3.5    -5.0
                  40-        GRID    30              110.0   0.0     -5.0
                  41-        GRID    31              110.0   3.5     -5.0
                  42-        GRID    32              110.0   3.5     0.0
                  43-        MAT1    20      2.+5            0.3
                  44-        PSOLID  10      20
                  45-        SPC1    30      123     1       THRU    8
                  46-        SPC1    30      456     1       THRU    32
                             ENDDATA
                       TOTAL COUNT=        47
```

```
  GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     4

USER INFORMATION MESSAGE
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
 RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
                                                OLOAD    RESULTANT

                T1             T2             T3             R1             R2             R3
       1   0.0000000E+00  6.0000000E+03  0.0000000E+00  0.0000000E+00  0.0000000E+00  5.9760000E+05
       2   0.0000000E+00  6.0000000E+03  0.0000000E+00 -3.0000000E+04  0.0000000E+00  5.9760000E+05
```

```
  GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     5
                                                                                                           SUBCASE 1

*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
   LOAD SEQ. NO.             EPSILON               EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
               1          8.1898186E-16           3.3558350E+01
               2          1.3675490E-16           8.0560677E+01
```

Listing 13-6 Gear Tooth Results (continued)

```
    GEAR TOOTH EXAMPLE                                                          SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     6
       GEAR TOOTH UNDER 600 N/MM LINE LOAD
                                                                                                                        SUBCASE 1
                            S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   ( H E X A )

0                CORNER        ------CENTER AND CORNER POINT STRESSES---------       DIR.  COSINES       MEAN
  ELEMENT-ID    GRID-ID        NORMAL              SHEAR             PRINCIPAL       -A-  -B-  -C-     PRESSURE       VON MISES
         1          0GRID CS 20 GP

                CENTER  X   3.259422E+00  XY   3.291709E+01   A   3.381539E+01  LX 0.73-0.68 0.00  -1.145106E+00    5.718108E+01
                        Y  -1.645274E+00  YZ  -1.774964E-15   B  -3.220124E+01  LY 0.68 0.73 0.00
                        Z   1.821169E+00  ZX   1.720846E-15   C   1.821169E+00  LZ 0.00 0.00-1.00

                     3  X   9.586081E+01  XY   1.681114E+01   A   1.038217E+02  LX 0.94-0.33-0.01  -5.882202E+01    6.854695E+01
                        Y   3.988232E+01  YZ   2.886330E+00   B   2.943023E+01  LY 0.24 0.70-0.67
                        Z   4.072294E+01  ZX  -1.569083E+01   C   4.321412E+01  LZ-0.22-0.63-0.74

                     5  X   9.586081E+01  XY   1.681114E+01   A   1.038217E+02  LX 0.94-0.33 0.01  -5.882202E+01    6.854695E+01
                        Y   3.988232E+01  YZ  -2.886330E+00   B   2.943023E+01  LY 0.24 0.70 0.67
                        Z   4.072294E+01  ZX   1.569083E+01   C   4.321412E+01  LZ 0.22 0.63-0.74

                     7  X  -1.049563E+02  XY   3.525449E+01   A  -2.709955E+01  LX 0.43 0.90-0.06   6.616337E+01    8.762088E+01
                        Y  -4.772841E+01  YZ  -2.680735E+00   B  -1.233072E+02  LY 0.80-0.41-0.44
                        Z  -4.580541E+01  ZX  -1.339709E+01   C  -4.808340E+01  LZ-0.42 0.14-0.89

                     1  X  -1.049563E+02  XY   3.525449E+01   A  -2.709955E+01  LX 0.43 0.90 0.06   6.616337E+01    8.762088E+01
                        Y  -4.772841E+01  YZ   2.680735E+00   B  -1.233072E+02  LY 0.80-0.41 0.44
                        Z  -4.580541E+01  ZX   1.339709E+01   C  -4.808340E+01  LZ 0.42-0.14-0.89

                    15  X   3.247527E+01  XY   6.892332E+01   A   6.460156E+01  LX 0.90-0.39 0.18   2.277133E+01    1.725774E+02
                        Y  -1.019971E+02  YZ   3.051623E-01   B  -1.312817E+02  LY 0.37 0.92 0.12
                        Z   1.207883E+00  ZX   1.500595E+01   C  -1.633868E+00  LZ 0.22 0.04-0.98

                    17  X   3.247527E+01  XY   6.892332E+01   A   6.460156E+01  LX 0.90-0.39-0.18   2.277133E+01    1.725774E+02
                        Y  -1.019971E+02  YZ  -3.051623E-01   B  -1.312817E+02  LY 0.37 0.92-0.12
                        Z   1.207883E+00  ZX  -1.500595E+01   C  -1.633868E+00  LZ-0.22-0.04-0.98

                    19  X  -4.222647E+01  XY   1.067942E+01   A  -9.214685E+00  LX 0.36 0.93 0.06   2.293951E+01    3.672278E+01
                        Y  -1.417941E+01  YZ  -8.758508E-01   B  -4.735876E+01  LY 0.66-0.30 0.68
                        Z  -1.241265E+01  ZX   7.411705E+00   C  -1.224508E+01  LZ 0.66-0.20-0.73

                    13  X  -4.222647E+01  XY   1.067942E+01   A  -9.214685E+00  LX 0.36 0.93-0.06   2.293951E+01    3.672278E+01
                        Y  -1.417941E+01  YZ   8.758508E-01   B  -4.735876E+01  LY 0.66-0.30-0.68
                        Z  -1.241265E+01  ZX  -7.411705E+00   C  -1.224508E+01  LZ-0.66 0.20-0.73

         2          0GRID CS 20 GP

                CENTER  X   4.882296E+00  XY  -4.862728E-01   A   4.904797E+00  LX 1.00 0.05 0.00  -7.478473E-01    9.708314E+00
                        Y  -5.604282E+00  YZ   8.803369E-16   B  -5.626782E+00  LY-0.05 1.00 0.00
                        Z   2.965527E+00  ZX   3.146788E-15   C   2.965527E+00  LZ 0.00 0.00-1.00

                    15  X  -6.527560E-01  XY  -3.658327E+01   A   1.337905E+01  LX 0.92 0.33-0.23   3.386906E+01    1.117319E+02
                        Y  -9.447928E+01  YZ  -3.515887E+00   B  -1.073635E+02  LY-0.30 0.94 0.1
                        Z  -6.475166E+00  ZX  -6.853597E+00   C  -7.622752E+00  LZ-0.26 0.06-0.96
```

Listing 13-6 Gear Tooth Results (continued)

```
   GEAR TOOTH EXAMPLE                                                      SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     7
      GEAR TOOTH UNDER 600 N/MM LINE LOAD
                                                                                                                      SUBCASE 1
                          S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   ( H E X A )

               CORNER        ------CENTER AND CORNER POINT STRESSES---------       DIR.  COSINES       MEAN
  ELEMENT-ID    GRID-ID       NORMAL              SHEAR             PRINCIPAL       -A-  -B-  -C-     PRESSURE       VON MISES
                   17  X  -6.527560E-01  XY  -3.658327E+01   A   1.337905E+01  LX 0.92 0.33 0.23   3.386906E+01    1.117319E+02
                       Y  -9.447928E+01  YZ   3.515887E+00   B  -1.073635E+02  LY-0.30 0.94-0.14
                       Z  -6.475166E+00  ZX   6.853597E+00   C  -7.622752E+00  LZ 0.26-0.06-0.96

                   19  X  -1.064864E+01  XY   3.353643E+01   A   2.346755E+01  LX 0.69-0.69 0.20   8.570051E+00    5.989574E+01
                       Y  -1.259695E+01  YZ   5.501396E+00   B  -4.522928E+01  LY 0.68 0.72 0.14
                       Z  -2.464565E+00  ZX   3.459682E+00   C  -3.948429E+00  LZ 0.24-0.04-0.97

                   13  X  -1.064864E+01  XY   3.353643E+01   A   2.346755E+01  LX 0.69-0.69-0.20   8.570051E+00    5.989574E+01
                       Y  -1.259695E+01  YZ  -5.501396E+00   B  -4.522928E+01  LY 0.68 0.72-0.14
                       Z  -2.464565E+00  ZX  -3.459682E+00   C  -3.948429E+00  LZ-0.24 0.04-0.97

                   27  X  -4.605958E+00  XY   1.404592E+01   A   3.862373E+01  LX 0.31 0.95 0.01  -1.087944E+01    4.297141E+01
                       Y   3.401790E+01  YZ   1.107342E+00   B  -9.173705E+00  LY 0.95-0.31 0.03
                       Z   3.226375E+00  ZX   3.496277E-01   C   3.188291E+00  LZ 0.03 0.00-1.00

                   29  X  -4.605958E+00  XY   1.404592E+01   A   3.862373E+01  LX 0.31 0.95-0.01  -1.087944E+01    4.297141E+01
                       Y   3.401790E+01  YZ  -1.107342E+00   B  -9.173705E+00  LY 0.95-0.31-0.03
                       Z   3.226375E+00  ZX  -3.496277E-01   C   3.188291E+00  LZ-0.03 0.00-1.00

                   31  X  -3.306385E+01  XY  -1.294417E+01   A  -1.647081E+01  LX-0.24 0.44-0.87   3.483593E+01    4.006144E+01
                       Y  -5.363751E+01  YZ  -4.772546E+00   B  -6.081178E+01  LY-0.04 0.89 0.46
                       Z  -1.780642E+01  ZX  -4.617607E+00   C  -2.722520E+01  LZ 0.97 0.15-0.19

                   25  X  -3.306385E+01  XY  -1.294417E+01   A  -1.647081E+01  LX 0.24 0.44-0.87   3.483593E+01    4.006144E+01
                       Y  -5.363751E+01  YZ   4.772546E+00   B  -6.081178E+01  LY 0.04 0.89 0.46
                       Z  -1.780642E+01  ZX   4.617607E+00   C  -2.722520E+01  LZ 0.97-0.15 0.19
```

Listing 13-6 Gear Tooth Results (continued)

```
   GEAR TOOTH EXAMPLE                                                         SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     8
      GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
                                                                                                                     SUBCASE 2
                        S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   ( H E X A )

               CORNER        ------CENTER AND CORNER POINT STRESSES---------       DIR.  COSINES       MEAN
  ELEMENT-ID    GRID-ID       NORMAL              SHEAR             PRINCIPAL       -A-  -B-  -C-     PRESSURE       VON MISES
         1          0GRID CS 20 GP

