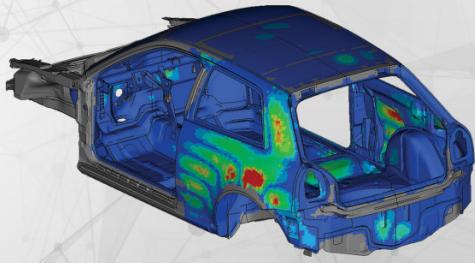




MSC **Nastran** 2018



Getting Started Guide

Corporate

MSC Software Corporation
4675 MacArthur Court, Suite 900
Newport Beach, CA 92660
Telephone: (714) 540-8900
Toll Free Number: 1 855 672 7638
[Email:](mailto:americas.contact@mscsoftware.com) americas.contact@mscsoftware.com

Japan

MSC Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
[Email:](mailto:MSCJ.Market@mscsoftware.com) MSCJ.Market@mscsoftware.com

Worldwide Web

www.mscsoftware.com

Disclaimer

MSC Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright © 2017 MSC Software Corporation. Printed in U.S.A. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC Software Corporation is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from MSC Software suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at <http://www.mscsoftware.com/thirdpartysoftware> (or successor website designated by MSC from time to time).

PCGLSS 8.0, Copyright © 1992-2016, Computational Applications and System Integration Inc. All rights reserved. PCGLSS 8.0 is licensed from Computational Applications and System Integration Inc.

MSC, Dytran, Marc, MSC Nastran, Patran, the MSC Software corporate logo, e-Xstream, Digimat, and Simulating Reality are trademarks or registered trademarks of the MSC Software Corporation and/or its subsidiaries in the United States and/or other countries.

NASTRAN is a registered trademark of NASA. LS-DYNA is a trademark or registered trademark of Livermore Software Technology Corporation. FLEXlm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. All other trademarks are the property of their respective owners.

Use, duplicate, or disclosure by the U.S. Government is subjected to restrictions as set forth in FAR 12.212 (Commercial Computer Software) and DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), as applicable.

U.S. Patent 9,361,413

November 10, 2017

NA:V2018:Z:Z:DC-GET-PDF

Europe, Middle East, Africa

MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
[Email:](mailto:europe@mscsoftware.com) europe@mscsoftware.com

Asia-Pacific

MSC Software (S) Pte. Ltd.
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
[Email:](mailto:APAC.Contact@mscsoftware.com) APAC.Contact@mscsoftware.com

Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on.)
- Your suggestions for correcting/improving documentation

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at: <http://msc-documentation.questionpro.com>

Note: The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact the Technical Support team at: <http://www.mscsoftware.com/support/technical-support-programs-and-offerings>



Contents

MSC Nastran Getting Started Guide

Preface

How to Use this Guide	10
Prerequisites	10
Goals of this Guide.....	10
Contents of This Guide	10
List of MSC Nastran Guides	12
Using other Manuals	13
Typographical Conventions	14
Accessing MSC Nastran Manuals	15
Downloading the PDF Documentation Files	15
Navigating the PDF Files	15
Printing the PDF Files	15
Training and Internet Resources	16
Technical Support	18
Visit SimCompanion.....	18
Help Us Help You.....	19

1 Overview of MSC Nastran

Introduction to MSC Nastran	22
History of MD and MSC Nastran	23
Working With MSC Nastran	25
Workflow of MSC Nastran with Patran	25
Pre-processor.....	26
Solver.....	28
Post-processing	28
Workflow of MSC Nastran with SimXpert	28
Capabilities of MSC Nastran	30
Benefits of using MSC Nastran	31



2 MSC Nastran Capabilities

General Capabilities	34
Solution Sequence (SOL)	35
Linear Analysis	36
Solution Flowchart	36
The Elemental Stiffness Matrix	37
The Global Stiffness Matrix	40
Dynamic Analysis	42
Normal Modes Analysis	43
Frequency Response Analysis	45
Transient Response Analysis	46
Complex Eigen analysis	47
Response Spectrum Analysis	47
Component Mode Synthesis	47
Random Vibration Analysis	48
Nonlinear Analysis	48
Material Nonlinearity	50
Geometric Nonlinearity	51
Nonlinear Boundary Conditions	54
Nonlinear Transient Analysis	56
Design Sensitivity and Optimization	56
Application Examples	57
Aeroelasticity	59
Static Aeroelastic Response	59
Aerodynamic Flutter	59
Dynamic Aeroelastic Response	60
Aerodynamic Methods	60
Thermal Analysis	61
Transient Thermal Analysis	61
Composite Materials	62
Fluid-Structure Interaction	62
Direct Matrix Abstraction Program	63
Fatigue Analysis	64
Five Box Trick of Fatigue Analysis	64
Life Prediction Methods	66
Finite Element Analysis Methods	66
When to Use Which Method	67
High Performance Computing	68
Hardware Factors Affecting Simulation Performance	68



3 MSC Nastran Files

Types of MSC Nastran Files	74
MSC Nastran Input Files	74
Delimiters	77
Nastran Statement	77
File Management Section (FMS)	77
Executive Control Section	78
Summary of Executive Control Statements	79
ID Statement	79
SOL Statement	80
CEND Statement	80
Case Control Section	80
Bulk Data Section	82
Format of Bulk Data Entries	82
Summary of Bulk Data Entries	83
Continuation Entries	85
Field Formats in Input Files	86
Input Format Rules	89
Common Errors in Entry Format	89
MSC Nastran Output Files	90

4 Modeling Issues

Units	92
Format of the Input Data	92
Real, Integer, and Character Input Data	92
Free, Small, and Large Field Formats	93
Continuations	96
Meshes and Mesh Transitions	97
Designing the Model	98
Using Test Models	100
Experiment With Small Test Models	100
Inability to Predict the Response of your Structure	100

5 Model Geometry

Model Geometry	102
Grid Points	103
Coordinate Systems	105



6 Basic Element Library

Introduction	112
Spring Element (CELAS2)	113
CELAS2	113
Line Elements	115
Rod Element (CONROD)	116
Rod Element (CROD)	118
Rod Element Property (PROD)	118
Simple Beam Element (CBAR)	119
Surface Elements	127
Quadrilateral Plate Element (CQUAD4)	128
Triangular Plate Element (CTRIA3)	131
Shell Element Property (PSHELL)	133
Other Surface Elements	139
Solid Elements	139
Six-Sided Solid Element (CHEXA)	140
Five-Sided Solid Element (CPENTA)	142
Four-Sided Solid Element (CTETRA)	145
Solid Element Property (PSOLID)	147
Rigid Bar Element (RBE2)	148

7 Material Properties

Basic Material Property Definitions	152
Material Definition (MAT1)	153
Other Material Types	155

8 Applying Constraints

Description	160
Rigid Body Motion and Mechanisms	160
Static Equilibrium	161
Mechanisms	161
Inertia Relief	161
Single Point Constraints	161
Permanent Constraint on the GRID Entry	161
Single Point Constraint (SPC)	162
Automatic Identification and Removal of Singularities (AUTOSPC)	162
Boundary Condition Examples	163
Example 1: Cantilever Beam	163



Example 3: Fixed-Hinged Beam	165
--	-----

9 Applying Static Loads

Overview of Basic Static Loads	168
Concentrated Loads	173
Forces (FORCE)	173
Moments (MOMENT)	175
Distributed Load on a 1-D Element (PLOAD1)	177
Pressure Loads	182
Uniform Normal Pressure Load on Surface (PLOAD)	182
Uniform Normal Pressure Load on a 2-D Element (PLOAD2)	183
Normal or Traction Pressure Load on a Face (PLOAD4)	184
Acceleration Loads (GRAV)	186
Enforced Displacements	189
Static Element Deformation (DEFORM)	189
Enforced Displacement Value (SPCD)	190
Combining Loads (LOAD)	190
Using Subcases (SUBCASE, SUBCOM, SUBSEQ, SUBSEQ1)	192
In SUBCOM 10:	193
In SUBCOM 20:	193

10 Controlling the Analysis Output

Printing the Input File (ECHO)	196
Output Titles (TITLE, SUBTITLE, LABEL)	198
Case Control Sets (SET)	199
Requesting Analysis Results	200
Grid Point Output (DISP, SPCF, OLOAD, GPFORCE)	200
Element Output (STRESS, FORCE, STRAIN, ESE)	200

11 Additional Considerations

MSC Nastran User and System Messages	204
Message Format	204
User Messages	204
System Messages	205
Epsilon-A Measure of Numerical Behavior	205
Element Distortion and Accuracy	206

CQUAD4 Plate Element Distortion	206
CHEXA Solid Element Distortion	208
The Current Error List.....	209
The Comment (\$) Entry.....	209

12 Performing Analysis

Defining the Problem	212
Specifying the Type of Analysis	212
Designing the Model.....	213
Creating the Model Geometry	214
Defining the Finite Elements	216
Specifying Material Properties.....	220
Representing Boundary Conditions	221
Applying the Loads.....	223
Controlling the Analysis Output	224
Completing the Input File and Running the Model	225
MSC Nastran Output.....	226
Reviewing the Result	230
Comparing the Results with Theory	235

13 Executing MSC Nastran

Run MSC Nastran	238
Using nastran Command	238
Using Pre-processor to Execute MSC Nastran	240
Checking the Analysis	242
Interpreting the .f04 File	243
Physical File Information	243
Memory Map	244
Day Log	245
Memory and Disk Usage Statistics	246
Database Usage Statistics	247
.....	248

14 Performance Considerations

Introduction	250
---------------------------	------------



Geometric Reduction Methods	250
Superelement Analysis	250
Axisymmetric Analysis	252
Cyclic Symmetry Analysis	252
Memory and I/O Performance.....	253
Default Memory Settings	253
I/O Performance	253
Buffer Pool Usage	253
Parallel Performance.....	254

15 Example Problems

Beam with a Distributed Load and a Moment.....	256
Problem Statement	256
The Finite Element Model	257
MSC Nastran Results.....	258
Comparison with Theory	263
Rectangular Plate with a Uniform Pressure Load	264
Problem Statement	264
The Finite Element Model	264
MSC Nastran Results.....	268
Gear Tooth with Solid Elements	274
MSC Nastran Results.....	279

Glossary

Bibliography



Preface

- How to Use this Guide 10
- List of MSC Nastran Guides 12
- Using other Manuals 13
- Corrected and Fixed Error List 14
- Typographical Conventions 14
- Accessing MSC Nastran Manuals 15
- Training and Internet Resources 16
- Technical Support 18



How to Use this Guide

This guide is written for a new MSC Nastran user. No prior experience with commercial finite element software is assumed, however, basic knowledge of the finite-element method and applied mechanics is a pre-requisite.

Prerequisites

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 Guide

This guide aims to give you insight into performing structural analysis using MSC Nastran. You will also gain a basic understanding of some of the intermediate and advanced features of MSC Nastran.

Contents of This Guide

A brief description on concept of each chapter is as follows:

Chapter	Description
Chapter 1: Overview of MSC Nastran	Provides an overview of MSC Nastran as well as using it with a graphical user interface like Patran or SimXpert.
Chapter 2: MSC Nastran Capabilities	Provides an overview of the capabilities available with MSC Nastran including aeroelasticity, optimization, and high performance computing.
Chapter 3: MSC Nastran Files	Describes the basic MSC Nastran input file, including the Case Control and Bulk Data sections, in addition to the various output files types.
Chapter 4: Modeling Issues	Describes the basics required to build and design a model.
Chapter 5: Model Geometry	Explains the usage of correct geometry for a model, where the information may come from CAE or CAD systems.
Chapter 6: Basic Element Library	Explains the general and special finite elements.
Chapter 7: Material Properties	Describes the various materials—linear, elastic, homogeneous, and isotropic.
Chapter 8: Applying Constraints	Explains the types of constraints in using MSC Nastran. The chapter also describes how and when these constraints are used in MSC Nastran.
Chapter 9: Applying Static Loads	Provides the most commonly used loads for linear static analysis.
Chapter 10: Controlling the Analysis Output	Explains on how a .dat text file is submitted to MSC Nastran and .f06 result file is produced.

Chapter	Description
Chapter 11: Additional Considerations	Describes the various messages which are related to the status of the FEM model and the computer system.
Chapter 12: Performing Analysis	Explains how to perform a complete MSC Nastran Analysis.
Chapter 13: Executing MSC Nastran	Explains in detail how to run MSC Nastran both on the command line and through Patran. It also interprets the basic output files like the F04 and the log file.
Chapter 14: Performance Considerations	Explains the methods used to solve large system of equations using sparse matrix algorithms.
Chapter 15: Example Problems	Provides different types of examples on performing an MSC Nastran Analysis step by step.

MSC Nastran Getting Started Guide contains many highlighted links (in blue) to other MSC Nastran documents and all the documents are delivered together as a documentation set. If you keep the documentation set together the links between documents will work.

Two ways of working with links are as follows:

- Click on the link to open another document. Use alt ← (alt + backspace) to return to the original document.
- If you know to which document the hyperlink is referring to, open that document in Adobe Reader. Perform the following steps:
 - a. Choose Edit → Preferences → Documents Edit
 - b. Uncheck the **Open cross-document links in same window** option
 - c. Select **OK**

List of MSC Nastran Guides

Installation and Release Guides

- Installation and Operations Guide
- Release Guide

Reference Guides

- Quick Reference Guide
- DMAP Programmer's Guide
- Reference Guide
- Utilities Guide

User's Guides

- Getting Started
- Embedded Fatigue
- Embedded Vibration Fatigue
- Linear Static Analysis
- Dynamic Analysis
- Thermal Analysis
- High Performance Computing
- Superelements
- Design Sensitivity and Optimization
- Rotordynamics
- Nonlinear (SOL 400)
- Aeroelastic Analysis
- User Defined Services
- Demonstration Problems



Using other Manuals

After reading the *Getting Started Guide*, we recommend that you go through either the *Linear Static Analysis User's Guide*, the *Dynamic Analysis User's Guide*, or the *Nonlinear User's Guide* depending on the type of simulation you are interested in performing. Details of some of the manuals are listed here for your reference.

This guide contains many excerpts from the *Quick Reference Guide (QRG)*. Most of the excerpts have been edited-some extensively-to eliminate material that is not relevant to the topics covered in this book.

- **MSC Nastran Quick Reference Guide:** The *QRG* contains a complete description of all the input entries for MSC Nastran. It contains complete descriptions of all the finite element input data and options available in MSC Nastran. Each entry provides a description, formats, examples, details on options, and general remarks. You will find the full descriptions for all SOL 400 input entries in the *QRG*.
- **MSC Nastran Reference Manual:** It provides supporting information that relates to the theory of MSC Nastran inputs, element libraries, and loads and boundary conditions.
- **MSC Nastran Linear Static Analysis User's Guide:** It provides support information on the basic use of MSC Nastran which can also be applied to SOL 400.
- **Nonlinear User's Guide SOL 400:** This guide provides a complete background to MSC Nastran Implicit Nonlinear (SOL 400) and describes how to use SOL 400 within the MSC Nastran environment. The theoretical aspects of nonlinear analysis methods, types, and techniques are included along with thorough descriptions for nonlinear material models, properties, and loads and constraints.
- **Dynamic Analysis User's Guide:** This guide describes the proper use of MSC Nastran for solving various dynamic analysis problems. This guide serves as both an introduction to dynamic analysis for the new user and a reference for the experienced user.
- **MSC Nastran High Performance Computing User's Guide:** It describes the strategy for getting the best performance out of MSC Nastran for SOL 101, 103, 108, 111, and 400. The guide also gives details on selecting the hardware system that provides the optimal performance for your models.
- **DMAP Programmer's Guide:** It provides example problems and includes description of the input, procedures, and results information that relates to the practical use of the MSC Nastran inputs, element libraries, and loads and boundary conditions.

This guide contains many highlighted links (in blue) to other MSC Nastran documents and all the documents were delivered together as a collection. If you keep the collection together the links between documents will work.

Two ways of working with links are as follows:

- Use alt ← to return back to the window your cursor is in
- Open the other “linked to” documents in a new window from an Adobe Reader
 - a. Choose Edit → Preferences → Documents → □
 - b. Open cross-document links in the same window.

- c. Un-check the checkbox.
- d. Select OK.

Corrected and Fixed Error List

For a list of corrected defects in the current release of MSC Nastran, visit our Simcompanion site at:

<https://simcompanion.mscsoftware.com/infocenter/index?page=content&id=K18008006>

Typographical Conventions

To distinguish special uses of words, we use certain typographic conventions. This section provides overview of the typographical conventions used in the document.

This section describes some syntax that will help you in understanding text in the various chapters and thus in facilitating your learning process. It contains stylistic conventions to denote user action, to emphasize particular aspects of a MSC Nastran run or to signal other differences within the text.

Courier New	Represents command-line options of MSC Nastran and results from <code>f04/f06</code> files. Example: <code>nast20180 memorymax=16gb myjob.dat</code>
Quoted Text	Represents command-line options of MSC Nastran for in-line text. Example: <code>memorymax=16gb</code>
Arial font	To represent elements. Example: RBE3 and RSPLINE are interpolation elements and are not rigid.
Red Text	Represents items in the examples that we want to emphasize. Example: <code>smp=16</code>
Bold Text	Represents items in the text that we want to emphasize. Example: dmp=4
Italic Text	Represents references to manuals/documents. Example: <i>MSC Nastran Quick Reference Guide</i>

Note: As MSC Nastran does not have a User Interface, we use bold text to emphasize particular information. For the products that have User Interface (UI), typically Bold text is used to mention UI entities.

Accessing MSC Nastran Manuals

This section describes how to access the MSC Nastran documentation outside of MSC Software. MSC Nastran documentation is available through PDF files. The PDF files can be obtained from the following sources:

- MSC Nastran documentation installer
- SimCompanion
- Combined documentation

The PDF documentation files are appropriate for viewing and printing with Adobe Acrobat Reader (version 5.0 or higher), which is available for most Windows and Linux systems. These files are identified by a .pdf suffix in their file names.

Downloading the PDF Documentation Files

You can download the PDF documentation from SimCompanion (<http://simcompanion.mscsoftware.com>).

Navigating the PDF Files

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, links to other guides, hyperlinks to all cross-references to chapters, sections, figures, tables, bibliography, and index entries have been applied.

Printing the PDF Files

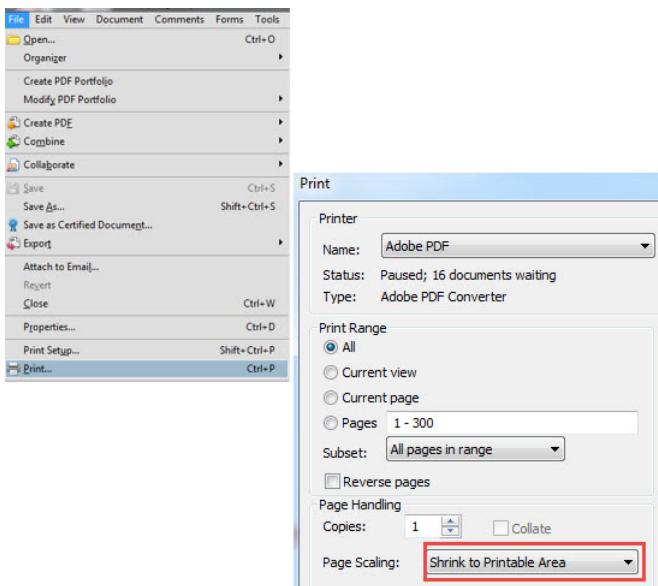
Adobe Acrobat PDF files are provided for printing all or part of the manuals. You can select the paper size to which you are printing in Adobe Acrobat Reader by doing the following:

1. Click **File**.
2. Select the **Print....** option. The **Print** dialog box is displayed.
3. Select **Page Setup....**
4. Choose the required paper size in the **Page Setup** menu.

The PDF files are recommended when printing long sections since the printout will have a higher quality.

If the page is too large to fit on your paper size, you can reduce it by doing the following:

1. Select the **File -> Print**.
2. Under **Page Scaling**, choose the **Shrink to Printable Area** option.



Training and Internet Resources

Information about MSC's products, services and latest events is available on our website www.mssoftware.com.

The information about MSC Seminars is available on the training link <http://www.mssoftware.com/msc-training>. You can also use this link to schedule the seminars.

If you are a new MSC Nastran user, we recommend the following courses:

NAS101A: Linear Static and Normal Modes Analysis using MSC Nastran

This course serves as an introduction to fundamental use of MSC Nastran. Through this course you will learn the following:

- Finite Element Analysis (FEA).
- Features available in MSC Nastran for solving structural engineering problems.
- To create all finite element models and edit using a text editor.
- Proper data structure of the MSC Nastran input file.

NAS101B: Advanced Linear Analysis using MSC Nastran

This course is a continuation of NAS101A. Through this course you will learn the following:

- Theory of buckling analysis and how to perform a buckling analysis.
- About rigid elements—MPC, RBAR, RBE2, and RBE3.

- Modeling with interface element CINTC and connectors.
- Lamination theory and composite materials.
- MSC Nastran composite theory.
- Failure theories.
- Linear contact and permanent glued contact.
- Different model checks.
- Modeling tips and tricks.

NAS 102A: Dynamic Analysis using MSC Nastran

Through this course you will learn the following:

- Fundamentals of dynamics
- Normal modes
- Direct and modal methods
- Frequency response
- Transient response
- Damping
- Enforced motion
- Response spectrum
- Random analysis
- Residual vectors

NAS 102B: Advanced Dynamic Analysis using MSC Nastran

This course will help you learn the following:

- Review and expand upon the basic dynamic analysis topics.
- Introduce model checkout tools for dynamics.
- Introduce advanced dynamic analysis capabilities.
- Show where to get technical support.
- Show the available training for MSC Nastran.

NAS 124: Thermal Analysis using MSC Nastran and Patran

This course will help you learn the following:

- Heat transfer
- Modes of heat transfer
- Types of analysis
- Thermal model elements and material properties

- Thermal model boundary conditions
- Thermal model loads
- Steady and transient state equation

NAS120: Linear Static Analysis using MSC Nastran and Patran

This seminar provides the foundation required for intermediate and advanced MSC Nastran applications:

- Linear static
- Normal modes
- Buckling analysis of structures with the use of MSC Nastran and Patran

In this course MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran is an integral part of the examples and workshops. It will be used to generate and verify illustrative MSC Nastran models, manage analysis submission requests, and to visualize results.

Technical Support

If you encounter difficulties while using MSC Nastran, first please refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve.

Visit SimCompanion

The product documentation is available in SimCompanion (<http://simcompanion.msccsoftware.com>). The SimCompanion gives you access to a wealth of resources for MSC Software products. You will find various information such as:

- Product documentations
- Knowledge base articles
- Product error lists (fixed and known issues for each release)
- SimAcademy webinars
- Product and support contact information

SimCompanion is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from MSC Software, allows you to manage your support cases, and participate in our discussion forums.

Help Us Help You

Clients frequently call up the support engineers at MSC Software with enquiry regarding 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.

- For information on the latest events, products and services for all products, refer to the MSC Software corporate site (www.mscsoftware.com).
- For technical support phone numbers and contact information, please visit:
<http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx>.

1

Overview of MSC Nastran

- Introduction to MSC Nastran 22
- History of MD and MSC Nastran 23
- Working With MSC Nastran 25
- Workflow of MSC Nastran with Patran 25
- Workflow of MSC Nastran with SimXpert 28
- Capabilities of MSC Nastran 30
- Benefits of using MSC Nastran 31



Introduction to MSC Nastran

MSC Nastran is a structural analysis application used by engineers to perform static, dynamic, and thermal analysis across the linear and nonlinear domains. It has High Performance Computing (HPC) capabilities and can take advantage of shared memory and distributed memory parallelism.

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. It is used extensively by aerospace, automobile, energy, biomedical, and other industries.

MSC Nastran allows engineers to virtualize physical prototypes and simulate their structural behavior when subjected to real world loading events. Engineers use MSC Nastran to compliment and drive product development. The reasons for selecting MSC Nastran are as follows:

- Accuracy and reliability
- Reduction in the number of expensive physical prototypes
- Rapid consideration of *what if* scenarios
- Have virtual prototype early in the design process, saving costs traditionally associated with physical prototyping
- Remedy of structural issues that may occur during a product's service, reducing downtime and costs
- Optimized performance of existing designs and unique product differentiations leading to industry advantages over competitors.

The clients of MSC Software either lease or purchase executable-only versions of the program. MSC Nastran is available on both Windows and Linux. Regardless of the machine used, MSC Nastran is optimized to run efficiently and provide equivalent 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.

MSC Nastran is mainly written in Fortran with scattered usage of C and C++ and contains over one million lines of code. It 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 pre-packaged 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 from you, except in choosing a solution sequence.



History of MD and MSC Nastran

From 2006-2011, there were two versions of Nastran:

- MD Nastran (MD stands for multi discipline)
- MSC Nastran

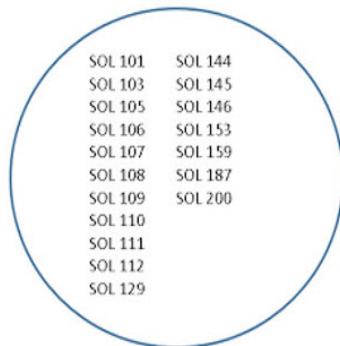
MD Nastran was the exclusive Nastran version to contain the powerful nonlinear solution SOL 400. MSC Nastran element code base was a subset of MD Nastran. All MSC Nastran input files ran on MD Nastran exactly the way they ran on MSC Nastran. Also, MD models using specific or advanced features did not run on MSC Nastran.

For SOL 101 to 200, MSC Nastran and MD Nastran were the same. Any version of MSC Nastran or MD Nastran was capable of running a .bdf file where the solution sequence is any of the following:

SOL 101	SOL 112
SOL 103	SOL 129
SOL 105	SOL 144
SOL 106	SOL 145
SOL 107	SOL 146
SOL 108	SOL 153
SOL 109	SOL 159
SOL 110	SOL 187
SOL 111	SOL 200

2001

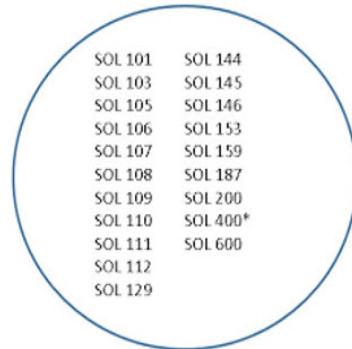
MSC Nastran



This version of MSC Nastran can run SOL 101-200 jobs.

2004

MSC Nastran



This version can run SOL 101-200, SOL 600, and SOL 400 jobs.