               CENTER  X   4.571454E+00  XY   3.291783E+01   A   3.922741E+01  LX 0.69 0.70 0.20  -1.679391E+00    5.933091E+01
                       Y   7.224079E+00  YZ   5.886761E+00   B  -2.786473E+01  LY 0.72-0.69-0.07
                       Z  -6.757360E+00  ZX  -7.090083E-02   C  -6.324505E+00  LZ 0.09 0.19-0.98

                    3  X   1.985495E+01  XY   1.187322E+01   A   2.949835E+01  LX 0.86-0.44-0.26  -1.143677E+01    4.453531E+01
                       Y   6.537606E+00  YZ  -1.615572E+01   B  -1.800151E+01  LY 0.09 0.63-0.77
                       Z   7.917767E+00  ZX   1.433615E+01   C   2.281349E+01  LZ 0.50 0.64 0.58

                    5  X   1.661192E+02  XY   2.175381E+01   A   1.814674E+02  LX 0.94-0.31-0.11  -1.037578E+02    1.308156E+02
                       Y   7.332195E+01  YZ  -2.795475E+01   B   3.062253E+01  LY 0.11 0.63-0.77
                       Z   7.183234E+01  ZX   3.914438E+01   C   9.918353E+01  LZ 0.31 0.72 0.63

                    7  X  -1.487215E+02  XY   8.178657E+00   A  -4.295859E+01  LX-0.28 0.92-0.26   9.001753E+01    1.143501E+02
                       Y  -5.901127E+01  YZ   1.136970E+01   B  -1.654868E+02  LY 0.45-0.11-0.88
                       Z  -6.231984E+01  ZX  -3.968795E+01   C  -6.160725E+01  LZ 0.84 0.37 0.39

                    1  X  -6.489181E+01  XY   6.221058E+01   A   1.336910E+01  LX 0.62 0.76 0.17   4.375712E+01    1.177515E+02
                       Y  -3.608615E+01  YZ   1.234834E+01   B  -1.189491E+02  LY 0.78-0.61-0.14
                       Z  -3.029339E+01  ZX  -1.599290E+01   C  -2.569135E+01  LZ-0.01 0.22-0.97

                   15  X  -9.783761E+00  XY   2.190359E+02   A   1.531718E+02  LX 0.70-0.36 0.62   1.437844E+02    6.305996E+02
                       Y  -4.481931E+02  YZ   1.083072E+02   B  -5.499730E+02  LY 0.37 0.92 0.12
                       Z   2.662374E+01  ZX   5.490842E+01   C  -3.455194E+01  LZ 0.62-0.14-0.78

                   17  X   5.878865E+01  XY  -7.572263E+01   A   2.186662E+02  LX-0.44 0.77 0.47  -9.227718E+01    1.919590E+02
                       Y   1.483216E+02  YZ  -6.093703E+01   B   1.170096E+01  LY 0.81 0.56-0.17
                       Z   6.972133E+01  ZX   2.121105E+01   C   4.646441E+01  LZ-0.39 0.31-0.86

                   19  X  -4.857354E+01  XY   1.472993E+01   A   5.522838E+01  LX-0.18 0.80-0.58   1.066104E+01    1.095192E+02
                       Y  -1.509605E+01  YZ   3.695835E+01   B  -7.084710E+01  LY 0.43-0.47-0.77
                       Z   3.168645E+01  ZX  -2.810862E+01   C  -1.636441E+01  LZ 0.89 0.39 0.26

                   13  X  -4.758395E+01  XY   1.283193E+00   A   1.204274E+01  LX-0.61 0.23-0.76   5.867738E+01    1.085369E+02
                       Y  -1.016174E+02  YZ  -1.684200E+01   B  -1.072945E+02  LY-0.12 0.92 0.38
                       Z  -2.683081E+01  ZX  -4.637775E+01   C  -8.078040E+01  LZ 0.78 0.33-0.53

         2          0GRID CS 20 GP

               CENTER  X   4.884711E+00  XY  -4.864016E-01   A   5.807430E+00  LX-0.41-0.03-0.91  -2.043858E+00    9.684574E+00
                       Y   2.091794E+00  YZ   4.776686E+00   B  -4.381088E+00  LY 0.74-0.59-0.32
                       Z  -8.449320E-01  ZX  -3.765759E-02   C   4.705231E+00  LZ 0.53 0.80-0.27

                   15  X  -2.824835E+01  XY  -1.569827E+02   A   7.118480E+01  LX-0.41 0.35-0.84   1.191875E+02    4.959504E+02
                       Y  -3.729666E+02  YZ   8.750310E+01   B  -4.484842E+02  LY 0.31 0.92 0.23
                       Z   4.365232E+01  ZX   9.658312E+00   C   1.973675E+01  LZ 0.85-0.17-0.49
```

Listing 13-6 Gear Tooth Results (continued)

```
  GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE     9
     GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
                                                                                                                     SUBCASE 2
                        S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   ( H E X A )

             CORNER        ------CENTER AND CORNER POINT STRESSES---------       DIR.  COSINES       MEAN
 ELEMENT-ID   GRID-ID       NORMAL              SHEAR             PRINCIPAL       -A-  -B-  -C-     PRESSURE       VON MISES
                 17  X   3.985225E+01  XY   8.883585E+01   A   1.923506E+02  LX 0.45 0.70-0.56  -7.316022E+01    2.038985E+02
                     Y   1.231419E+02  YZ  -6.007481E+01   B  -4.303396E+01  LY 0.84-0.54 0.00
                     Z   5.648651E+01  ZX   2.036191E+01   C   7.016407E+01  LZ-0.30-0.47-0.83

                 19  X   2.995373E+00  XY  -4.761217E+01   A   1.047138E+02  LX-0.38 0.68-0.63  -2.687507E+01    1.271617E+02
                     Y   1.995684E+01  YZ   4.607714E+01   B  -4.112845E+01  LY 0.60 0.70 0.39
                     Z   5.767300E+01  ZX  -1.472099E+01   C   1.703989E+01  LZ 0.70-0.22-0.67

                 13  X  -3.080114E+01  XY   1.097670E+02   A   4.568690E+01  LX 0.81-0.55-0.21   6.197749E+01    2.154415E+02
                     Y  -1.260158E+02  YZ  -1.312900E+01   B  -1.981381E+02  LY 0.54 0.84-0.11
                     Z  -2.911551E+01  ZX  -1.319591E+01   C  -3.348130E+01  LZ-0.24 0.02-0.97

                 27  X   1.384408E+01  XY   6.899274E+01   A   2.069963E+02  LX 0.34 0.06-0.94  -5.168850E+01    2.342441E+02
                     Y   1.735428E+02  YZ  -4.039832E+01   B  -4.009761E+01  LY 0.92 0.17 0.35
                     Z  -3.232135E+01  ZX  -1.512994E+01   C  -1.183315E+01  LZ-0.18 0.98 0.00

                 29  X   4.168195E+01  XY  -3.819234E+01   A   8.848639E+01  LX-0.62 0.35-0.70  -2.957272E+01    9.502836E+01
                     Y   3.132346E+01  YZ   3.619569E+01   B  -2.005870E+01  LY 0.67 0.70-0.25
                     Z   1.571276E+01  ZX  -8.393348E+00   C   2.029048E+01  LZ 0.41-0.63-0.67

                 31  X   8.506835E+01  XY   1.017675E+01   A   1.344014E+02  LX 0.19-0.08-0.98  -8.568668E+01    8.255613E+01
                     Y   1.283673E+02  YZ  -1.923362E+01   B   3.914836E+01  LY 0.96 0.22 0.17
                     Z   4.362441E+01  ZX   1.647656E+00   C   8.351028E+01  LZ-0.20 0.97-0.12

                 25  X  -1.289453E+02  XY  -3.887628E+01   A  -4.716179E+01  LX 0.29 0.76 0.58   1.001211E+02    1.037033E+02
                     Y  -1.184131E+02  YZ   1.273300E+00   B  -1.650881E+02  LY-0.14 0.64-0.76
                     Z  -5.300490E+01  ZX   1.947105E+01   C  -8.811346E+01  LZ 0.95-0.14-0.29
```

```
 GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    10
```

```
  GEAR TOOTH EXAMPLE                                                   SEPTEMBER  10, 2001  MSC.NASTRAN   9/10/01   PAGE    11


* * * *  D B D I C T   P R I N T  * * * *      SUBDMAP = PRTSUM  ,  DMAP STATEMENT NO.     13

                                  * * * *  A N A L Y S I S  S U M M A R Y  T A B L E  * * * *

SEID  PEID PROJ VERS APRCH      SEMG SEMR SEKR SELG SELR MODES DYNRED SOLLIN PVALID SOLNL LOOPID DESIGN CYCLE SENSITIVITY
-------------------------------------------------------------------------------------------------------------------------
    0     0    1    1 '        '    T    T    T    T    T     F      F      T      0     F     -1            0           F

SEID = SUPERELEMENT ID.
PEID = PRIMARY SUPERELEMENT ID OF IMAGE SUPERELEMENT.
PROJ = PROJECT ID NUMBER.
VERS = VERSION ID.
APRCH = BLANK FOR STRUCTURAL ANALYSIS.  HEAT FOR HEAT TRANSFER ANALYSIS.
SEMG = STIFFNESS AND MASS MATRIX GENERATION STEP.
SEMR = MASS MATRIX REDUCTION STEP (INCLUDES EIGENVALUE SOLUTION FOR MODES).
SEKR = STIFFNESS MATRIX REDUCTION STEP.
SELG = LOAD MATRIX GENERATION STEP.
SELR = LOAD MATRIX REDUCTION STEP.
MODES = T (TRUE) IF NORMAL MODES OR BUCKLING MODES CALCULATED.
DYNRED = T (TRUE) MEANS GENERALIZED DYNAMIC AND/OR COMPONENT MODE REDUCTION PERFORMED.
SOLLIN = T (TRUE) IF LINEAR SOLUTION EXISTS IN DATABASE.
PVALID = P-DISTRIBUTION ID OF P-VALUE FOR P-ELEMENTS
LOOPID = THE LAST LOOPID VALUE USED IN THE NONLINEAR ANALYSIS.  USEFUL FOR RESTARTS.
SOLNL = T (TRUE) IF NONLINEAR SOLUTION EXISTS IN DATABASE.
DESIGN CYCLE = THE LAST DESIGN CYCLE (ONLY VALID IN OPTIMIZATION).
SENSITIVITY = SENSITIVITY MATRIX GENERATION FLAG.