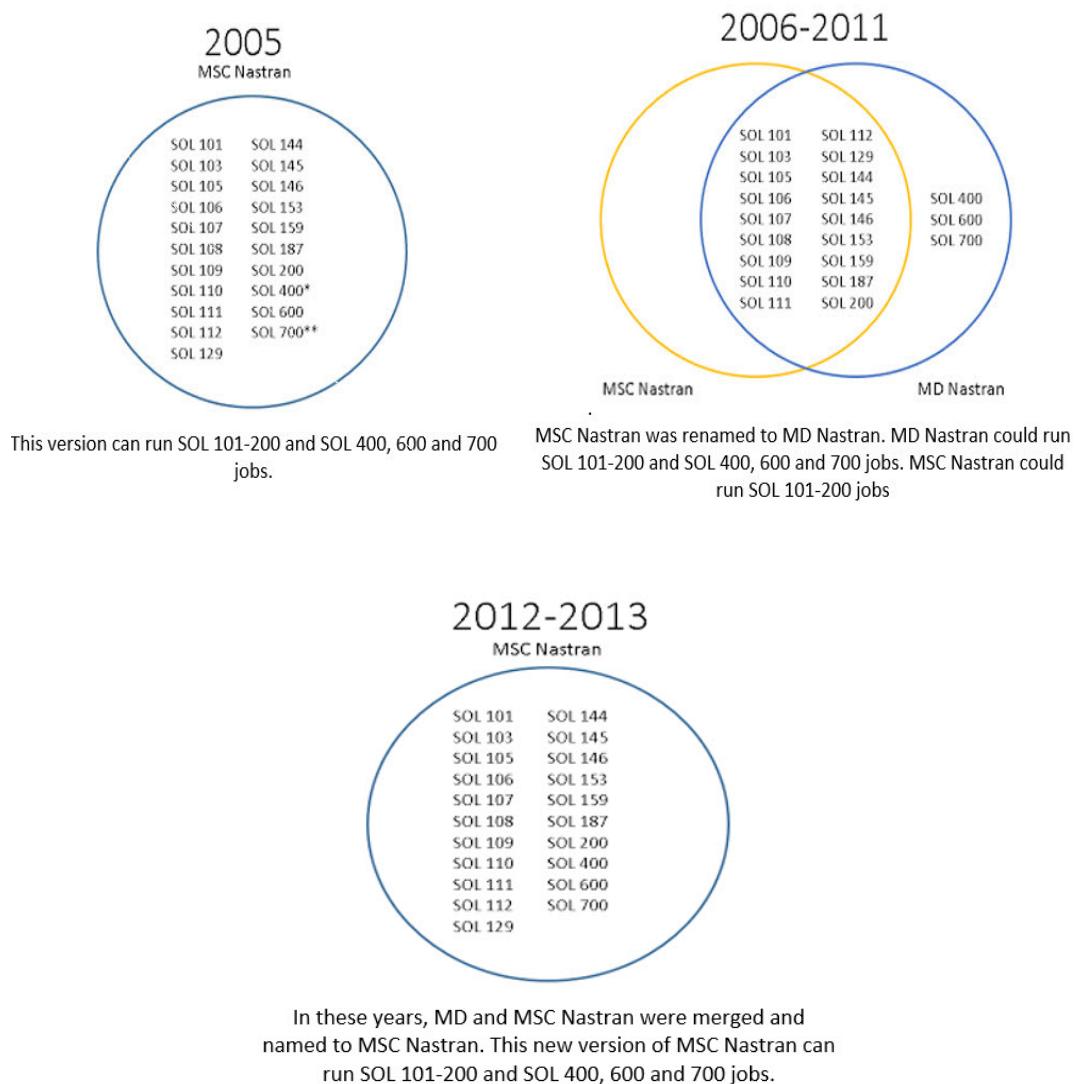


Figure 1-1 MD Nastran Vs MSC Nastran

In 2012, MSC and MD Nastran were merged into one and was named MSC Nastran 2012. So, any older tutorials or manuals labelled either MSC or MD Nastran can be of use to the new user.



Working With MSC Nastran

Developing a finite element model manually 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.

Hence, we use the following:

1. Pre-processor: It is primarily designed to aid in the development of a finite element model. Pre-processing software also helps the analyst modify the original model if results show that changes and subsequent reanalysis are required. The pre-processors are able to import geometric data from solid modeling, computer-aided design (CAD), or computer-aided manufacturing (CAM) software to be used as a basis for the FEM.
2. Post-processor: It is used to aid in the display and interpretation of analysis results. The processor might be integrated with the analysis software, or it might be a standalone software package.

We offer graphics-based software packages, MSC Patran and SimXpert as the pre-processor and post-processor.

Refer to the [Figure 1-2](#) for the functionality of pre- and postprocessors in finite element analysis.

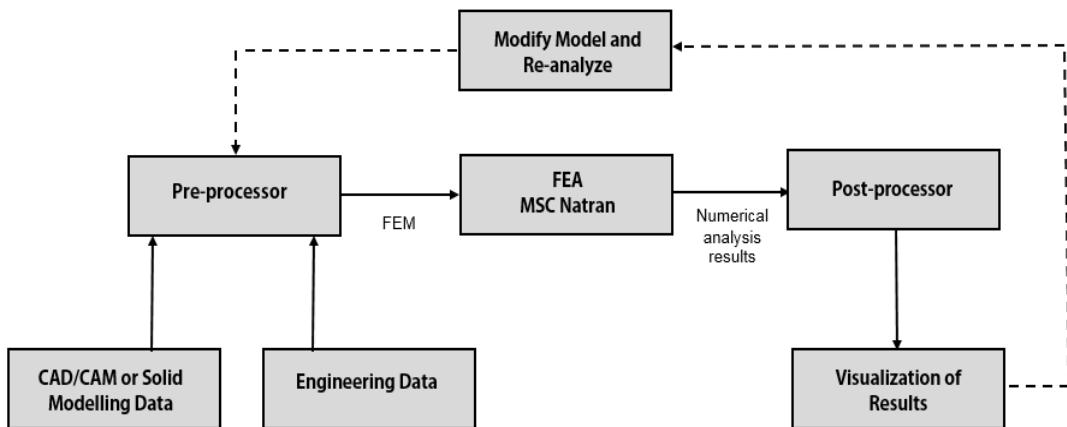


Figure 1-2 Pre- and Postprocessors in FEA

Workflow of MSC Nastran with Patran

Patran is a pre/post-processing software for FEA, providing solid modeling, meshing, analysis setup and post-processing for multiple solvers. Patran provides a rich set of tools that streamline the creation of analysis ready models for linear, nonlinear, explicit dynamics, thermal, and other finite element solutions.

From geometry cleanup tools that make it easy for engineers to deal with gaps and slivers in CAD, to solid modeling tools that enable creation of models from scratch, Patran makes it easy for anyone to create FE models.



Meshes are easily created on surfaces and solids alike using fully automated meshing routines, manual methods that provide more control, or combinations of both. Finally, loads, boundary conditions, and analysis setup for most popular FE solvers is built in, minimizing the need to edit input decks.

Patran has the ability to display, sort, combine, scale, and query in a general way a single results database.

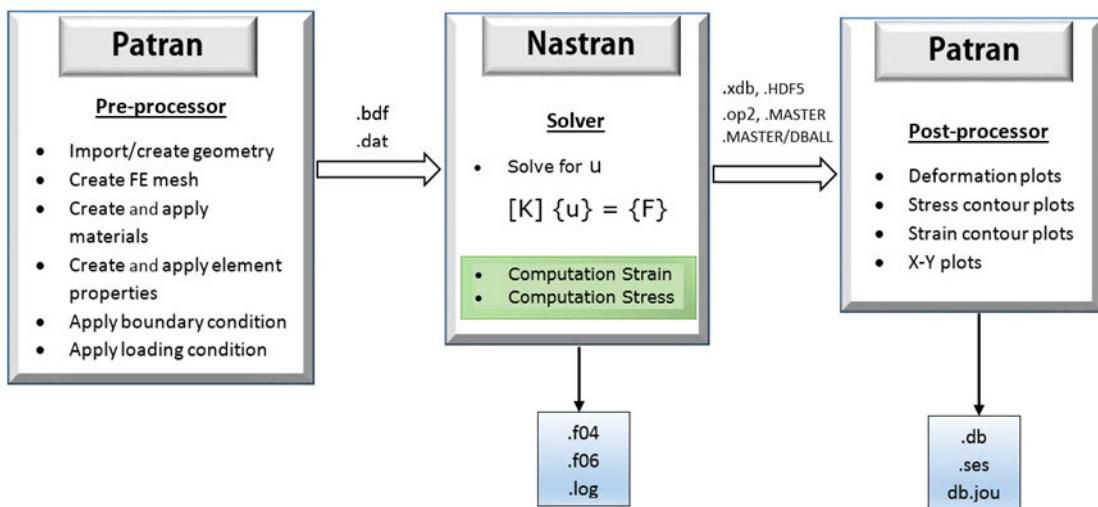


Figure 1-3 Patran and MSC Nastran Workflow and Files

Pre-processor

The job of the pre-processor is to create the model and prepare the geometry and the mesh. It also defines the properties, materials, loads, and the boundary conditions. Then, a text file which includes all these data under a specific standard format is generated and send to the solver.

Patran lets you manage and carry out several phases of the development process in one place. It is a product design model builder, an environment simulator, a finite element analysis manager, and a numerical results interpreter all in one. You can use Patran by itself to complete all of your simulation tasks, or you can use it in conjunction with other CAD software, modeling packages, and analysis codes.

As a preprocessor, Patran does the following:

- **Simulates a Product Design:** The simulation or model must describe the shape and size of the product, describe what materials are used to construct the product, stipulate how the product is assembled, and define the environmental forces that the product endures. The majority of the tools in Patran help you construct this product design model.
- **Converts Product Designs into Geometry Models:** Patran provides its own set of tools for geometry creation and editing, using which you can quickly create parts with 2D and 3D wireframe, surface, and solid geometry. The CAD interface in Patran also allows you to import and edit CAD data from many leading CAD programs.



- **Meshing and Creating Elements:** After importing/creating the geometry, you can create and verify the FEM using a suite of meshing tools—automeshers for curves, surfaces, and solids, as well as mapped meshing and paved surface meshing.
- **Modeling Materials:** You can define the materials for analysis in the Materials application in Patran. A material model is a group of material properties that describe what your model is made of (such as steel or a composite) and the attributes of that material (stiffness, density, and so on). After defining the materials for your model, assign them to model regions.
- **Simulating Forces and Loads:** The FEA tests a particular model's reaction to certain loads and constraints imposed as boundary conditions. Loads are environmental factors such as force, pressure, temperature, and voltage. Boundary conditions are described in terms of degrees-of-freedom, that are the directions in which the edges of the model are free to move in 3D space, along a translational (straight-line) or a rotational path.
- **Analyzing the Model with Environmental Loads:** The analysis stage of the simulation project begins after the product design model phase. You can run a finite element analysis using Patran. You may need to complete several tasks in Patran to format and setup the analysis.
- **Selecting an Analysis Code:** Select an analysis code when you begin each simulation project. Patran assumes that you are using a single analysis code. As you build the product design model, Patran stores the information using the formats and naming conventions of the analysis code. You can change the analysis code at any time during the project. When you do so, Patran attempts to convert all of the data into the new format.
- **Tailoring the Model for a Selected Analysis Code:** You need to define element types (such as beam, shell, etc) and element-related properties for regions of your model, then assign these definitions to geometric or FEM entities. Element type selection is based on the finite element code, the dimensions of the model, and your assumptions about the model's behavior. Additional properties describe attributes such as the thickness of a plate, the spring constant for a spring, an area for a bar element, materials, and so on.
- **Running FEA:** The analysis application provides the link between the Patran environment and the analysis solvers. The analysis application provides the means to:
 - Identify a desired analysis type
 - Define translation and solution parameters
 - Select a sequence of load cases
 - Select desired output
 - Send the model data to the analysis solver
 - Read result quantities from results files
- **Compiling the Analysis Results:** Finite element results generated in the second stage of the project generally take the form of numbers such as the amount of displacement at a point in the model. However, it is difficult to gain a real understanding of how a model behaves by looking at a stack of numbers on paper.



Solver

The job of the solver (here, MSC Nastran) is to do the following:

- Take the input data from the pre-processor
- Process the data using specific algorithms
- Give an output that corresponds to the result of the analysis.

After successful execution of the job, different files are automatically created. The table below lists some of the files that are created after the execution of the job.

.dball:	It contains permanent data for database runs.
.f04:	It contains database file information and a module execution summary.
.f06:	It contains the MSC Nastran analysis results, as listed in A Simple MSC Nastran Model example.
.log:	It contains system information and system error messages.
.master:	It is the master directory for database runs.
.op2	It contains results like displacements, stresses, etc. from the analysis.
.h5	It contains results stored in the modern HDF5 file format.

Note: These are some of the files which are generated, and a complete list is not mentioned.

Post-processing

Visualizing Numerical Results

After execution, analysis results are loaded directly into the Patran relational database and sorted by time step, frequency, temperature, or spatial location. Patran has the ability to visualize results using computer graphics, animation, and other results tools.

The data from the solver (MSC Nastran) is then sent back to the post-processor (Patran) which opens the files and displays the data in a form that will be understandable for the user (contours, graphs, tables, etc.).

The Patran results application gives users control of powerful graphical capabilities to display results quantities in a variety of ways. The results application treats all results quantities in a very flexible and general manner. For maximum flexibility, results can be sorted, reported, scaled, combined, filtered, derived, or deleted.

Workflow of MSC Nastran with SimXpert

SimXpert is an integral component of the enterprise simulation environment. It incorporates direct integration with SimManager and SimDesigner. SimXpert is a multi-disciplinary simulation environment for the analysis including work-spaces between which one common model can be shared.



The work-spaces provide different tools appropriate to the discipline:

- **Structures:** Linear and nonlinear, static, and dynamic Finite Element Analysis (FEA) using MSC Nastran.
- **Motion:** Multi-body dynamics of rigid and flexible bodies using the Adams C++ solver.
- **MD Explicit:** Nonlinear explicit dynamic FEA using MSC Nastran.
- **Template Builder:** Captures simulation procedures consisting of SimXpert commands and macros.
- **Systems and Controls:** Mechanical, electrical, hydraulic, pneumatic, thermal, gas dynamics, powertrain, vehicle dynamics analysis using Easy5.

All solvers are included in SimXpert. Workspaces filter the simulation model and only the parts of the model that have relevance to a workspace are visible.