                                        * * * END OF JOB * * *
```

Stress Results

First we examine the output for error or warning messages-none are present-and find epsilon, which is reported for each subcase (page 5 of the output). Epsilon is very small in both cases.

CHEXA stress results are reported at each element's center and corner grid points. Stresses at midside grid points are not available. For gear teeth in alignment (subcase 1), the peak von Mises stress is 1.73E2 MPa at grid points 15 and 17 of CHEXA element 1 (page 6 of the output in **Listing 13-6**). For misaligned gear teeth (subcase 2), the peak von Mises stress is 6.31E2 MPa at grid point 15 (page 8 of the output).

Observe that for both subcases the von Mises stresses at grid points shared by two adjacent elements differ. Solid element stresses are calculated inside the element and are interpolated in toward the element's center and extrapolated outward to its corners. The numerical discrepancy between shared grid points is due to interpolation and extrapolation differences between adjoining elements in regions where high stress gradients exists (which is often the case in a model with an inadequate number of elements). This discrepancy between neighboring element stresses can be reduced by refining the element mesh.

Note also that solid elements result in a considerable volume of printed output. If printed output is desired for larger solid element models, you may want to be somewhat selective in requesting output using the Case Control Section of the input file.

CHAPTER

14 Overview of Other MSC.Nastran Capabilities

- Techniques for Analyzing Large Models
- Dynamic Analysis
- Nonlinear Analysis
- Design Sensitivity and Optimization
- Aeroelasticity
- Thermal Analysis
- Composite Materials
- Fluid-Structure Interaction

14.1 Techniques for Analyzing Large Models

MSC.Nastran offers many capabilities to aid analysts in solving large and complex finite element models. These techniques include the following:

- Efficient solution of large matrix problems
- Superelement analysis
- Component mode synthesis
- Cyclic symmetry analysis
- Axisymmetric analysis

Efficient Solution of Large Matrix Problems

MSC.Nastran can solve equations of virtually unlimited size-problems have been solved in excess of 1,000,000 degrees of freedom. MSC.Nastran solves large systems of equations using sparse matrix algorithms, resulting in faster solutions and reduced computer storage space. MSC.Nastran also takes advantage of vector processing and parallel processing on computers that support these features, resulting in faster computations.

In addition to sparse matrix algorithms, MSC.Nastran offers numerical algorithms to handle other types of matrices. In general, the choice of algorithms is made automatically by the program, which selects the fastest method for solving a given equation based on matrix density, computer speed, and available computer memory.

Superelement Analysis

Superelement analysis is a special feature of MSC.Nastran that can be used to reduce the complexity and resource demands of large analysis problems. MSC.Nastran's superelement capabilities can be used in all types of analyses, including statics, normal modes, buckling, transient response, frequency response, heat transfer, and nonlinear analysis.

Using the superelement technique involves breaking down a large structure into a set of smaller substructures called superelements. Superelements may be processed individually or all at once. The final solution, in which all of the individual superelement solutions are combined, involves much smaller matrices than would be required if the entire model were solved in one run. Superelement analysis has the advantage of reducing computer resource requirements, especially if changes are made to only one portion (superelement) of the model-only the affected superelement need be reanalyzed.

Superelement analysis offers procedural advantages as well, particularly when multiple engineering firms are involved in an analysis. Imagine a model of a rocket and satellite payload: one firm models the space booster, another firm models the engines, and another models the satellite. Each firm can reduce its model to so-called boundary degrees of freedom suitable for superelement analysis. The primary analysis firm then combines these reduced models into one model for a liftoff analysis. Superelement analysis has the advantage that matrices are passed from one organization to another without revealing proprietary modeling details and without concern about whether the same grid point and element numbers were used by every participant.

An example of superelement analysis applied to a car door assembly is shown in **Figure 14-1**:

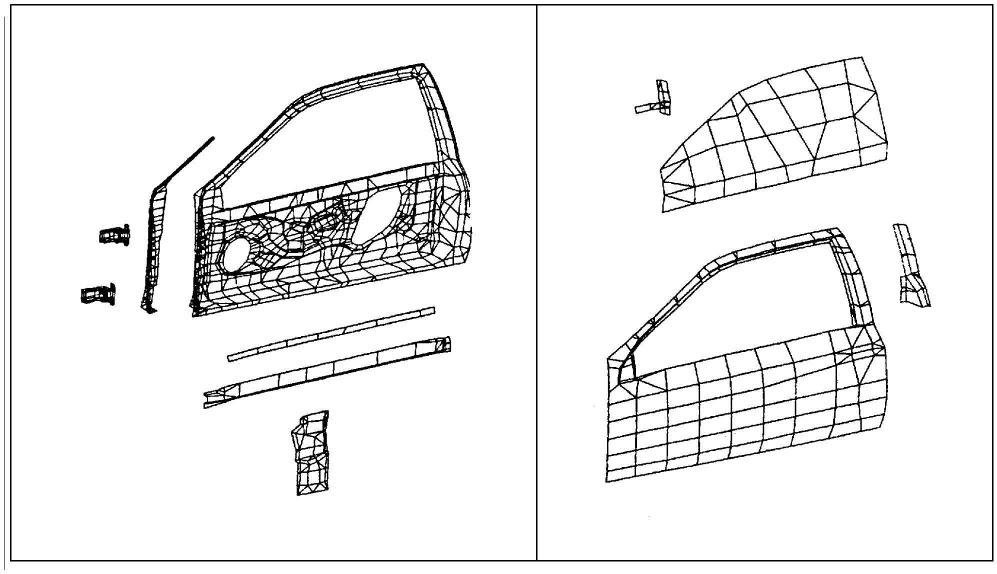
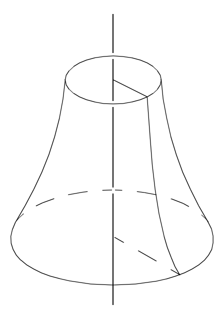


Figure 14-1 Superelements Used to Model a Car Door

As finite element models become larger and more complicated, engineers must take advantage of all tools available to them. Superelements are one of the most powerful tools available for the analysis of large problems. By using sound engineering judgement to develop superelement models, MSC.Nastran engineers have been able to obtain performance improvements as large as 29:1 compared to conventional analysis.

Axisymmetric Analysis

Pressure vessels and other similar containers are often axisymmetric and can be treated as shells of revolution. MSC.Nastran offers several axisymmetric elements for modeling these types of structures. Using axisymmetric elements, only a small portion of the structure need be modeled, saving considerable engineering time and computer cost.

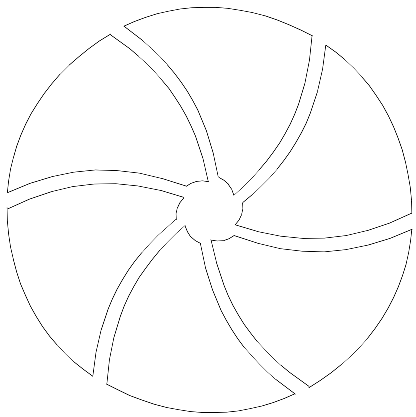


Axisymmetric Structure

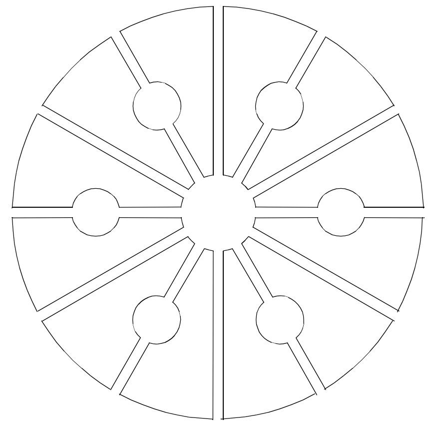
Cyclic Symmetry Analysis

Many structures, including rotating machines and antennas for space structures, are composed of virtually identical segments that are symmetrically oriented with respect to an axis. These kinds of structures can be analyzed using cyclic symmetry analysis, which allows the analyst to model only one of the identical segments and yet obtain results for the entire model.

MSC.Nastran can analyze structures with rotational symmetry and with dihedral symmetry, as shown in the following figure. Cyclic symmetry analysis can be performed for statics, normal modes, buckling, and frequency response analysis.



Rotational Symmetry



Dihedral Symmetry

14.2 Dynamic Analysis

Dynamic analysis involves loads and responses that vary with time. Common forms of dynamic excitation include earthquakes, vibrations due to rotating machinery, and structural impact. MSC.Nastran offers a wide variety of capabilities for dynamic analysis; these are summarized as follows:

Normal Modes Analysis

Normal modes analysis determines the natural frequencies and mode shapes for a structure. The natural frequencies of a structure are the frequencies at which the structure will naturally tend to vibrate if subjected to a disturbance. For example, the strings of a piano are each tuned to vibrate at a specific frequency. Other terms for natural frequency include characteristic frequency, fundamental frequency, resonance frequency, and normal frequency. The deformed shape of a structure when vibrating at a specific natural frequency is called its normal mode of vibration, or mode shape. Each mode shape is associated with a specific natural frequency. Finite element model plots of the first four mode shapes of a cantilever beam are shown in **Figure 14-2**.

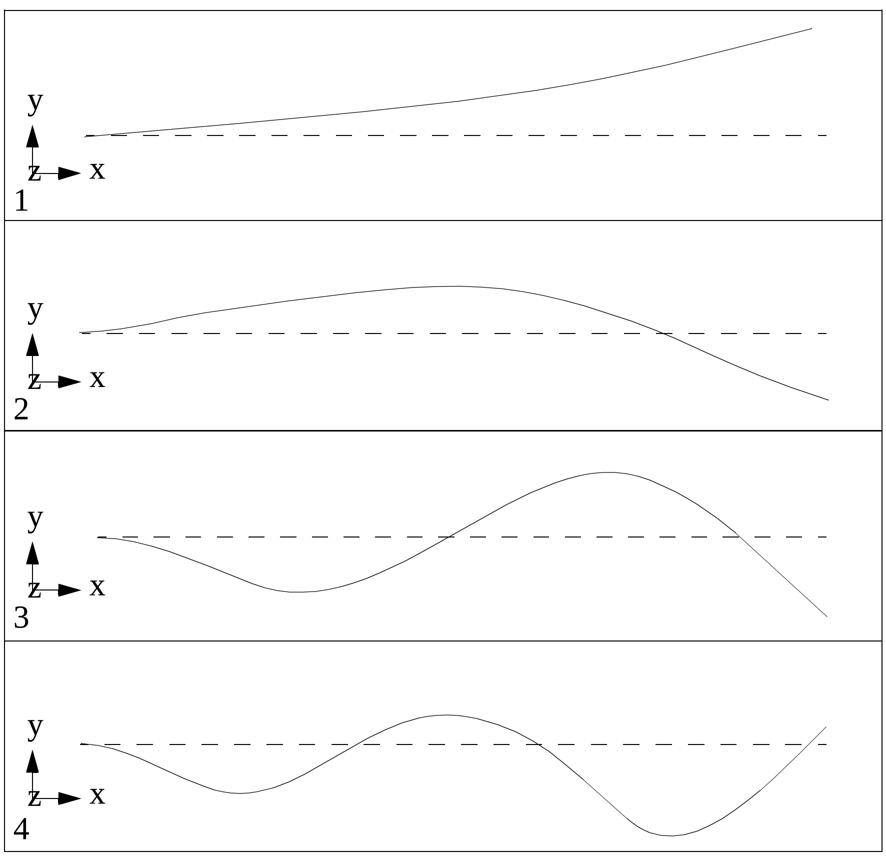


Figure 14-2 The First Four Mode Shapes of a Cantilever Beam

Frequency Response Analysis

Frequency response analysis is a method to compute structural response to steady-state oscillatory excitation (for example, a piece of equipment which rotates at a set frequency but has an imbalance). In frequency response analysis, the excitation is explicitly defined in the frequency domain, and all of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and / or enforced motions (displacements, velocities, or accelerations).

The results obtained from a frequency response analysis usually include grid point displacements, grid point accelerations, and element forces and stresses. The computed responses are complex (number) magnitude and phase (with respect to the applied force) or as real and imaginary components. For example, **Figure 14-3** shows the displacement magnitude of a grid point vs. frequency.

Two different numerical methods can be used in frequency response analysis. Direct frequency response analysis solves the coupled equations of motion in terms of forcing frequency. Modal frequency response analysis uses the normal modes of the structure to uncouple the equations of motion, with the solution for a particular forcing frequency obtained through the summation of the individual modal responses.

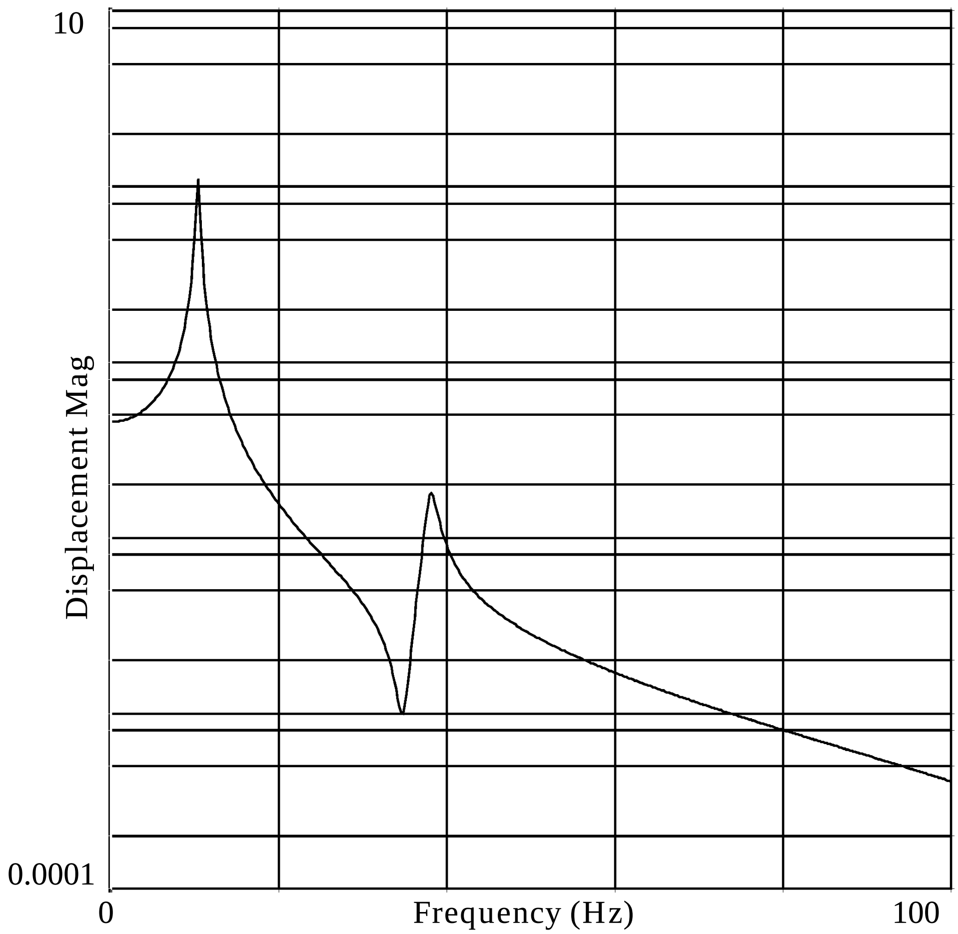


Figure 14-3 Displacement Magnitude from Frequency Response Analysis (Logrithmic Scale)

Transient Response Analysis

Transient response analysis is the most general method for computing forced dynamic response. The purpose of a transient response analysis is to compute the behavior of a structure subjected to time-varying excitation. The transient excitation is explicitly defined in the time domain. All of the forces applied to the structure are known at each instant in time. Forces can be in the form of applied forces and/or enforced motions. The results obtained from a transient analysis are typically displacements and accelerations of grid points, and forces and stresses in elements. The displacement response of a grid point versus time is shown in **Figure 14-4**.