Refer to [Figure 1-4](#) for the SimXpert and MSC Nastran workflow and files:

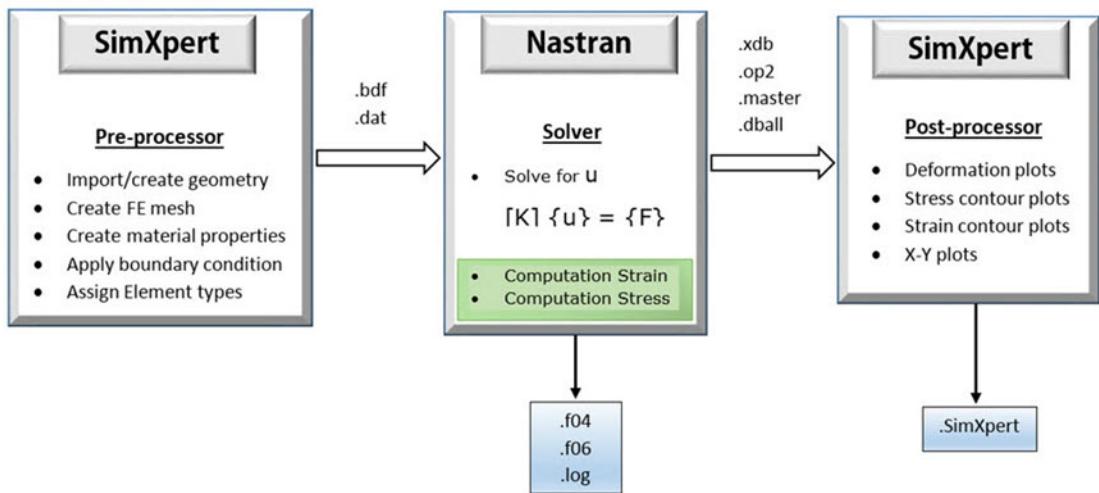


Figure 1-4 SimXpert and MSC Nastran Workflow and Files

The FEA process is completed in three stages with SimXpert:

1. **Pre-processing:** This is the first stage where you create the model with all its properties. After creating the entire model you create a Nastran job and export the file to a defined location. Creating a Nastran job adds all the elements and properties in a file called as the input file/deck (which is understood by Nastran).
2. **Solver:** After the input file (.bdf) is generated by the SimXpert, there are two ways to execute the problem. One is by clicking *Run* Option in SimXpert, and second way is by externally running Nastran. According to the input values provided, the solver (Nastran) generates result files and output files.
3. **Post-processing:** The generated output files are then attached to the model using SimX UI.



Capabilities of MSC Nastran

MSC Nastran uses FEA to solve static and dynamic problems. The numerical method comes in two types, both included in MSC Nastran

- Linear FEA
- Nonlinear FEA

The nonlinear FEA type can be solved in one of two ways: implicit or explicit. More specific capabilities that are provided with MSC Nastran are aero-elasticity, thermal analysis, and fatigue life problems. In addition, MSC Nastran provides a number of methods to use in achieving an optimal design based on the user's chosen criteria.

Table 1-1 Matrix of MSC Nastran Capabilities

FEA Type	Capabilities	
	Linear	Nonlinear
	<ul style="list-style-type: none"> • Material stress-strain behavior is linear and elastic • Deformation and/or rotations are small • The boundary conditions remain unchanged 	<ul style="list-style-type: none"> • Material stress-strain behavior is nonlinear elastic, plastic and hyperelastic • Deformation and/or rotations can be large • The boundary conditions can be changed
Static	<ul style="list-style-type: none"> • Statics • Buckling (stability) 	<ul style="list-style-type: none"> • Non-linear Statics
Dynamic	<ul style="list-style-type: none"> • Real (normal modes) and complex Eigenvalue • Transient Response • Frequency Response 	<ul style="list-style-type: none"> • Transient Response • Frequency Response
Thermal	<ul style="list-style-type: none"> • Temperature dependent conductivity 	<ul style="list-style-type: none"> • Free convection boundaries

For more information refer to [Chapter 2: MSC Nastran Capabilities](#).



Benefits of using MSC Nastran

The benefits of using MSC Nastran are as follows:

- Accelerate innovation through improved event simulation
- Design optimization to achieve reduced material costs
- Deliver products that meet certification and safety requirements
- Reduce risk by using simulation to save time and cost



2

MSC Nastran Capabilities

- General Capabilities 34
- Linear Analysis 36
- Dynamic Analysis 42
- Nonlinear Analysis 48
- Design Sensitivity and Optimization 56
- Aeroelasticity 59
- Thermal Analysis 61
- Composite Materials 62
- Fluid-Structure Interaction 62
- Direct Matrix Abstraction Program 63
- Fatigue Analysis 64
- High Performance Computing 68

General Capabilities

MSC Nastran is optimized across multiple, integrated disciplines with the power to handle large-scale problems and to take advantage of the latest high performance computing hardware. MSC Nastran's focus on integrated and multidisciplinary solutions helps you achieve across-the-board efficiency by driving early design validation, improved product performance and rapid insight into product lifecycle performance.

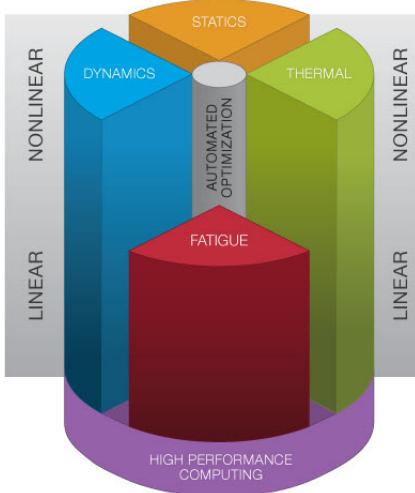


Figure 1-1 Multidisciplinary Structural Analysis: Linear or Nonlinear Static

Some of the general capabilities of MSC Nastran are as follows:

- Optimized for component, system and assembly level simulations
- Strength, durability and vibrations assessment of structures
- Structural dynamic response simulation of loads that vary with time or frequency
- Automated Component Modal Synthesis (ACMS) for large modal based analyses and NVH solutions
- Simulation of interior and exterior acoustics for coupled structural acoustic analysis
- Static and transient analysis of structures involving material, geometric and boundary condition nonlinearities
- Linear and nonlinear contact analyses with intuitive contact definitions
- Heat transfer analysis with contact including conduction, convection and radiation
- Failure analysis of structures and composites
- Rotor dynamic stability studies of rotating machinery
- Aeroelastic analysis



- Efficient optimization using sizing, shape and topology optimization with manufacturing constraints
- Efficient reduction in iterations for large models and data security while sharing using Superelement techniques
- Enhanced iterative and in-core sparse solvers
- Multi-model optimization to simultaneously optimize two or more structural models
- Stochastic simulation
- Use Graphics Processing Units (GPUs) for improved solver efficiency
- Add customized element formulations, materials, contact definition, and more with User Defined Services (UDS)
- Powerful Direct Matrix Abstraction Program (DMAP) options to built an own program or customize existing modules

Solution Sequence (SOL)

An MSC Nastran solution sequence (SOL) is a pre-programmed 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 the following [Table 1-1 on page 35](#).

Table 1-1 Commonly used 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



Table 1-1 Commonly used Solution Sequences

SOL Number	SOL Name	Description
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
190	DBTRANS	Database Transfer
200	DESOPT	Design Optimization
400	NONLIN	Nonlinear Static and Implicit Transient Analysis and all linear sequences from statics, modes, frequency, and transient inclusive with perturbation analysis based on previous nonlinear analysis
600	SESTATIC	MSC Nastran API into MSC MARC nonlinear
700	NLTRAN	Nonlinear Explicit Transient Analysis

Linear Analysis

A linear static analysis is an analysis where there is a linear relation between applied forces and displacements. In practice, this is applicable to structural problems where stresses remain in the linear elastic range of the used material. In this case, the response of the structure (deformation, stress and strain) is linearly proportional to the magnitude of the load (force, pressure, moment, torque, temperature etc.). In short, linear analysis does not consider any change in stiffness matrix.

Solution Flowchart

A flowchart summarizing what MSC Nastran does in solving a linear static structural problem is shown in [Figure 1-2](#).



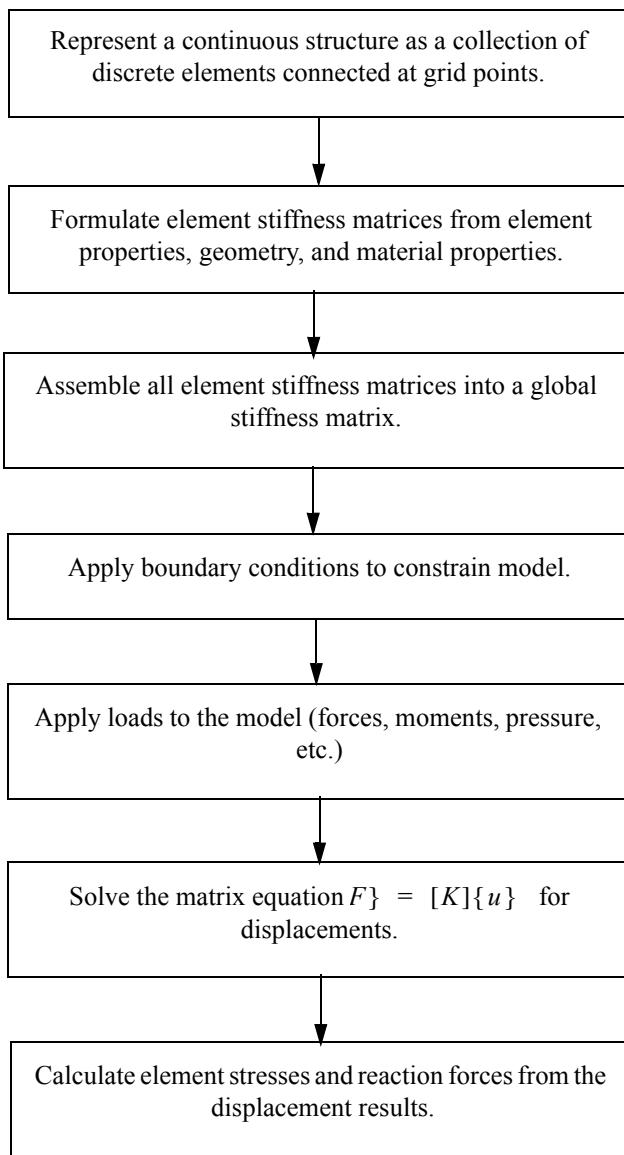


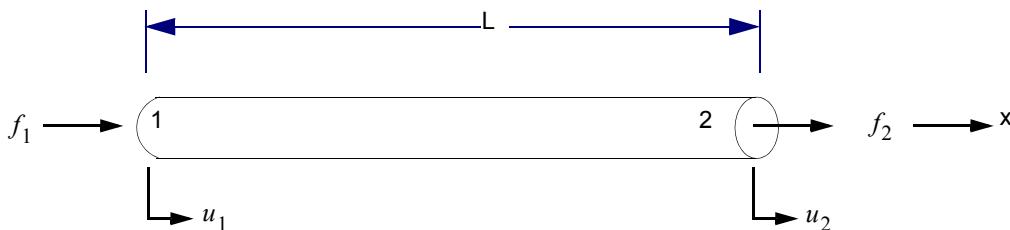
Figure 1-2 MSC Nastran Solution Flow

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 1-3](#). The rod is subjected to end loads (cross-sectional loads) and is in static equilibrium.



[Figure 1-3](#) 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

$$f_2 = -f_1 \quad (2-1)$$

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:

$$\epsilon_x = \frac{\delta}{L} = \frac{u_2 - u_1}{L} \quad (2-2)$$

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

$$\sigma_x = E\epsilon_x \quad (2-3)$$

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



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

and

$$\text{Grid 2: } \sigma_x = +\frac{f_2}{A} \quad (2-4)$$

Force can now be related to displacement by substituting equations (2-2) and (2-3) into equation (2-4). For grid point 1,

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

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

$$-f_1 = \frac{EA}{L}u_2 - \frac{EA}{L}u_1 \quad (2-5)$$

Similarly, for grid point 2,

$$f_2 = \frac{EA}{L}u_2 - \frac{EA}{L}u_1 \quad (2-6)$$

Equations (2-5) and (2-6) represent two linear equations in two unknowns. In matrix form, equations (2-5) and (2-6) can be expressed as

$$\begin{pmatrix} f_1 \\ f_2 \end{pmatrix} = \underbrace{\frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}}_{[k]} \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} \quad (2-7)$$

or

$$\{f\} = [k]\{u\} \quad (2-8)$$

where



- $[k]$ = element stiffness matrix
- $\{f\}$ = vector of element forces
- $\{u\}$ = vector of element displacements

$[k]$ is the individual rod's stiffness matrix and equation (2-8) 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 1-4](#).

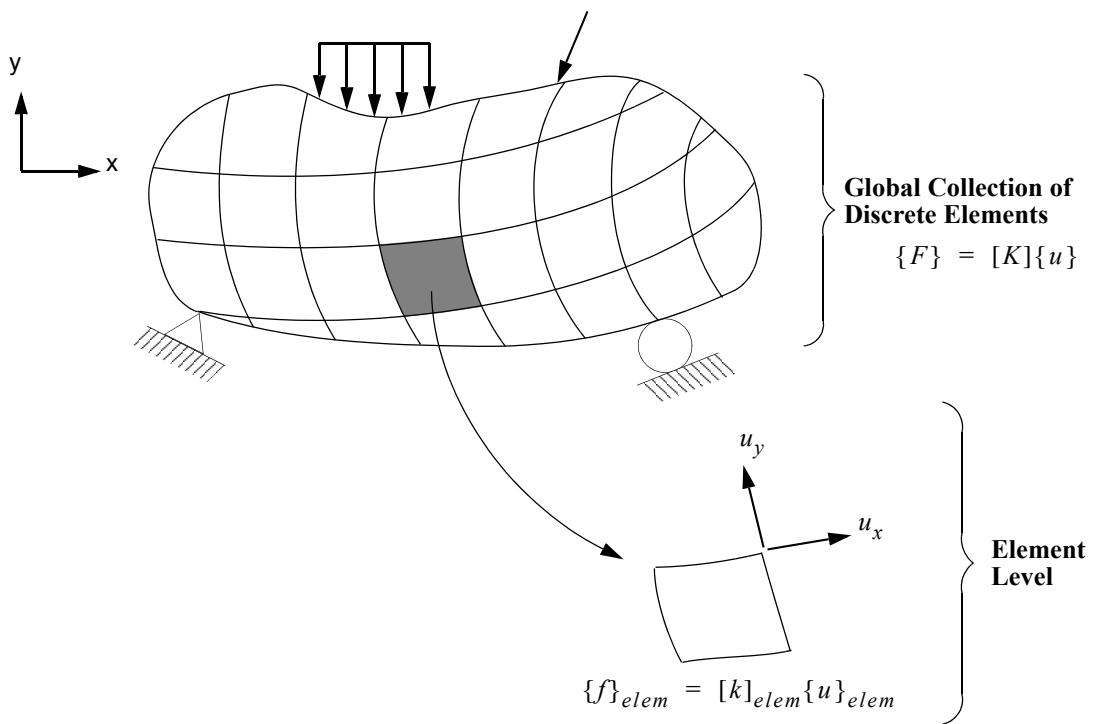


Figure 1-4 The Generalized Finite Element Model

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 .