Depending upon the structure and the nature of the loading, two different numerical methods can be used for a transient response analysis: direct or modal. Direct transient response analysis performs a numerical integration of the complete coupled equations of motion. Modal transient response analysis uses the normal modes of the structure to uncouple the equations of motion with the solution obtained through the summation of the individual modal responses.

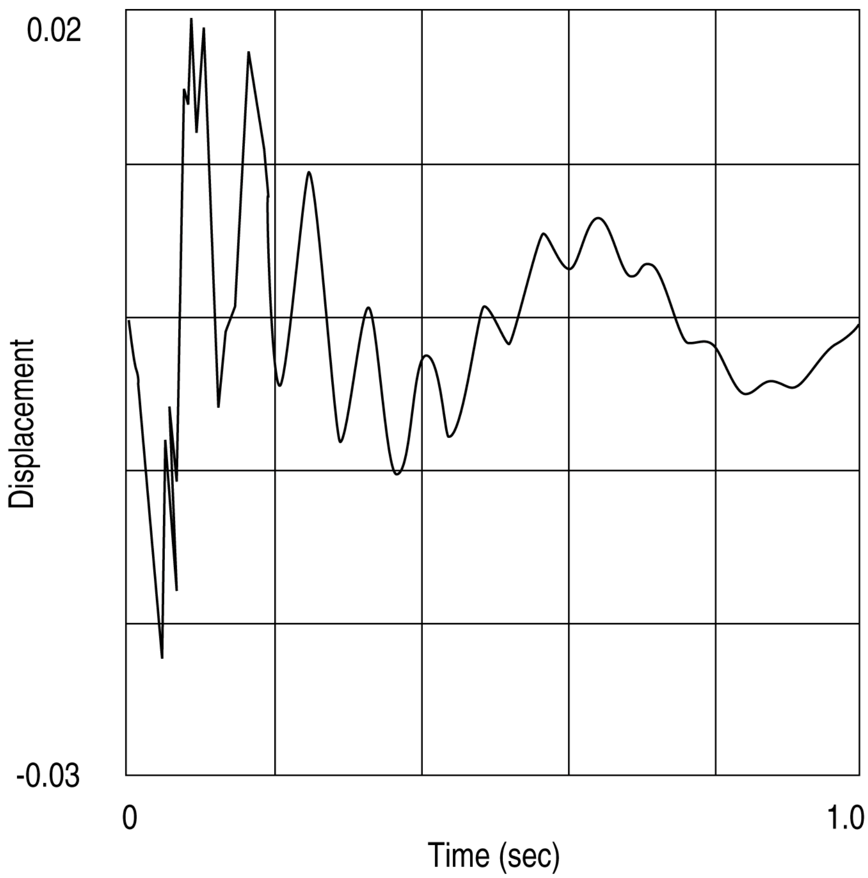


Figure 14-4 Displacement Response from Transient Response Analysis

Complex Eigenanalysis

Complex eigenanalysis is used to compute the damped modes of structures and to assess the stability of systems modeled with transfer functions (including servomechanisms and rotating systems).

The governing equation is similar to that for normal modes analysis except that damping has been added. In addition, the mass and stiffness matrices may be unsymmetric and they may contain complex elements.

Response Spectrum Analysis

Response spectrum analysis, sometimes known as shock spectrum analysis, provides an alternative to more lengthy dynamic analysis by transient response methods. A response spectrum represents the peak (maximum) response of a set of single-degree-of-freedom oscillators. The independent variable of a response spectrum is natural frequency, with each frequency corresponding to a single oscillator (each oscillator has a single unique natural frequency). So, a single curve of a response spectrum is a plot of peak response (e.g. displacement) as a function of oscillator natural frequency. The curve corresponds to a single value of oscillator damping; each oscillator has the same damping.

From a response spectrum, assuming the structure behaves linearly, it is possible to determine the peak response of each structural mode exactly. Having done that it is possible to combine the mode responses approximately (not exactly because this is not a transient analysis) using such methods as SRSS, sum of absolute values, or NRL.

In addition to response spectrum analysis, MSC.Nastran can also compute the following types of response spectra: absolute and relative displacement, absolute and relative velocity, and absolute acceleration.

Component Mode Synthesis

Component mode synthesis (CMS) is a form of superelement reduction wherein matrices are cast in terms of modal coordinates (resonant frequencies, normal modes, modal mass, stiffness, and damping) in addition to physical coordinates. CMS has the advantage that there are fewer modal coordinates than physical coordinates-perhaps only one-tenth as many. In addition, CMS can utilize modal test data, thereby increasing the accuracy of the overall analysis.

Random Vibration Analysis

Random vibration is vibration that can be described in only a statistical sense. The instantaneous value (e.g. $u_i(t)$) is not known at any given time. Rather, the magnitude is expressed in terms of statistical properties such as mean value, standard deviation, and probability of exceeding a certain value.

Examples of random vibration include earthquake ground motion, ocean wave heights and frequencies, wind pressure fluctuations on aircraft and tall buildings, and acoustic excitation due to rocket and jet engine noise. These random excitations are usually described in terms of a power spectral density (PSD) function.

Random response output consists of the response power spectral density, autocorrelation functions, number of zero crossings with positive slope per unit time, and the RMS (root mean square) values of response.

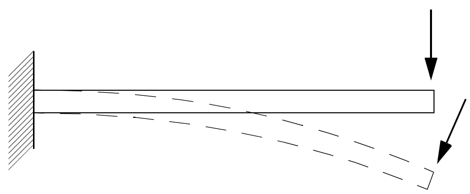
14.3 Nonlinear Analysis

Linear static problems are solved in one step-a single decomposition of the stiffness matrix. Nonlinear problems of any type require iterative solution methods and incremental loading to obtain (converge to) a solution, and are generally far more computationally intensive than linear problems. Nonlinear problems are classified into two broad categories: geometric nonlinearity and material nonlinearity. MSC.Nastran can also handle nonlinear problems that are time dependent (transient).

Geometric Nonlinearity

Geometrically nonlinear problems involve large displacements; “large” means that the displacements invalidate the small displacement assumptions inherent in the equations of linear analysis. For example, consider a classical thin plate subject to a lateral load; if the deflection of the plate’s midplane is anything close to the thickness of the plate, then the displacement is considered large and a linear analysis is not applicable.

Another aspect of geometric nonlinear analysis involves follower forces. Consider a slender cantilever beam subject to an initially vertical end load. The load is sufficient to cause large displacements.



In the deformed shape plot, the load is no longer vertical-it has “followed” the structure to its deformed state. Capturing this behavior requires the iterative update techniques of nonlinear analysis.

Material Nonlinearity

Recall that linear analysis assumes a linear relationship between stress and strain.

Material nonlinear analysis solution sequences can be used to analyze problems in static analysis where the stress-strain relationship of the material is nonlinear. In addition, moderately large strain values can be analyzed. Examples of material nonlinearities include metal plasticity, materials such as soils and concrete, or rubbery materials (where the stress-strain relationship is nonlinear elastic). Various plasticity theories such as von Mises or Tresca (for metals), and Mohr-Coulomb or Drucker-Prager (for frictional materials such as soils or concrete) can be selected by the user. Three choices for the definition of subsequent yield surfaces are available in MSC.Nastran. They are isotropic hardening, kinematic hardening, or combined isotropic and kinematic hardening. With such generality, most plastic material behavior, with or without the Bauschinger effect, can be modeled. In addition, gaps can be used to model effects due to structural separation or contact, and hyperelastic large strain analysis is also available.

Nonlinear Transient Analysis

Nonlinear transient analysis solves for the response of structures in which the stress-strain relationship of the material is nonlinear and/or the strain-displacement relationship is nonlinear. In addition, the structure is subjected to time-varying loads. Damping in the system can be either viscous or structural. Creep effects may also be analyzed.

14.4 Design Sensitivity and Optimization

Overview

Design sensitivity and optimization are used when we modify a design whose level of structural complexity exceeds our ability to make appropriate design changes. What is surprising is that an extremely simple design task may easily surpass our decision-making abilities. Experienced designers, perhaps with decades of experience, are often adept at poring through great quantities of data and coming up with improved designs. Most of us, however, cannot draw upon such intuition and experience. The goal of design optimization is to automate the design process by using a rational, mathematical approach to yield improved designs. Possible applications include:

- Reducing the weight of a structure.
- Producing more efficient designs having maximum margins of safety.
- Performing trade-off or feasibility studies.
- Assisting in design sensitivity studies.
- Correlating test data and analysis results.

Design sensitivity and optimization are closely related topics. In fact, the class of optimizers used in MSC.Nastran require sensitivity analysis results to perform design optimization. An optimizer is nothing more than a formal algorithm, or plan, for searching for a "best" configuration. Defining what characteristics this configuration might possess is part of the challenge for the design engineer. Sensitivity analysis is the procedure that determines the changes in some structural response quantity for an infinitesimal change in a design parameter. These derivatives, called sensitivity coefficients, may yield useful design information alone, but are of greatest utility if used by a numerical optimization algorithm to suggest possible design alternatives.

MSC.Nastran Capabilities

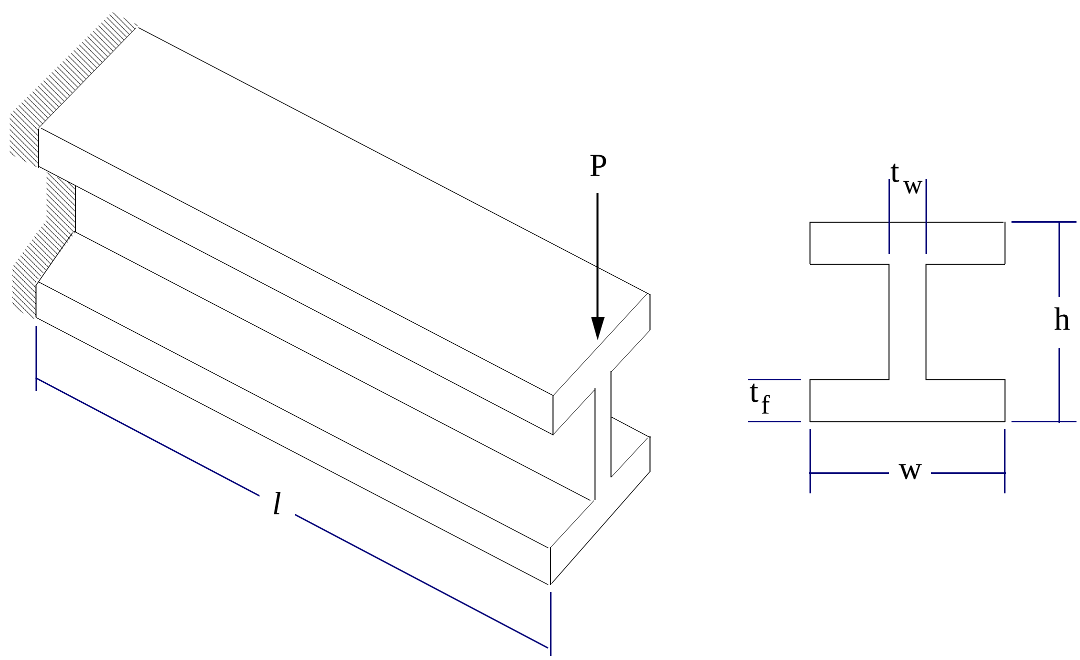
Available analysis disciplines include statics, normal modes, buckling, direct and modal frequency response, and modal transient response. Possible analysis response types include the following:

- weight
- volume
- eigenvalues
- buckling load factor
- displacement
- stress
- strain
- element forces
- composite stress, strain and failure criterion

- frequency response displacement, velocity, acceleration, stress, and force
- transient response displacement, velocity, acceleration, stress, and force

Application Examples

As a simple example of design optimization, consider the cantilevered I-beam shown below.



A frequent task of the designer is to minimize the total structural weight (thereby reducing the cost of materials) while ensuring that the design still satisfies all of the performance-related constraints. The structural weight is termed the design objective function, and the optimizer will be used to find a minimum of this function. With the tip load shown in the figure, one constraint on the design might be transverse displacement at the tip, while other constraints might be placed on natural frequencies. These performance constraints are called design constraints, since they place limits on the optimizer's ability to reduce the weight of the structure.