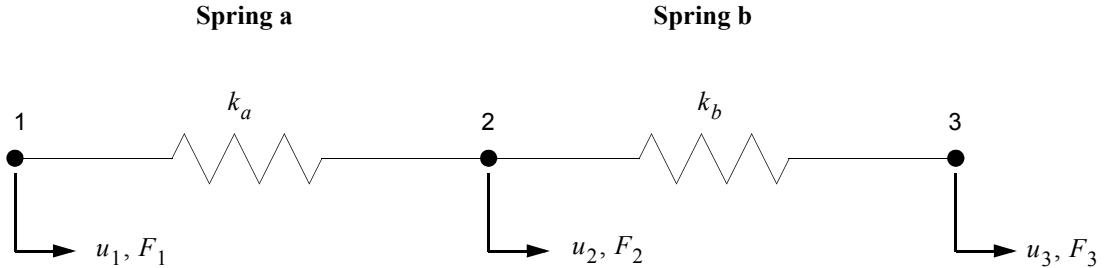
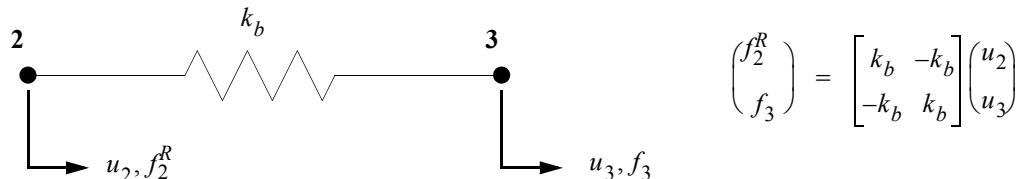
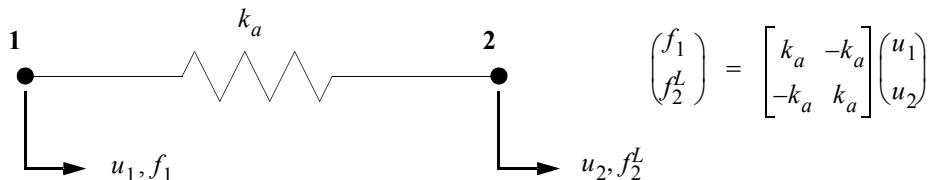


Figure 1-5 Structural System with Two Axial Springs

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 equation (2-7) to obtain an element matrix equation for each model.



Combine these last two equations by adding them as follows:

$$\begin{pmatrix} f_1 \\ f_2^L + f_2^R \\ f_3 \end{pmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix} \quad (2-9)$$

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:



$$\begin{pmatrix} F_1 \\ F_2 \\ F_3 \end{pmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix} \quad (2-10)$$

or

$$\{F\} = [k]\{u\} \quad (2-11)$$

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.

Assumptions for Linear Static Analysis

- All loads are applied gradually and slowly until they reach their full magnitude
- After reaching full magnitude the loads remain constant
- Inertial and damping forces to small velocities and accelerations are neglected

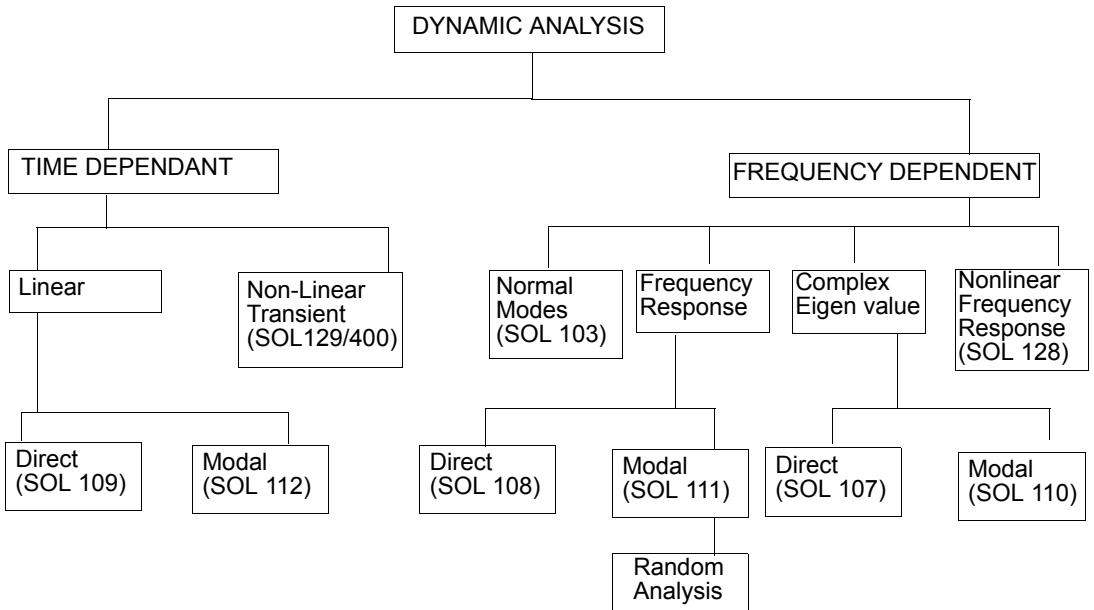
You can make linearity assumption only in the following conditions:

- All material in the model comply with Hooke's Law
- The induced displacements are so small that they cause negligible change in the geometric and material properties and hence the stiffness
- The structure subjected to loading has negligibly small accelerations and velocities
- The boundary conditions doesn't change during loading.

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 deals with a range of capabilities for dynamic analysis which are classified as follows:





Normal Modes Analysis

Normal modes analysis determines the natural frequencies and mode shapes of a structure. The natural frequencies of a structure are the frequencies at which the structure naturally tends to vibrate if subjected to any disturbance. For example, the strings of a piano which are each tuned to vibrate at a specific frequency.

Other forms for natural frequency include the following:

- Characteristic frequency
- Fundamental frequency
- Resonance frequency
- Normal frequency

The deformed shape of a structure when vibrates 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 the [Figure 1-6](#)



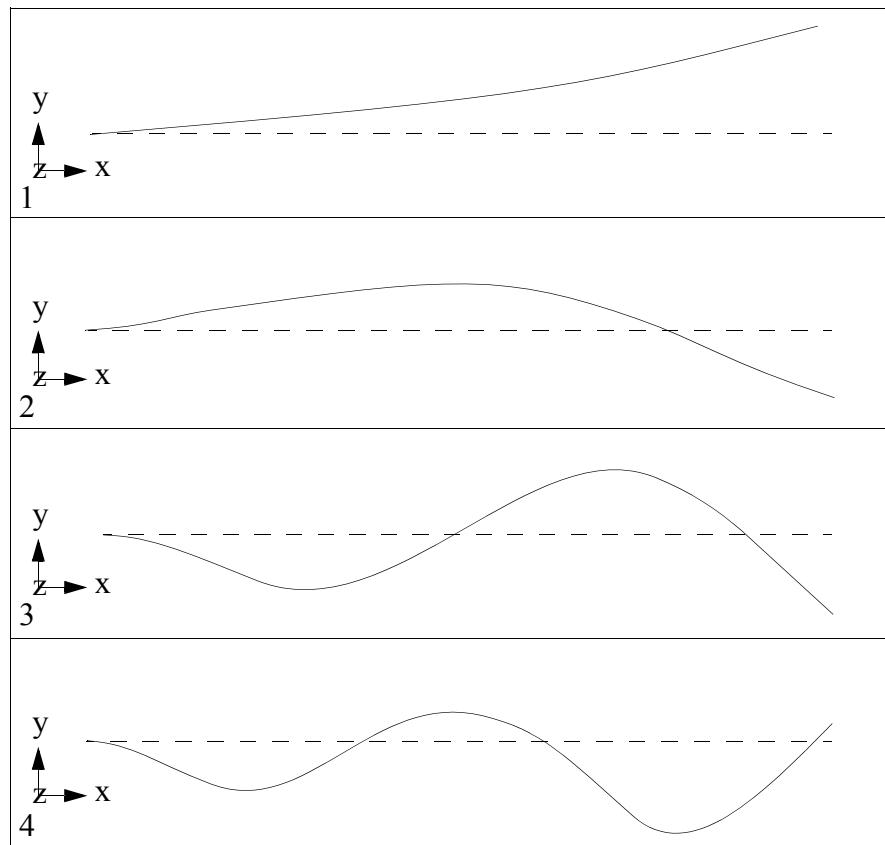


Figure 1-6 The First Four Mode Shapes of a Cantilever Beam



Frequency Response Analysis

Frequency response analysis method measures structural response to steady-state oscillatory excitation. For example, an equipment which rotates at a set frequency but has an imbalance. In frequency response analysis, the excitation is clearly defined in the frequency domain, and all of the applied forces are known at each forcing frequency.

Note: 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 consists of:

- Grid point displacements, velocities and accelerations
- Element forces and stresses

The measured responses are complex (number) magnitude and phase (with respect to the applied force) or as real and imaginary components. For example, [Figure 1-7](#) shows the displacement magnitude of a grid point vs. frequency.

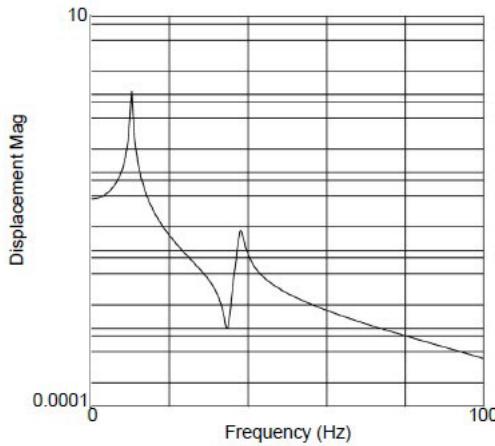


Figure 1-7 Frequency and Response Analysis



Two different numerical methods can be used in frequency response analysis, which are described as follows:

- 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.

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 determine the behavior of a structure subjected to time-varying excitation. The transient excitation is explicitly defined in the time domain. The displacement response of a grid point versus time is shown in [Figure 1-8](#).

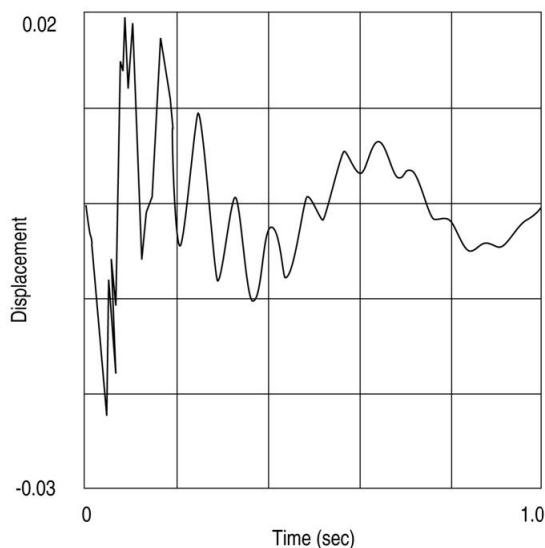


Figure 1-8 Displacement Response from Transient Response Analysis

Two different numerical methods can be used for a transient response analysis which depends on the structure and the nature of the loading, which are as follows:

- Direct transient response analysis: Performs a numerical integration of the complete coupled equations of motion.
- Modal transient response analysis: This 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.



Complex Eigen analysis

Complex Eigenanalysis calculates the damped modes of structures, to assess the stability of systems modeled with transfer functions (including servomechanisms and rotating systems).

The governing equation is similar to normal modes analysis except, the damping has been added. In addition, the mass and stiffness matrices may be unsymmetric, and they can consist of complex elements.

Response Spectrum Analysis

Response spectrum analysis (RSA) is a linear-dynamic statistical analysis method which measures the contribution from each natural mode of vibration to indicate the likely maximum seismic response of an essentially elastic structure.

Response-spectrum analysis provides insight into dynamic behavior by measuring pseudo-spectral acceleration, velocity, or displacement as a function of structural period for a given time history and level of damping. It is practical to envelope response spectra such that a smooth curve represents the peak response for each realization of structural period.

Response-spectrum analysis is useful for design decision-making because it relates structural type-selection to dynamic performance. Structures of shorter period experience greater acceleration, whereas those of longer period experience greater displacement. Structural performance objectives should be taken into account during preliminary design and response-spectrum analysis.

From a response spectrum, when you assume that 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) with the use of such methods as Square Root of the Sum of the Squares (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
- Absolute acceleration

Component Mode Synthesis

Component Mode Synthesis (CMS) is a form of superelement reduction wherein the 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

This analysis enables you to determine the response of structures to vibration loads that are random in nature. An example would be the response of a sensitive electronic component mounted in a car subjected to the vibration from the engine, pavement roughness, and acoustic pressure.

Few examples of random vibration include the following:

- Aerospace and electronic packaging components subject to engine vibration
- Turbulence and acoustic pressures
- Tall buildings under wind load
- Structures subject to earthquakes
- Ocean wave loading on offshore structures

The above random excitations are usually described in terms of a Power Spectral Density (PSD) function. Random response output consists of the following:

- Response power spectral density
- Autocorrelation functions
- Number of zero crossings with positive slope per unit time
- Root Mean Square (RMS) values of response

Nonlinear Analysis

Nonlinear analysis becomes necessary when the stiffness of the part changes under its operating conditions. All physical processes are inherently nonlinear to a certain extent. For example, when you stretch a rubber band, it gets harder to pull as the deflection increases; or when you flex a paper clip, permanent deformation is achieved. Several common every day applications like these exhibit either large deformations and/or inelastic material behavior. Failure to account for nonlinear behavior can lead to product failures, safety issues, and unnecessary cost to product manufacturers.

Nonlinear response could be caused by any of several characteristics of a system, like large deformations and strains, material behavior or the effect of contact or other boundary condition nonlinearities. In reality many structures exhibit combinations of these various nonlinearities.

MSC Software provides solutions to help you simulate accurately and efficiently systems with any or all of the nonlinearities, with applications encompassing multiple industries.

Table 1-2 Types of Nonlinear Simulations

Nonlinear Simulations	
Large displacement and large strain analysis	Manufacturing processes like forming, extrusion, welding, curing and more
Post-buckling analysis	Bolt modeling
Plasticity and viscoplasticity	Material damage
Creep and stress relaxation	Fracture and crack propagation



Table 1-2 Types of Nonlinear Simulations

Nonlinear Simulations	
Nonlinear elasticity	Material wear
Shape memory alloys	Pyrolysis and abrasion
Powder metallurgy	Multiphysics
Hyperelasticity	General large displacement 3D contact
Thermal variation of material properties	Friction analysis
Electrical-thermal-structural coupling	Automatic remeshing

The following table gives you a basic understanding on when to consider nonlinear analysis:

Table 1-3 Considerations for Nonlinear Analysis

Application	Explanation
Strength analysis	How much load can the structure support before failure
Deflection analysis	When deflection control is of primary importance
Stability options	Finding critical points (limit points or bifurcation points) closest to operational range
Service configuration analysis	Finding operational equilibrium form of certain slender structures when the fabrication and service configurations are quite different (e.g cables, inflatable structures, helicoids)
Reserve strength analysis	Finding the load carrying capacity beyond critical point to assess safety under abnormal conditions
Progressive failure analysis	A variety of stability and strength analysis in which progressive deterioration
Envelope analysis	A combination of previous analyses in which multiple parameters are varied and the strength information thus obtained is condensed into failure envelopes

Linear static problems are solved in one step-a single decomposition of the stiffness matrix. Structures whose stiffness is dependent on the displacement which they may undergo are termed geometrically nonlinear. Geometric nonlinearity accounts for phenomena such as the stiffening of a loaded clamped plate, and buckling or 'snap-through' behavior in slender structures or components.

The nonlinear behaviour occurs as stiffness and loads become functions of displacement or deformation $[K]\{D\}=\{R\}$, both the structural stiffness matrix $[K]$ and possibly the load vector $\{F\}$ becomes functions of the displacements $\{D\}$. Hence, it is not possible to solve immediately $\{D\}$ as in the case of $[K]$ and $\{R\}$.



Nonlinear problems of any type require iterative solution methods and incremental loading to get (converge to) a solution, and are generally far more computationally difficult than linear problems. Nonlinear problems are classified into three broad categories as follows:

- Geometric nonlinearity: Nonlinear strain-displacement relation
- Material nonlinearity: Nonlinear constitutive relation
- Kinematic nonlinearity: Non-constant displacement BCs, contact
- Force nonlinearity: Follow-up loads

MSC Nastran can also handle nonlinear problems that are time dependent (transient). For more information, refer to the [Nonlinear User's Guide SOL 400](#) or the [Explicit Nonlinear \(SOL 700\) User's Guide](#).

Material Nonlinearity

Linear analysis assumes a linear relationship between stress and strain. Material nonlinearity results from the nonlinear relationship between stress and strain. In addition, large strain can influence the material behavior.

Material nonlinearity is associated with the inelastic behavior of a component or system. Inelastic behavior may be characterized by a force-deformation (F-D) relationship, also known as a backbone curve, which measures strength against translational or rotational deformation.

The general F-D relationship shown to the right indicates that once a structure achieves its yield strength, additional loading will cause response to deviate from the initial tangent stiffness (elastic behavior). Nonlinear response may then increase (hardening) to an ultimate point before degrading (softening) to a residual strength value.

Nonlinear stress-strain relationships are a common cause of nonlinear structural behavior. For any material besides steel, reviewing the stress-strain curves is the best way to understand the nonlinearity of the problem. Even if you use a linear material model, knowing the nonlinearity is important for interpreting results.

Many factors can influence the stress-strain properties of a material, including the following:

- Load history (as in elasto-plastic response)
- Environmental conditions (such as temperature)
- The amount of time that a load is applied (as in creep response).



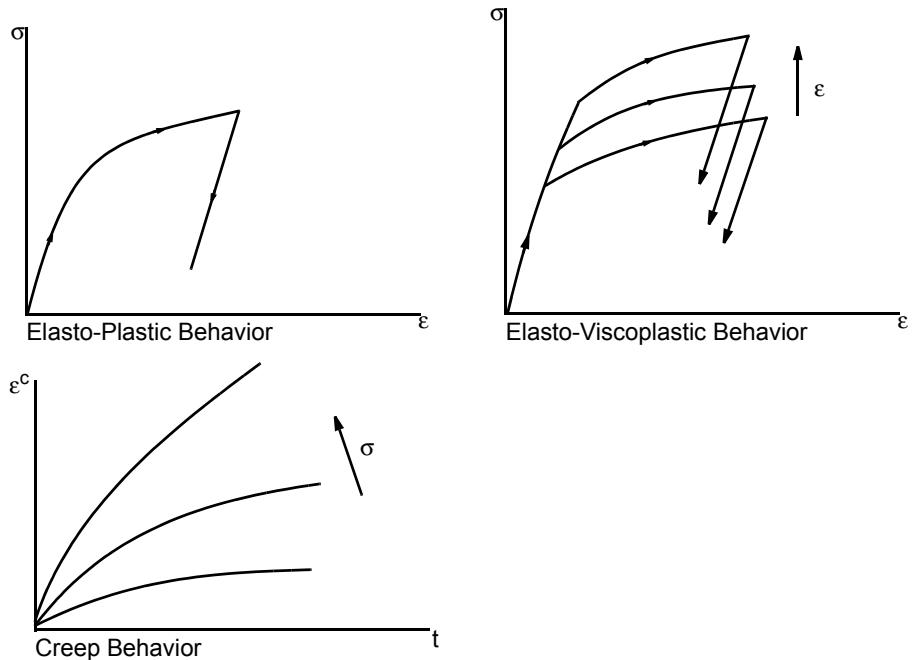


Figure 1-9 Material Nonlinearity in Different Types of Materials

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 you. Three choices for the definition of subsequent yield surfaces are available in SOL 400 as follows:

- Isotropic hardening
- Kinematic hardening or combined isotropic
- Kinematic hardening

Most plastic material behavior, with or without the Bauschinger effect, can be modeled.

Geometric Nonlinearity

Structures whose stiffness is dependent on the displacement which they may undergo are termed geometrically nonlinear. Geometric nonlinearity accounts for phenomena such as the stiffening of a loaded clamped plate, and buckling or snap-through behavior in slender structures or components.

Important: In most engineering problems, the deformations are so small that the deviation from the original geometry is not perceptible. The small error introduced by ignoring the



deformations does not guarantee the added mathematical complexity generated by a more sophisticated theory. This is the reason behind a vast majority of analyses which are made with an assumption of geometric linearity.

In a geometry nonlinear solution, deflections of the structure are large when compared with the original dimensions of the structure. Changes in stiffness and loads occur as the structure deforms. In a geometric nonlinear problem, you can simulate the following:

- Large deflections
- Deflection-dependent loads
- Deflection-dependent stiffness

Geometric nonlinearity is a result of the nonlinear relationship between strain and displacement on the one hand and the nonlinear relation between stresses and forces on the other hand.

If the stress measure is conjugate to the strain measure, both sources of nonlinearity have the same form. This type of nonlinearity is mathematically well defined, but often difficult to treat numerically.

If a structure experiences large deformations, its changing geometric configuration can cause the structure to respond nonlinearly. An example would be the fishing rod shown in Figure. Geometric nonlinearity is characterized by large displacements and/or rotations.

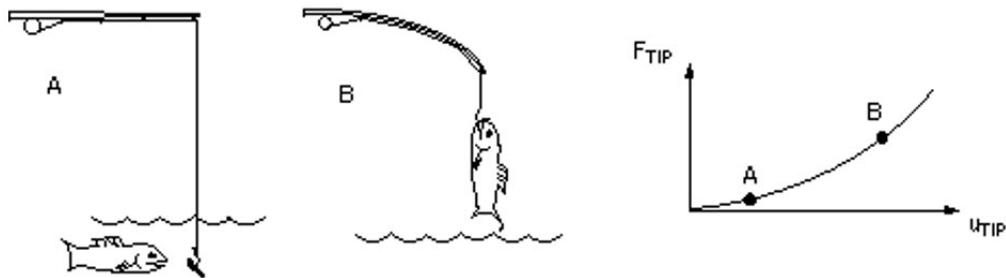


Figure 1-10 Example of Nonlinearity

Three important types of geometric nonlinearity occur:

- Problems where large rotation occur
- The analysis of buckling and snap-through problems (see the figures below)
- Large strain problems such as manufacturing, crash, and impact problems. In such problems, due to large strain kinematics, the mathematical separation into geometric and material nonlinearity is not unique



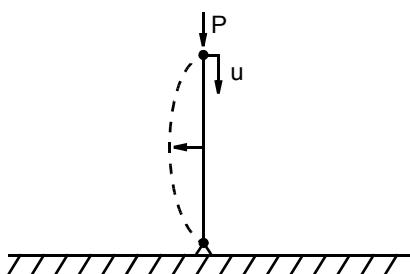


Figure 1-11 Buckling

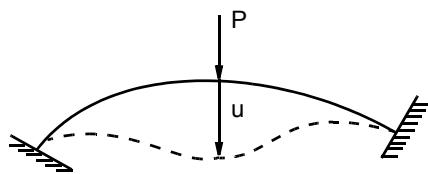
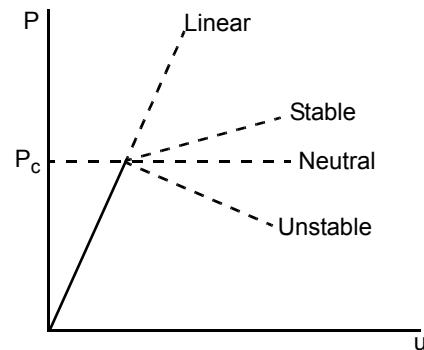
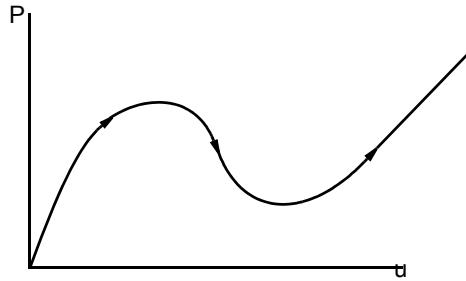


Figure 1-12 Snap-Through



Nonlinear Boundary Conditions

A model exhibits boundary nonlinearity when the loads, constraints, or load paths change throughout the solution. If the orientation, distribution, or magnitude of applied loads or the load path changes as loading is increased, a nonlinear model may be required.

- Conservative, as in the case of a centrifugal force field

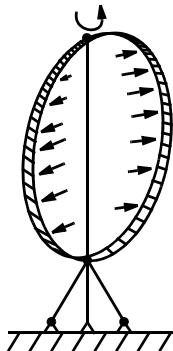


Figure 1-13 Centrifugal Load Problem (Conservative)

- Non conservative, as in the case of a follower force on a cantilever beam (the following figure)

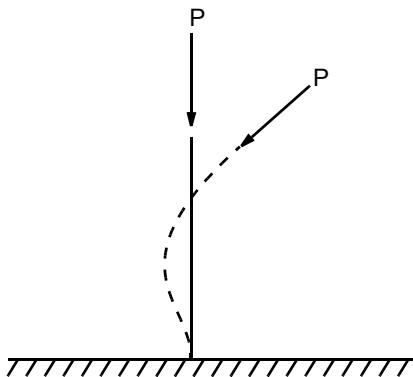


Figure 1-14 Follower Force Problem (Non conservative)



- Locally nonconservative, but represent a conservative loading system when integrated over the structure. A pressurized cylinder is an example of this.

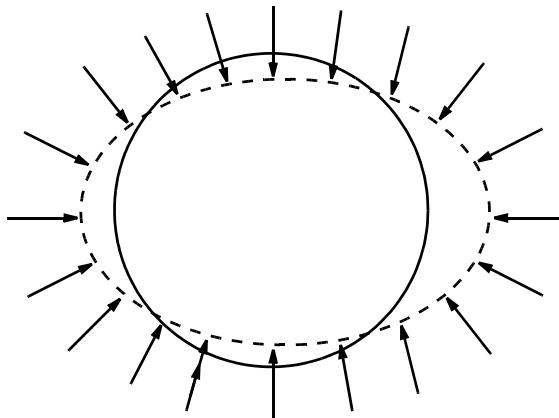


Figure 1-15 Pressurized Cylinder (Globally Conservative)

Loads on a structure cause nonlinearity if they vary with the displacements of the structure. If there is a change in constraints due to contact during loading, the problem may be classified as a boundary nonlinear problem and would require the use of BCTABLE/ BCTABLE1, BCNECT, BCONPRG, BCONPRP, BCBODY, or BSURF bulk data entry options. CGAP elements would have been used in the traditional nonlinear sequences of SOL 106 or 129; however, the use of GAP elements is not recommended in SOL 400.

The most common boundary nonlinearities are:

Contact

Contact conditions model the interaction of two separate parts or different surfaces on the same part. Boundary conditions such as surface contact are generally regarded as nonlinear. However, a new trend has emerged lately that allows a contact analysis to run in a linear solution in some FEA applications. In deciding between a linear and nonlinear contact analysis it is best to ask these questions:

- Are there large movements in the model or any of the other nonlinear effects mentioned above?
- Is there significant sliding between contact bodies in the model? Is the contact solution path dependent (for instance a snap-fit)?
- Are detailed contact stresses needed in the model?