In order to reduce the weight of this structure, the designer might allow the cross-sectional dimensions to vary. These design variables may include any or all of the width w, height h, thickness of the web t_w, and thickness of the flanges t_f. However, for analysis, the corresponding cross-sectional properties (A, I1, I2, and so on) must be determined. Thus, it is up to the designer to specify the dependence of these properties on the chosen set of design variables. This process, in addition to specification of the design objective and constraints, is known as design modeling. The design model is simply a statement of the objective, constraints, and admissible variations which can be made to the structure in pursuit of this goal.

With both a design and analysis model at hand, the optimizer can investigate changes in the analysis model responses for finite changes in the design variables. In the process of searching for an optimum, the optimizer utilizes the results of the finite element analysis as well as structural response derivatives, or sensitivities, computed by the program. The resultant solution, or optimal design, is in this case that combination of design variable values which minimize the weight of the structure while satisfying the constraints on tip displacement and natural frequencies.

It is unlikely that the resulting optimal I-beam cross sectional dimensions would be an exact match to a readily available manufactured beam. So, the final stage of design usually consists of using the design optimization results to assist in choosing appropriate sizes of I-beams from a list of available sections, followed by reanalysis / redesign to verify these selections. In practice, optimization is most useful as a design tool to suggest possible design improvements, rather than as a “solution” technique.

The following examples illustrate some other possible applications of design optimization:

- A complex spacecraft is in a conceptual design stage. The total weight of the spacecraft cannot exceed 3,000 pounds. The nonstructural equipment including the payload is 2,000 pounds. Static loads are prescribed based on the maximum acceleration at launch. Also, the flexibility requirement in space requires that the fundamental elastic frequency must be above 12 Hz. It is crucial to reduce the structural weight since it costs several thousand dollars to place one pound of mass in a low earth orbit. There are three proposed designs: truss, frame, and stiffened shell configurations. Currently all the designs fail to satisfy at least one design requirement and are expected to be overweight. At this stage, we want to determine which configuration(s) promises the best performance and warrants detailed design studies. Also, the payload manager needs to know how much weight could be saved if the frequency requirement were relaxed from 12 Hz to 10 Hz. The spacecraft’s structure contains about 150 structural parameters which we may want to vary simultaneously.
- One part of a vehicle’s frame structure is found to be overstressed. Unfortunately, it is too expensive to redesign that particular frame component at this stage in the engineering cycle. However, other structural components nearby can be modified without severe cost increases. There are nearly 100 structural design parameters that can be manipulated. The design goal is to reduce the magnitude of the stresses by reducing the internal load to the overstressed member.
- A frame structure which supports a set of sensitive instruments must withstand severe in-service dynamic loads. Modal test results are available from comprehensive tests performed on the prototype structure. We would like to create a finite element model for dynamic analysis that is much less detailed than the original model created for stress analysis since the costs of dynamic analysis using a complex model would be prohibitive. However, we want to ensure that the first ten modes obtained from our simplified model are in close agreement with those obtained from the test results. The goal is to determine suitable properties for the lumped quantities in our simplified dynamic model such that the first ten eigenvalues correlate well with the prototype.

14.5 Aeroelasticity

MSC.Nastran provides efficient solutions of the problems of aeroelasticity, which is a branch of applied mechanics that deals with the interaction of aerodynamic, inertial, and structural forces. Aeroelasticity is important in the design of airplanes, helicopters, missiles, suspension bridges, and even tall chimneys and power lines.

The primary concerns of aeroelasticity include flying qualities (that is, stability and control), flutter, and structural loads arising from maneuvers and atmospheric turbulence. Methods of aeroelastic analysis differ according to the time dependence of the inertial and aerodynamic forces that are involved. For the analysis of flying qualities and maneuvering loads wherein the aerodynamic loads vary relatively slowly, quasi-static methods are applicable. The remaining problems are dynamic, and methods of analysis differ according to whether the time dependence is arbitrary (that is, transient or random) or simply oscillatory in the steady state.

MSC.Nastran considers three classes of aeroelastic problems:

- Static Aeroelastic Response
- Aerodynamic Flutter
- Aeroelastic Response

Each is summarized as follows.

Static Aeroelastic Response

For the analyses of flying qualities and maneuvering loads, the assumption of quasi-steady motion is valid; i.e., the dynamics of the flexible structure are neglected and quasi-static methods are applicable. By assuming linear behavior of the aerodynamic, inertial, and structural forces during the motion, the equations of equilibrium in quasi-steady flight are solved in closed algebraic (matrix) form. The aeroelastic stability and control characteristics are found along with the external load distributions and the internal loads and stresses in the structural elements.

The static aeroelastic analysis solves for the trim condition in a maneuver that is assumed to be quasi-steady so that dynamic structural effects are ignored. The formulation of the equilibrium equations provides the aerodynamic stability and control derivatives as an integral part of the trim process. The external flight loads and the corresponding internal loads and stresses in the finite elements are available as postprocessing operations on the trim solution.

Aerodynamic Flutter

Flutter is the oscillatory aeroelastic instability that occurs at some airspeed at which energy extracted from the airstream during a period of oscillation is exactly dissipated by the hysteretic damping of the structure. The motion is divergent in a range of speeds above the flutter speed. Flutter analysis utilizes complex eigenvalue analysis to determine the combination of airspeed and frequency for which the neutrally damped motion is sustained.

Three methods of flutter analysis are provided: the American flutter method (called the K-method in MSC.Nastran), an efficient K-method (called the KE-method) for rapid flutter evaluations, and the British flutter method (called the PK-method) for more realistic representation of the unsteady aerodynamic loads as frequency dependent stiffness and damping terms. The complex eigenvalue analysis is specified by the user with the K-method, and the QR-transformation method is used with the KE- and PK-methods. Linear and/or surface splines may be used to connect the aerodynamic and structural grid points.

Dynamic Aeroelastic Response

The aeroelastic dynamic response problem is one of determining the response of the aircraft to time-varying excitations. Atmospheric turbulence is the primary example of this type of excitation, but store ejection loads and landing gear impact can also have an aeroelastic component. Methods of generalized harmonic (Fourier) analysis are applied to the linear system to obtain the response to the excitation. The turbulence model may be regarded either as a stationary random loading or as a discrete gust.

The gust analysis capability computes response to random atmospheric turbulence and to discrete one-dimensional gust fields. The random response parameters calculated are the power spectral density, root mean square response, and mean frequency of zero-crossings. The response to the discrete gust is calculated by direct and inverse Fourier transform methods since the oscillatory aerodynamics are only known in the frequency domain. Time histories of response quantities are the output in the discrete case.

Aerodynamic Methods

Five oscillatory aerodynamic theories are available for flutter analysis. There are two subsonic methods, Doublet-Lattice Method with body interference and Strip Theory, as well as three supersonic methods: the Mach Box Method, Piston Theory, and a new (Aero II option) multiple interfering surface method called ZONA51. The Static and Dynamic Aeroelastic Response solutions use both the subsonic Doublet-Lattice and supersonic ZONA51 aerodynamics methods.

14.6 Thermal Analysis

Thermal analysis attempts to predict the temperatures and heat flows in and around a structure. The results of a thermal analysis are often used in subsequent analysis to determine a structure's thermally induced response. MSC.Nastran provides for convenient quasi-static coupling between thermal and structural analyses. Heat transfer problems are categorized as linear or nonlinear, and steady-state or transient. In addition, there are three modes of heat transfer: conduction, convection, and radiation. Each mode displays its own character and each can be analyzed with MSC.Nastran.

Conduction deals with the flow of heat in the interior of a body obeying Fourier's Law. For steady-state analysis, a material's thermal conductivity determines the relative ease with which heat can flow throughout the body. In transient analysis, the material's ability to store energy is also taken into consideration with specific heat or heat capacitance. In addition, a phase change, largely an isothermal process, can be involved in a large release or absorption of thermal energy in the interior of a body.

MSC.Nastran can analyze free and forced convection behavior wherein thermal communication between a conduction-dominated solid structure and its surroundings occurs. Convection boundary conditions provide for heat transfer between the structure and the surrounding environment due to fluid motion. Free convection heat transfer involves fluid motion adjacent to a surface as a result of local density variations in that fluid. This buoyancy-generated fluid motion is described by a free convection heat transfer coefficient. Forced convection heat transfer is based upon an understanding of the externally driven fluid flow field. Problems in which heat transfer coefficients are functions of temperature are nonlinear.

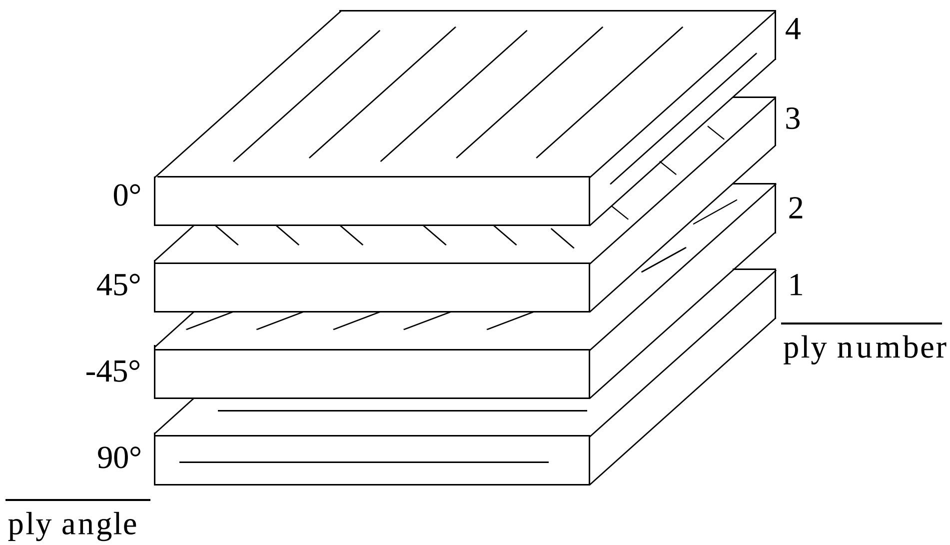
Radiation heat transfer can be analyzed within enclosures and as radiative communication to space. In radiation exchange, no transport media is required for heat transfer to occur. All radiation exchange problems are nonlinear.

Other types of boundary conditions include surface heat flux loads, temperature-specified boundary conditions, and internal heat generation loads.

Transient thermal analysis deals with the study of systems in transition, typically from one operating state point to another. In transient analysis, we solve for the thermal response due to time varying fluctuations in boundary conditions, or the relaxation of a set of initial conditions as the body heads toward a steady-state or equilibrium temperature distribution.

14.7 Composite Materials

The term composite refers to an engineering material that is made up of more than one material. Classical lamination theory takes the term composite another step to mean a material that is composed of stacks of layers or plies, each ply having its own orthotropic properties. These plies are oriented at angles one to another as shown in the four ply example. When the plies are glued together, they act as one material. The ability to orient the fibers in particular directions allows designers to tailor the mechanical properties of a composite material to match the loading environment.



Plies are typically made of fibers glued in place by a matrix. If the ply is a tape, all the fibers are oriented in the same direction. Cloth plies have fibers woven in two directions. Many different materials can be used as fibers or matrices. Examples of fibers include graphite, glass, boron, silicon carbide, and tungsten. Examples of matrices include epoxy and aluminum.

MSC.Nastran plate elements can have composite properties. The element entries reference a composite property entry that defines the stacking order, thickness, angle, and material of each ply. Several composite failure theories are available in MSC.Nastran (Tsai-Wu, Hoffmann, Hill, and maximum strain) to predict ply failures.

14.8 Fluid-Structure Interaction

MSC.Nastran can perform fully coupled fluid-structure analysis. Its principal application is in the area of acoustic and noise control analysis; for example, in the design of passenger compartments of automobiles and aircraft.

The approach used in this analysis is called the Pressure Method, which is analogous to the Displacement Method in structural analysis except that pressures, instead of displacements, are computed at the fluid points. The velocities and accelerations of the fluid points are analogous to forces in structural analysis.

The fluid may be modeled with existing three-dimensional elements: CHEXA, CPENTA, and CTETRA. These elements can assume the properties of irrotational and compressible fluids suitable for acoustic analysis or other types of analyses governed by the three-dimensional wave equation. In addition, two new elements are available for acoustic analysis, the barrier (CHACBR) and absorber (CHACAB). The absorber and barrier elements may be used to analyze acoustic noise control devices.

The interface between the fluid and the structure may be modeled so that the grid points of the fluid are coincident with those of the structure. This is called a matching mesh. If not, then it is called a nonmatching mesh. In either case and by default, coupling for the stiffness and mass is automatically computed.

Coupled fluid-structural analysis is available in the dynamic solution sequences using the Direct Method (SOLs 107 through 109), and the Modal Method (SOLs 110 through 112). It should be noted that in SOLs 110 through 112, the normal modes are computed separately for the fluid and structural parts of the model; in other words, the uncoupled modes of the fluid and structure are used in the modal formulation of the stiffness, mass, and damping.

It is sometimes useful to find the contribution of a set of grid points to the noise level in an acoustic cavity. For example, in an automobile the entire roof assembly contributes significantly to the noise level inside the passenger cabin; as such it is necessary to compute and minimize the contribution of this “panel.” A “panel” may be defined with the PANEL entry, which references a set of structural grid points.

APPENDIX

A

Summary of Basic Case Control Commands

The Case Control commands described in this book are summarized as follows:

$	Comment Entry
DISPLACEMENT	Displacement Output Request
ECHO	Bulk Data Echo Request
FORCE	Element Force-Output Request
LABEL	Output Label
LOAD	External Static Load Set Selection
OLOAD	Applied Load Output Request
SET	Set Definition
SPC	Single Point Constraint Set Selection
SPCF	Single Point Forces of Constraint Output Request
STRESS	Element Stress Output Request
SUBCASE	Subcase Delimiter
SUBTITLE	Output Subtitle
TITLE	Output Title

Full descriptions of these and other Case Control commands can be found in the "**Case Control Commands**" on page 175 of the *MSC.Nastran Quick Reference Guide.*

APPENDIX

B

Summary of Basic Bulk Data Entries

The Bulk Data entries described in this book are summarized as follows:

$	Comment Entry
CBAR	Simple Beam Element Connection
CBEAM	Beam connection with offset shear center
CELAS1	Scalar Spring Connection
CELAS2	Scalar Spring Property and Connection
CHEXA	Six-Sided Solid Element Connection
CONM2	Concentrated Mass Element Connection, Rigid Body Form
CONROD	Rod Element Property and Connection
CORD1C	Cylindrical Coordinate System Definition, Form 1
CORD1R	Rectangular Coordinate System Definition, Form 1
CORD1S	Shperical Coordinate System Definition, Form 1
CORD2C	Cylindrical Coordinate System Definition, Form 2
CORD2R	Rectangular Coordinate System Definition, Form 2
CORD2S	Shperical Coordinate System Definition, Form 2
CPENTA	Five-Sided Solid Element Connection
CTETRA	Four-Sided Solid Element Connection
CQUAD4	Quadrilateral Plate Element Connection
CROD	Rod Element Connection
DEFORM	Static Element Deformation
FORCE	Static Force
GRAV	Acceleration or Gravity Load
GRID	Grid Point
LOAD	Static Load Combination (Superposition)
MAT1	Isotropic Material Property Defintion
MOMENT	Static Moment
PARAM	Parameter
PBAR	Simple Beam Property
PLOAD	Static Pressure Load
PLOAD1	Applied Loads on CBAR, CBEAM or CBEND Elements
PLOAD4	Pressure Loads on Face of Structural Elements
PROD	Rod Property

PSOLID	Properties of Solid Elements
RBAR	Rigid Bar
SPC	Single-Point Constraint
SPC1	Single-Point Constraint, Alternate Form
SPCD	Enforced Displacement Value
TEMP	Grid Point Temperature Field
TEMPD	Grid Point Temperature Field Default

Full descriptions of these and other Bulk Data entries can be found in the "**Bulk Data Entries**" on page 849 of the *MSC.Nastran Quick Reference Guide.*

Bibliography

Using MSC.Nastran: Statics and Dynamics. Cifuentes, A. O.; Springer-Verlag Inc., July, 1989.

> *Using MSC.Nastran: Statics and Dynamics* contains numerous well-narrated examples of static and dynamic analysis arranged in a tutorial-style format. (Softcover, 472 pages.)

What Every Engineer Should Know About Finite Element Analysis (Second Edition, Revised and Expanded). Brauer, John R., Ed.; Marcel-Dekker, Inc., 1993.

> This book contains overviews of finite element theory and applications, including structural, thermal, electromagnetic, and fluid analyses. (Hardcover, 325 pages.)

MSC.Nastran Bibliography (Second Edition). Miller, M. and Navarro, E.; The MacNeal-Schwendler Corporation, 1993.

> The *MSC.Nastran Bibliography* contains approximately 1600 citations of MSC.Nastran-related publications listed by author and by analysis topic. Over 60 topic categories are included. (Softcover, 222 pages.)

MSC.Nastran Verification Problem Manual (Version 65). The MacNeal-Schwendler Corporation, 1988.

> The *MSC.Nastran Verification Problem Manual* contains a variety of sample problems that can be verified by hand calculation. A theoretical evaluation and MSC.Nastran input file is included for each problem, and MSC.Nastran results are compared to theory. The examples include statics, normal modes, transient and frequency response, heat transfer, cyclic symmetry, superelement, buckling, geometric and material nonlinear, and aeroelastic analysis. (Softcover, 274 pages.)

MSC.Nastran Demonstration Problem Manual (Version 64). The MacNeal-Schwendler Corporation, 1985.

> The *MSC.Nastran Demonstration Problem Manual* is a collection of example problems oriented toward the new user. The MSC.Nastran input file and selected results are shown for each problem. (Loose-leaf binder, approximately 325 pages.)

Glossary

A

Axisymmetric - A type of finite element (or problem) in which the element's cross section is symmetric about an axis of rotation. Used to model structures such as shafts and rotationally-symmetric pressure vessels (which geometrically represent surfaces or solids of revolution).