If the answer is yes to any of the three questions above, it is generally recommended to run a nonlinear solution to get the best accuracy.

Follower forces

This nonlinear effect simply means that the direction of the forces move with the deformations or movement of the part. Pressure loads are a perfect example of follower forces since they always act



normal to a surface. As a part deforms, follower forces will adjust the direction of the loads to ensure they stay normal to the surface.

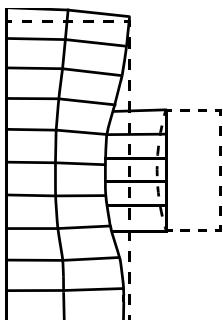


Figure 1-16 Contact and Friction Problem

Nonlinear Transient Analysis

Nonlinear transient analysis should be used if the effects of inertia, damping, and transient loading are significant. Additionally, quasi-static models that undergo buckling or other instable loading conditions will often converge better in a nonlinear transient analysis due to the inertia effects keeping the model stable.

A nonlinear transient analysis requires both dynamic and nonlinear setup steps.

Design Sensitivity and Optimization

Design sensitivity and optimization are used when you modify a design whose level of structural complexity exceeds our ability to make appropriate design changes. 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 an 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 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.



Application Examples

As a simple example of design optimization, consider the cantilevered I-beam shown below.

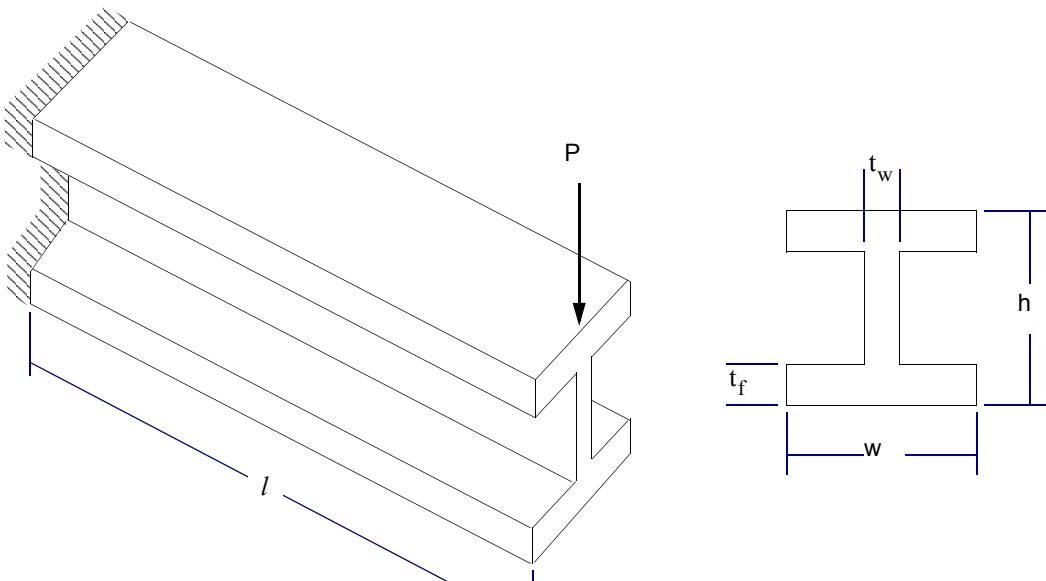


Figure 1-17 Design Optimization in Cantilevered I-Beam

A frequent task of the designer is to minimize the total structural weight (thereby reducing the cost of materials) while making sure 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 analyses, the corresponding cross-sectional properties (A , I_1 , I_2 , 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 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 explain in brief 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.
- The various types of optimizations which can be carried out in Nastran are:
 - Sizing Optimization
 - Shape Optimization
 - Topology Optimization



Aeroelasticity

MSC Nastran provides efficient solutions for the problems related to 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 factors 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 analyses 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

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 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 encountered at a relative airspeed, where 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 listed below:



1. American flutter method (called the K-method in MSC Nastran),
2. Efficient K-method (called the KE-method) for rapid flutter evaluations,
3. 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 method determines 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 given below:

- Power spectral density
- Root mean square response
- 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 available for flutter analysis. Different types of Aerodynamic methods are listed below.

- Subsonic methods:
 - a. Doublet-Lattice Method with body interference
 - b. Strip Theory
- Supersonic methods:
 - a. Mach Box Method
 - b. Piston Theory
 - c. (Aero II option) multiple interfering surface method called ZONA51

Note: The Static and Dynamic Aeroelastic Response solutions uses both subsonic Doublet-Lattice and supersonic ZONA51 aerodynamics methods.



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 analyzes free and forced convection behavior when thermal communication between a conduction-dominated solid structure and its surroundings take place. Convection boundary conditions allows heat transfer between the structure and the surrounding environment due to fluid motion.

Free convection or Natural convection is a mechanism, or type of heat transport, in which the fluid motion is not generated by any external source (like a pump, fan, suction device, etc.) but only by density differences in the fluid occurring due to temperature gradients. 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
- Internal heat generation loads

Transient Thermal Analysis

This 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.



Composite Materials

The term composite refers to an engineering material that is made up of more than one material. Composite materials for construction, engineering, and other similar applications are formed by combining two or more materials in such a way that the constituents are still distinguishable and not fully blended. 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.

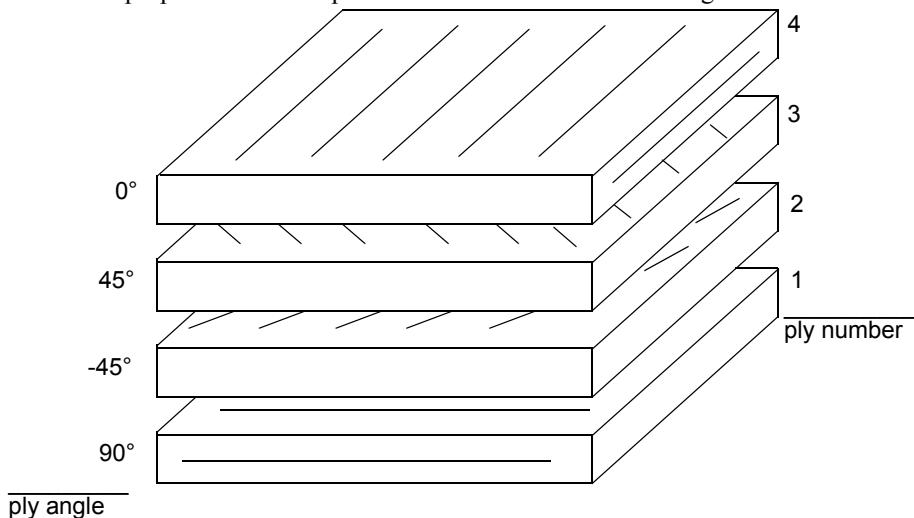


Figure 1-18 Four Ply Example

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 shell 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/stress) to predict ply failures. Nonlinear progressive failure analysis may also be performed.

Fluid-Structure Interaction

MSC Nastran performs fully coupled fluid-structure analysis, and this analysis is primarily applied in the areas of acoustic and noise control analysis. Such as, in the design of passenger compartments of automobiles and aircraft.

The methodology used fluid-structure analysis is called as Pressure Method, which is similar to the Displacement Method in structural analysis, where pressures instead of displacements are calculated at the



fluid points. The velocities and accelerations of the fluid points are similar to forces in structural analysis. The fluid can be modeled with existing three-dimensional elements as follows:

- CHEXA
- CPENTA
- CTETRA

The above elements can adopt to 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 elements are available for acoustic analysis.

- The barrier (CHACBR)
- Absorber (CHACAB)

Note: The absorber and barrier elements can be used to analyze acoustic noise control devices.

The interface between the fluid and the structure can be modeled so that the grid points of the fluid are similar with those of the structure. This is called a matching mesh. If not, it is called as non-matching mesh. In either case by default, coupling for the stiffness and mass is automatically computed.

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.

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.

Direct Matrix Abstraction Program

The Direct Matrix Abstraction Program (DMAP) is a macro language used to build and modify solution sequences in MSC Nastran. The DMAP language contains powerful matrix instructions as well as flexible scalar variable (parameter) operations. The File Management Section (FMS) is used for the attachment, initialization, and manipulation of databases. Details are presented on the structured solution sequence (SOLs 100 through 200), DMAP structure, and NDDL.

A DMAP program consists of a series of functional blocks called modules, each of which has a unique name and a specific function. Modules are executed sequentially; branching and looping operations are performed by DMAP control statements. Modules communicate through the MSC Nastran Executive System (NES) via logical collections of data called *data blocks* and *parameters*.

For additional information, please refer to DMAP Programmer's Guide



Fatigue Analysis

Very often the terms **fatigue**, **fracture**, and **durability** are used interchangeably. However, each of them convey a specific meaning.

- **Fatigue:** This is a failure under a repeated or otherwise varying load which never reaches a level sufficient to cause failure in a single application. It can also be thought of as the initiation and growth of a crack, or growth from a pre-existing defect, until it reaches a critical size, such as separation into two or more parts.

Fatigue analysis itself usually refers to one of two methodologies: either the **Stress-Life (S-N)** or S-N method, commonly referred to as **Total Life** since it makes no distinction between initiating or growing a crack, or the **Local Strain** or **Strain-Life (e-N)** method, commonly referred to as the **Crack Initiation** method which concerns itself only with the initiation of a crack.

- **Fracture:** A fracture is the growth or propagation of a crack once it has been initiated. Fracture also denotes sudden breakage of a component or structure in two. However for the purposes of this manual it refers to the life prediction method of crack growth as implemented using LEFM.

MSC Nastran fatigue analysis concerns itself only with the prior two types of fatigue analysis and is not applicable for crack growth or propagation. For this capability refer to MSC Nastran's cohesive zone modeling and/or virtual crack closure technique (VCCT) or MSC Fatigue, which uses a **LEFM** method for crack growth prediction.

- **Durability:** It is the conglomeration of all aspects that effect the life of a product and usually concerns itself with much more than just fatigue and fracture, but also loading conditions, environmental concerns, material characterizations, and testing simulations to name a few. A true product durability program in an organization that takes all of these aspects and more into consideration.

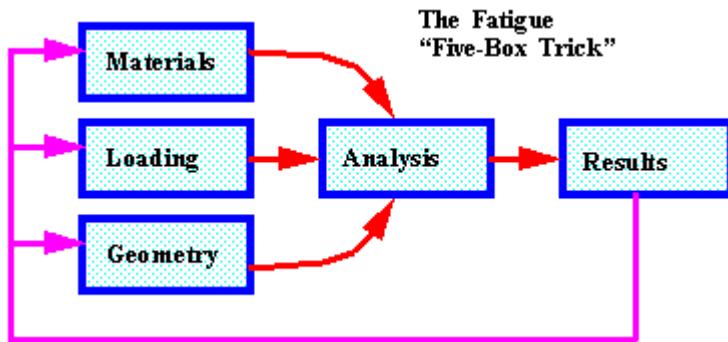
Durability is then the conglomeration of all aspects that affect the life of a product and usually involves much more than just fatigue and fracture, but also loading conditions, environmental concerns, material characterizations, and testing simulations, to name a few. A true product durability program in an organization takes all of these aspects (and more) into consideration.

Note: Fatigue cracks initiate and grow as a result of cyclic plastic deformation. Without plasticity there can be no fatigue failure. All attempts are made in this guide to explain how plasticity is taken into account when determining fatigue life from linear elastic finite element analysis.

Five Box Trick of Fatigue Analysis

These fatigue analysis example exercises are constructed around the concept of the fatigue “five-box trick.” The illustration below depicts this well. For any life analysis whether it be fatigue or fracture there are always three inputs. The first three boxes are the inputs; box four is the analysis; and box five contains the results.



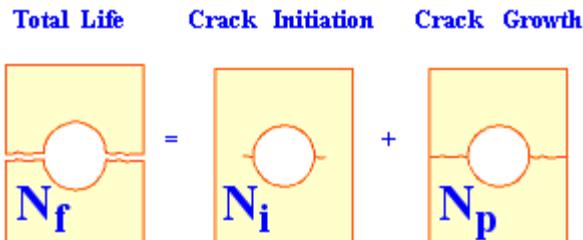


- 1. Cyclic Material Information:** Materials behave differently when they are subject to *cyclic* as opposed to *monotonic* loading. **Monotonic Properties** are the result of material tests where the load is steadily increased until the test coupon breaks. **Cyclic Properties** are obtained from material tests where the loading is reversed and cycled until failure at various load levels. These parameters differ depending on the fatigue analysis type involved.
- 2. Service Loading Information:** The proper specification of the variation of the loading is extremely important to achieve an accurate fatigue life prediction. The loading can be defined in various manners. Whether it be time based, frequency based, or in the form of some sort of spectra depends on the fatigue analysis type to be used. When working with finite element models the loading can be force, pressure, temperature, displacement, or a number of other types. Loading in the test world usually refers to the acquisition of a response measurement, usually from a strain gauge.
- 3. Geometry Information:** Geometry has different meanings depending on whether you are working from a finite element model or from a test specimen. In the testing world, the geometry input is the K_t (stress concentration factor) since the point of failure is usually away from the actual point of measurement. Therefore a geometry **Stress Concentration Factor, K_t** , is defined to relate the measured response to that at the failure location. You can think of this as a fudge factor. The corresponding role of finite elements is to produce fields of K_t over the entire model, with the additional complication that these local stress concentration factors are in the form of stress tensors.
- 4. Analysis:** The correctness and accuracy of each of the above inputs is important in that any error in any of these will be magnified through the fatigue analysis procedure, the fourth box, since this process is logarithmic. A ten percent error in loading magnitude could result in a 100% error in the predicted fatigue life. In a conventional finite element based fatigue analysis, the 4th box often contains both the stress prediction and fatigue life calculation.
- 5. Results:** The fifth box is the postprocessing or results evaluation. This can take on the form of color contours on a finite element model or a tabular listing but also quite often leads back into the three inputs to see what effect variations of these inputs will have on the life prediction. This is referred to as a sensitivity or a “what if” study. This is extremely useful at times when you are not quite sure about the accuracy of one of the inputs. This then leads to design optimization using fatigue life as a design constraint or possibly even as a design objective.