B

Basic Coordinate System - MSC.Nastran's default, built-in rectangular coordinate system. All user-defined (local) coordinate systems are referenced from the basic coordinate system. The basic coordinate system is sometimes called the global coordinate system in other finite element programs. See also *Local Coordinate System*.

Beam Orientation Vector - A user-defined vector ($\vec{v}$) which is used to orient cross sectional properties of CBAR and CBEAM elements with respect to the model's geometry.

Bulk Data Section - The section of the MSC.Nastran input (.DAT) file containing model geometry, element connections and properties, loads, constraints, and material properties. Follows the Case Control Section.

C

Case Control Section - The section of the MSC.Nastran input (.DAT) file containing commands which select the type of analysis output required (displacements, forces, stresses, etc.). Case Control also manages sets of Bulk Data input (e.g., loads and constraints) and defines analysis subcases. The Case Control Section follows the Executive Control Section and precedes the Bulk Data Section.

Comment Entry - An entry used to insert user-specified comments anywhere in the input (.DAT) file. The first character is a dollar sign ($) in column 1, followed by any characters out to column 80.

Constraint - The enforcement of a particular displacement (translation and / or rotation) on a grid point or points. The boundary conditions of a static structure typically require a zero displacement constraint on various degrees of freedom in the model. Constraints may also be defined in terms displacement with respect to other degrees of freedom in the model, or in terms of an enforced nonzero value of displacement.

Continuation - An extension of a Bulk Data entry when the entry requires more than 80 columns of input data. Continuations may or may not be required depending on the particular Bulk Data entry and its options.

Coordinate System - See specific type (*Basic, Displacement, Element, Global, Local,* and *Material*).

Current Error List (CEL) - A list of all known MSC.Nastran errors. Maintained and frequently updated by MSC Software. Required reading for all users. See also *General Limitations List*.

D

.DAT File - Also called the input file, the .DAT file contains the complete MSC.Nastran finite element model. The input file is submitted to MSC.Nastran which then executes the analysis. The input file has the following principal sections:

NASTRAN statement - optional
File Management Section (FMS) - optional
Executive Control Section - Required
Case Control Section - Required
Bulk Data Section - Required

.DBALL File - Created by running MSC.Nastran. .DBALL is the extension name of a file containing permanent data for database runs.

Decomposition - The first step in solving a large system of linear equations, decomposition breaks the stiffness matrix [K] into lower and upper triangular factors. This process is one of the most computationally time-consuming steps in linear static analysis.

Degree-of-Freedom (DOF) - In linear static analysis, each grid point can undergo at most three orthogonal translational and three orthogonal rotational components of displacement. Each component is called a degree of freedom and adds one unknown to the system of simultaneous linear equations representing the structure.

Delimiter - An entry in the MSC.Nastran input (.DAT) file which indicates the beginning or end of a section. CEND, BEGIN BULK, and ENDDATA are required delimiters in all input files.

Discretization - The basic process of finite element modeling wherein a continuous structure is broken up-discretized-into an assembly of individual elements. The goal is to choose types and quantities of elements such that the mathematical behavior of the model faithfully represents the behavior of the structure. Properly discretizing the structure requires knowledge of the structure and engineering judgement.

Displacement Coordinate System - Each grid point has a displacement coordinate system, as selected in field 7 of the GRID Bulk Data entry. Displacements, constraints, and other grid point-based quantities are determined and reported based on this coordinate system. The basic coordinate system is MSC.Nastran's default displacement coordinate system.

Displacement Method - A method of structural analysis in which displacements are the unknown quantities to be determined. MSC.Nastran uses the Displacement Method.

DMAP - Acronym for Direct Matrix Abstraction Program. DMAP is MSC.Nastran's high-level programming language. DMAP allows advanced users to access MSC.Nastran's internal modules to modify existing solution sequences or to create customized solution methods.

DOF - See *Degree-of-Freedom*.

E

Echo - A listing of the input file (.DAT) written into the .F06 results file. Can be unsorted (appears exactly like the .DAT listing), sorted (alphabetized, in small field format, comment entries removed), or both, as specified by the ECHO Case Control Command.

Element Coordinate System - Each element has an element coordinate system based on the element's particular geometry and grid point ordering sequence. Quantities such as element force and stress are output in the element coordinate system.

Elemental Stiffness Matrix - The stiffness matrix of an individual finite element, often denoted as [k]. The stiffness matrix describes the element's displacement response for a given load. See also *Global Stiffness Matrix*.

Epsilon - A measure of numerical accuracy and roundoff error provided in the .F06 results file of linear static analysis runs. A small value of epsilon, less than 10^{-3} (and typically much less), indicates a numerically stable problem. A large value of epsilon is evidence of numerical ill-conditioning. See also *Ill-conditioning*.

Executive Control Section - A required section of the input file. Appears before the Case Control Section. Contains requests for solution sequence type (the SOL statement), CPU time limits (the TIME statement), and an optional identification entry (the ID statement).

F

.f06 File - Created by running MSC.Nastran. .F06 is the extension name of the file containing the numerical results of the analysis (stresses, forces, displacements, etc.).

.f04 File - Created by running MSC.Nastran. .F04 is the extension name of the file containing database information and a module execution summary. The .F04 file is a valuable aid when debugging problems with the model.

Fatal Error - see *User Fatal Message*.

File Management Section (FMS) - An optional input file section used primarily to attach or initialize MSC.Nastran databases and FORTRAN files. The FMS section, if used, precedes the Executive Control Section.

Finite Element - The basic building block of the finite element method. Finite elements are regularly or nearly regularly shaped mathematical idealizations of simple structures (e.g., beams, plates, solids) with known mathematical solutions. When individual elements are combined to represent a complex irregular structure, the resulting mathematical model approximates the behavior of the real structure.

Free Field Format - Input data format in which data fields are separated by commas or blanks.

G

General Limitations List - A list maintained by MSC Software describing general limitations of MSC.Nastran. General limitations acknowledge and describe a lack of functionality in various areas of the program. See also *Current Error List*.

Global Coordinate System - A term used in MSC.Nastran to describe the collection of all displacement coordinate systems. Be aware that some other commercial finite element programs use the term “global coordinate system” to describe what MSC.Nastran calls its basic (default) coordinate system. See also *Basic Coordinate System*.

Global Stiffness Matrix - The stiffness matrix of the entire structure. The global stiffness matrix is an assembly of the elemental stiffness matrices of individual elements. See also *Elemental Stiffness Matrix*.

Grid Point - A geometric point that defines model geometry and to which finite elements are connected. Grid points are located in space with respect to a particular coordinate system and displace with the loaded structure. Analysis results such as displacements and reaction forces are reported at grid points. The basic equations of finite element analysis are written in terms of grid point displacement.

I

Ill-conditioning - A system of linear equations is said to be ill-conditioned if small perturbations in the system lead to large changes in the solution. MSC.Nastran checks for evidence of ill-conditioning in the system of equations representing the structural model. A high value of epsilon indicates a potential ill-conditioning problem. Ill-conditioning does notnecessarily result in a fatal error, but can result in inaccurate answers. Possible causes of ill-conditioning include a high difference in stiffness between adjacent elements in the model, unconnected degrees of freedom, rigid body motion, or the presence of mechanisms. See also *Epsilon*.

Information Message - see *User Information Message*

Input File - See *.DAT file.*

Isoparametric - A modern type of finite element formulation which offers high accuracy with good efficiency (i.e., relatively low computational cost).

L

Large Field Format - Input format for Bulk Data entries in which data fields are 16 columns wide, allowing numerical data to have a greater number of significant digits.

Line Element - Elements such as bars, rods, and beams. Typically connected by two grid points. Also known as one-dimensional elements.

Linear Structure - A structure whose displacements are directly proportional to load.

Load - A general term referring to forces, moments, pressure loads, thermal loads, electromagnetic loads, etc. In MSC.Nastran analysis, loads are known quantities that are applied to the structural model. MSC.Nastran then solves for the unknown displacements of the structure.

Local Coordinate System - A user-specified system that accommodates the inputting of geometry data or the reporting of analysis results. Local coordinate systems can be rectangular, cylindrical, or spherical, and are defined with respect to the basic (default) coordinate system. See also *Basic Coordinate System*.

.log File - Created by running MSC.Nastran. .log is the extension name of a file containing system information and system error messages.

M

Machine Zero - A value of zero with a small amount of computer roundoff error added. In a typical structural model, a number such as 3.01652E-11 is a machine zero.

.MASTER File - Created by running MSC.Nastran. .MASTER is the extension name of a file containing the master directory for database runs.

Material Coordinate System - An optional coordinate system used to orient orthotropic or anisotropic material properties with output results. For homogeneous isotropic materials, there is no need to define a material coordinate system.

Mechanism - A mechanism occurs when part of a structure is capable of rigid body (strain-free) motion. In linear static analysis, the presence of a mechanism produces a singularity failure in the solution.

Mesh - The pattern formed by a collection of finite elements. Relatively few elements result in a coarse mesh. Adding more elements produces a finer mesh, which can more closely represent an irregularly shaped structure. In general, a finer mesh is more accurate, but it is also more computationally expensive.

Module - A set of MSC.Nastran program subroutines designed to perform a particular mathematical or data-related task. Users have direct access to MSC.Nastran's modules via DMAP. See also *DMAP*.

N

NASTRAN Statement - An optional statement which, if used, appears at the beginning of the MSC.Nastran input file. The NASTRAN statement is used to override the default values for certain operational parameters of the program. See Section 1 of the *MSC.Nastran Quick Reference Guide* for further information.

Nonlinear (Geometric) - Structural displacements that are larger than those allowed by small displacement-based theory are said to be geometrically nonlinear. So-called large displacements require the use of special nonlinear solution sequences in MSC.Nastran.

Nonlinear (Material) - A material in which stress is not directly proportional to strain, such as rubber. Nonlinear materials require the use of special nonlinear solution sequences in MSC.Nastran.

Normal Modes Analysis - An analysis used to determine the natural frequencies and mode shapes of a structure.

O

Output File - See *.f06 File*.

P

PARAM, AUTOSPC - A parameter that controls the detection and constraining of singularities in the model. PARAM,AUTOSPC is in operation by default in most solution sequences. PARAM,AUTOSPC produces a Grid Point Singularity Table in the .f06 output file; this table lists the singularities that were detected and constrained.

Parameter - Parameters are used to request special program features and to input data relating to these features. Parameters are specified on PARAM Bulk Data entries and PARAM Case Control commands. A complete listing of parameter functions is given in "**Parameters**" on page 601 of the *MSC.Nastran Quick Reference Guide*.

Postprocessor - A software package designed to convert the wealth of numerical data generated by finite element analysis to a graphically-based, easily visualized format. Common postprocessing operations include x-y plots of numerical data, deformed shape plots of model geometry, and color stress contour plots.

Preprocessor - A software package designed to help build the finite element model. Preprocessors create geometry, mesh elements, apply loads and constraints, and check the model for certain types of errors.

R

Results File - See *.f06 File*.

Rigid Body Motion - Rigid body motion occurs when the structural model is free to displace in one or more directions (displacement without strain). A simple example of rigid body motion occurs when you move a pencil from one location on your desk to another. No strain occurs in the pencil-only translation and rotation as a rigid body. In static analysis, the possibility of rigid body motion due to an insufficiently constrained structure results in a singularity in the stiffness matrix. Consequently, the solution of the problem fails during decomposition of the stiffness matrix.

Rigid Element - The name "rigid" element is somewhat misleading-the so-called "R"-type elements are actually constraint elements and do not add additional physical stiffness to the model. For example, the RBE2 element described in this book simulates a rigid bar by mathematically linking the displacements of the connected grid points.

Rotation - Displacement about a coordinate axis. A grid point has three rotational degrees of freedom, one about each axis. See also *Degree-of-Freedom* and *Translation*.

S

Set - A collection, or grouping, of particular items in the MSC.Nastran model-examples include load sets, constraint sets, and collections of elements.

Single Point Constraint (SPC) - The constraint of one or more degrees of freedom at a grid point, thereby enforcing displacement (often zero displacement) of the grid point in the affected component directions. For example, the grid point at the fixed end of a cantilever beam is constrained (SPC'd) in all six DOFs. Reaction forces, called forces of single point constraint (SPCF), are recovered at these grid points.

Singularity - A mathematical condition prohibiting matrix inversion. Consequently, the system of equations representing the structure cannot be solved. Common sources of singularities in linear static analysis include the presence of unconnected (to structural stiffness) or very weakly connected degrees of freedom, or an inadequate prescription of constraints on the model resulting in rigid body motion.

"Slowly Applied" Loads - A basic assumption of static analysis: loads must be sufficiently "slowly applied" so as to cause no significant dynamic effects.

Small Displacements - A requirement of linear structural analysis. Displacements must be sufficiently small so as not to violate certain mathematical assumptions inherent in the design of the finite elements used. Large displacements require nonlinear solution methods.

Small Field Format - Input format for Bulk Data entries in which data fields are eight columns wide.

Solid Element - Elements resembling bricks (eight corners), wedges (six corners), or pyramids (four corners). Also called three-dimensional elements. Popular MSC.Nastran solid elements include the CHEXA, CTETRA, and CPENTA.

Solution Sequence - A prepackaged set of DMAP instructions designed to solve a particular type of engineering problem. The SOL command in the Executive Control Section is used to tell MSC.Nastran which solution sequence to use; for example, SOL 101 is used to specify linear static analysis. See also *DMAP*.

Spring Element - Elements representing simple single degree of freedom extensional or rotational springs. Also called a zero-dimensional elements or scalar elements. The CELASi family of elements are spring elements.

Static - In the MSC.Nastran sense, static means that the structural model is constrained to prevent rigid body motion (static equilibrium exists) and that loads are assumed to be "slowly applied," thereby inducing no dynamic effects.

Stiffness Matrix - see *Global Stiffness Matrix* and *Elemental Stiffness Matrix*.

Subcase - Subcases allow multiple individual load cases to be analyzed in the same MSC.Nastran run, thereby achieving greater computational efficiency than with separate runs (the stiffness matrix is only decomposed once). See also *Decomposition*.

Surface Element - Elements such as thin plates (which are flat) or shells (which are curved). Also called two-dimensional elements. Popular MSC.Nastran surface elements include the CTRIA3 triangular and CQUAD4 quadrilateral elements.

Symmetry - A geometric property in which a structure has one or more planes of symmetry. Structural symmetry can be exploited to produce a smaller model (appropriate constraints are used to model the boundary conditions on the axis or axes of symmetry).

System Message - System Messages refer to diagnostics associated with program errors. Analogous to User Messages.

T

Translation - Direct, linear displacement along a coordinate axis. A grid point has three translation degrees of freedom, one along each axis. See also *Degree-of-Freedom* and *Rotation*.

U

User Fatal Message (UFM) - An MSC.Nastran message describing an error severe enough to cause the program to terminate.

User Information Message (UIM) - An MSC.Nastran message that provides general information. Not necessarily indicative of a problem.

User Warning Message (UWM) - An MSC.Nastran message warning of an atypical situation; the user must determine whether or not a problem exists.

V

von Mises Stress - A convenient and commonly used value of stress in finite element work. Always a positive number, von Mises stress is related to the octahedral shear stress criterion for yielding of a ductile material. For a general stress state (nonprincipal axes), von Mises stress is given by the following equation:

$$\sigma_{von} = \frac{1}{\sqrt{2}}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{yz})^2 + 6(\tau_{zx})^2 + 6(\tau_{xy})^2]^{1/2}$$

I N D E X

Getting Started with MSC.Nastran User's Guide

A

B

F

G

I

J

K

L

M

N

O

P

R

S

T

U

V

W