Life Prediction Methods

There are three main life prediction methods as already mentioned earlier. These are [Total Life](#), [Crack Initiation](#), and [Crack Growth \(Propagation\)](#). Total life is aptly named in that only the total life of the component is of concern and not when a crack will initiate or how quickly it will grow.



The three methods are related to each other by the fact that the total number of cycles to failure, N_f , equals the number of cycles to initiate a crack, N_i , plus the number of cycles to propagate that crack, N_p . The three methods have grown out of different needs over the decades using different techniques and having different degrees of accuracy. So in theory this equation is true, but in practice when applying the three methods to the same problem, rarely, if ever does it add up.

In reality however, rarely are all three methods used on the same problem, mainly because different industries adopt different analysis methods depending on the driving design philosophy. See [Design Philosophies, 8](#).

Finite Element Analysis Methods

In order to do life predictions, MSC Nastran embedded fatigue analysis supports use of stress-life and strain-life methods using the stress/strain response results from different finite element (FE) analysis techniques. The table below summarizes which FE analysis types are applicable to which life prediction methods in this release of the software.



Table 1-4 Life Prediction Methods vs. FE Analysis Results

	Total Life (S-N)	Crack Initiation (ϵ -N)
Linear Static SOL 101	Supported	Supported
Normal Modes SOL 103	Supported	Supported
Modal Transient Response SOL 112	Supported	Supported
Frequency Response SOL 108 & SOL 111	Supported	Supported
Design Optimization*	One of either: ANALYSIS=STATIC ANALYSIS=MFREQ ANALYSIS=DFREQ	One of either: ANALYSIS=STATIC ANALYSIS=MFREQ ANALYSIS=DFREQ

*For frequency domain solutions, please see the [MSC Nastran Embedded Vibration Fatigue User's Guide](#).

When to Use Which Method

Of the three fatigue methods used to predict life, it is important to understand when to use which. This will become more evident as you proceed through this manual and work each exercise. As a quick answer to this question, the following guidelines are presented.

Stress-Life (S-N or Total Life)

- Long life or **High Cycle Fatigue (HCF)** problems, where there is little **Plasticity** since the S-N method is based on nominal stress
- Components where crack initiation or crack growth modeling is not appropriate, e.g., composites, welds, plastics, and other non-ferrous materials
- Situations where large amounts of pre-existing S-N data exist
- Components which are required by a control body to be designed for fatigue using standard data such as MIL handbook data.
- Spot weld analysis and random vibration induced fatigue problems

Strain-Life (Crack Initiation or Local Strain or ϵ -N)

- Mostly defect free, metallic structures or components
- Components where crack initiation is the important **Failure Criterion** - safety critical components
- Locating the point(s) where cracks may initiate, and hence the growth of a crack should be considered



- Evaluating the effect of alternative materials and different surface conditions
- Components which are made from metallic, isotropic ductile materials which have symmetric cyclic stress-strain behavior
- Components that experience short lives - [Low Cycle Fatigue \(LCF\)](#) - where plasticity is dominant

Crack Growth (Damage Tolerant Design)

- Pre-cracked structures or structures which must be presumed to be already cracked when manufactured such as welds
- Prediction of test programs to avoid testing components where cracks will not grow
- Planning inspection programs to ensure checks are carried out with the correct frequency
- To simply determine the amount of life left after crack initiation
- Components which are made from metallic, isotropic ductile materials which have symmetric cyclic stress-strain behavior

High Performance Computing

In 1971, when MSC Software first offered a version of MSC Nastran, the fastest supercomputer in the world was the CDC 7600 designed by Seymour Cray. It had a clock speed of 36.4 MHz and a 65 Kword primary memory with a peak flop rate of 36 MFLOPS. The CDE 7600 was slightly faster than the IBM System/360 mainframe computer. The chief architect of System/360 was Gene Amdahl. For MSC Nastran, the limiting factor in those days was the memory available to solve reasonably large problems. In the first few versions, customers were happy to solve a linear statics FE problem with 100 DOFs in 12 hours.

As of June 2016, the fastest supercomputer in the world is the Tianhe-2 supercomputer developed by China's National University of Defense Technology. It has available over 3 million processor cores to deliver a peak flop rate of 54 TFLOPS, which is over 1 million MFLOPs. While the number of processors cores has changed in today's supercomputers and the FLOP rates have greatly increased, software products like MSC Nastran still have similar demands on them that limit performance or model size or both.

Hardware Factors Affecting Simulation Performance

The performance of computer aided engineering (CAE) simulations is dependent on a computer's hardware as much as it is dependent on the underlying algorithms. I/O performance, memory bandwidth, memory capacity, and flops/core generally defines the overall performance of a single computer used for CAE simulations. The typical configuration for many users of CAE software is to either run on a single, separate workstation consisting of the above configuration or to run on an HPC Cluster used by many employees at the engineering company where these nodes are clustered together in a network.

Communication costs between these separate nodes is measured by network bandwidth and latency. The impact of the hardware, whether it is network latency or CPU speed, is felt differently for different applications due to the underlying algorithms.



A very enlightening image was published at *The Next Platform* that we have reproduced in [Figure 1-19](#). In the figure, we are considering structural analysis simulations, crash simulations, and fluid simulations. We can see that if we are modeling structures (with MSC Nastran SOL 101), or crash (with MSC Nastran SOL 700), or fluids (with ANSYS Fluent), then we get a very different dependency on performance with respect to network latency, network bandwidth, flops/core, cores, memory capacity, memory bandwidth, and I/O performance.

We can also observe that fluid simulations get the most benefit from more cores because fluid simulations are using explicit methods that are highly parallelizable. It is one of the reasons why we often observe massively parallel computing (1000s or 10,000s of processors) used with fluid simulations. Crash simulations also depend on explicit calculations that are highly parallelizable.

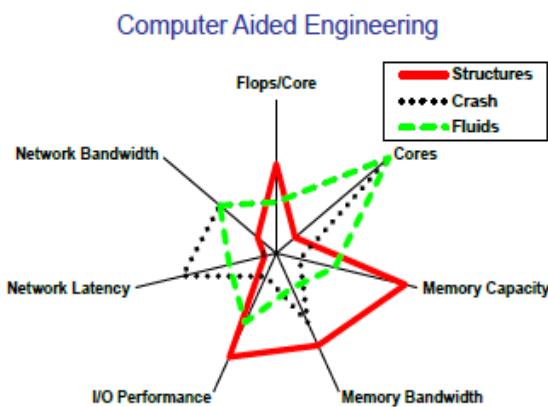


Figure 1-19 Relative Importance of Different Hardware Features for Various Simulations

The structural simulations described in this image generally correspond to MSC Nastran simulations, in particular, SOL 101, 103, and 400. The dominant aspect of these simulations is usually the matrix factorization of the stiffness matrix, which represents how the corresponding structural displacements correspond to loads. If the matrix and the factorization all fit within existing memory, then memory bandwidth is critical in determining performance.

Often the matrix and factorization process does not fit in existing memory and so I/O performance and memory capacity are critical. This can be the case with MSC Nastran although we will see in the remainder of this HPC Manual that there are times where flops/core or cores are equally beneficial.

Memory

Memory capacity and bandwidth is perhaps the most underestimated way to improve performance in FE analysis. Increasing memory capacity allows more of the problem to be solved in-core where access speeds can be orders of magnitude faster than out-of-core where disk speeds matter. Memory can also serve as a cache for I/O by the operating system, or as you will see below, by MSC Nastran itself with the use of Buffer Pool.



Memory may be purchased as DDR3/DDR4. The following table shows a comparison:

Memory Type	Speed
DDR3	800-2133Mb/s
DDR4	1600-3200Mb/s

Additional RAM may significantly reduce elapsed times for large models. The following is the chart of recommended RAM for linear or nonlinear statics models. Dynamic models may also perform better with additional RAM but it is more problem dependent and harder to gauge based on DOF as it also depends on frequency range in the dynamic problem.

DOF	Memory
< 100,000	20 GB
< 1,000,000	64 GB
< 10,000,000	128 GB
< 20,000,000	256 GB
< 40,000,000	512 GB

Disks

When MSC Nastran exceeds the system memory cache buffer, the I/O will go to disk. Since finite element analysis can be intensive, the disk configuration may impact the performance of the analysis. Improving disk performance can be important to MSC Nastran performance.

- Do not use a network drive.
- SSD disks are very fast and highly recommended for scratch space.
- Faster disks, e.g., 15000 RPM, may help performance.
- Run with SCR=YES if you do not need to save a database.

RAID 0 with multiple disks for SDIR is recommended. Other levels of RAID (for example, RAID 5) have redundancy which is not needed for scratch files and may have some performance degradation.

We describe in general terms how memory performance, CPU performance, I/O details, and parallel capabilities affect simulation performance of the following:

- Linear static analysis (SOL101). For details, see “[Ch. 3: Optimal Performance of Linear Static Analysis \(SOL 101\)](#)” of [MSC Nastran High Performance Computing User’s Guide](#).
- Normal mode analysis (SOL 103). For details, see “[Ch. 3: Optimal Performance of Linear Static Analysis \(SOL 101\)](#)” of [MSC Nastran High Performance Computing User’s Guide](#).
- Direct frequency response analysis (SOL 108). For details, see [Chapter 5: Optimal Performance of Direct Frequency Response Analysis \(SOL 108\)](#)” of [MSC Nastran High Performance Computing User’s Guide](#).



- Modal frequency response analysis (SOL 111). For details, see “[Chapter 6: Optimal Performance of Modal Frequency Response Analysis \(SOL 111\)](#)” of [MSC Nastran High Performance Computing User’s Guide](#).
- Nonlinear analysis (SOL 400). For details, see “[Chapter 7: Optimal Performance of Nonlinear Analysis \(SOL 400\)](#)” of [MSC Nastran High Performance Computing User’s Guide](#).

For information on memory performance, CPU performance, I/O details, and parallel capabilities please refer to *High Performance Computing User’s Guide*

Hardware Accelerators

Performance improvements derived from Hardware Accelerators vary a great deal. While in some cases, GPUs may greatly increase performance, in other cases, GPUs may degrade performance. Thus, caution should be used in predicting the amount of potential performance gain.

- GPUs help with large dense models.
- GPUs degrade performance for ACMS, thus they are disabled for ACMS.
- Static analysis for which GPUs help may benefit more from the CASI solver in cases where the CASI solver is allowed.
- In many cases using a high number of SMP processors may benefit as much as a GPU.
- Multiple GPUs are only used when MSC Nastran is running DMP. The command line option is as follows: `gpuid=0,1`

Where the numbers specified are the minor numbers from the `nvidia-smi -q` command.

- GPU performance will improve the following system cell settings. These settings will happen automatically in 2016 for GPU jobs:

```
system(655)=32 system(656)=1024 system(205)=320 system(219)=384
system(220)=320 system(221)=320
```

Network Connections

Network connections effects the analysis in the following scenarios:

- If a directory containing files is remotely mounted
- If an analysis is running DMP over a network.

The largest performance gains from MSC Nastran by using multiple nodes is that each process will have an increase of RAM available by dispersing the runs amongst different computers. If an analysis can take advantage of the additional RAM on multiple nodes, then faster networks may help for problem types that have a substantial amount of communication.

For details, refer to [MSC Nastran High Performance Computing User’s Guide](#).



3 MSC Nastran Files

- Types of MSC Nastran Files 74
- MSC Nastran Input Files 74
- Case Control Section 80
- Bulk Data Section 82
- Continuation Entries 85
- Field Formats in Input Files 86
- Sample MSC Nastran Model 91
- MSC Nastran Output Files 90



Types of MSC Nastran Files

In order to run MSC Nastran, it is mandatory to create an input file to define the entire finite element model. Once the file is prepared, you can execute MSC Nastran by invoking a special command followed by the name of the input file. Upon completion of the finite element calculations, MSC Nastran creates several files each containing different types of information. We refer to the files that contain the results as the output files. Different types of file extensions which are used in MSC Nastran are listed in the [Table 1-1](#).

Table 1-1 File Types

File Extension	File Type	Comment
.bdf	Input	Contains model definition.
.dat	Input	Data files containing model definition.
.h5	Input	It is a data model, library, and file format for storing and managing data. It supports an unlimited variety of data types, and is designed for flexible and efficient I/O and for high volume and complex data.
.MASTER/DBALL	Database	Useful for restart and can be used by Patran for post-processing
.f04	Execution Summary	Contains a time history of job execution.
.log	Output	Job log file.
.pch	Output	Punch file.
.plt	Output	Binary plot file.
.f06	Results	This is the main MSC Nastran output file. It contains the results of your analysis (such as displacements and stresses). It is in ASCII format so it can be viewed in any text editor. It also contains warning messages, error messages, and diagnostic messages to help you evaluate the quality of the analysis results.
.op2	Results	Used by Patran for post-processing.
.xdb	Results	Used by Patran for post-processing.

MSC Nastran Input Files

To do an analysis using MSC Nastran, you must generate an input file describing the structure's geometry, material properties, boundary conditions, and loads. MSC Nastran input file contains a complete description of the finite element model, including:

- Type of analysis to be performed
- The geometry of the model
- Node locations



- Element connections
- Element properties and material data
- Structure under consideration
- Loads acting on it
- Constraints (boundary conditions)
- Plots to be created
- Requests for the type of output quantities to be calculated

In addition to defining the physical structure, the input file also specifies the type of analysis to be performed and other pertinent information. The input file is an ASCII text file which can be created using any text editor or one of the many preprocessors that interface with MSC Nastran.

After the generation of the input file is complete, it is submitted for execution as a batch process. The MSC Nastran input file is useful in the following ways:

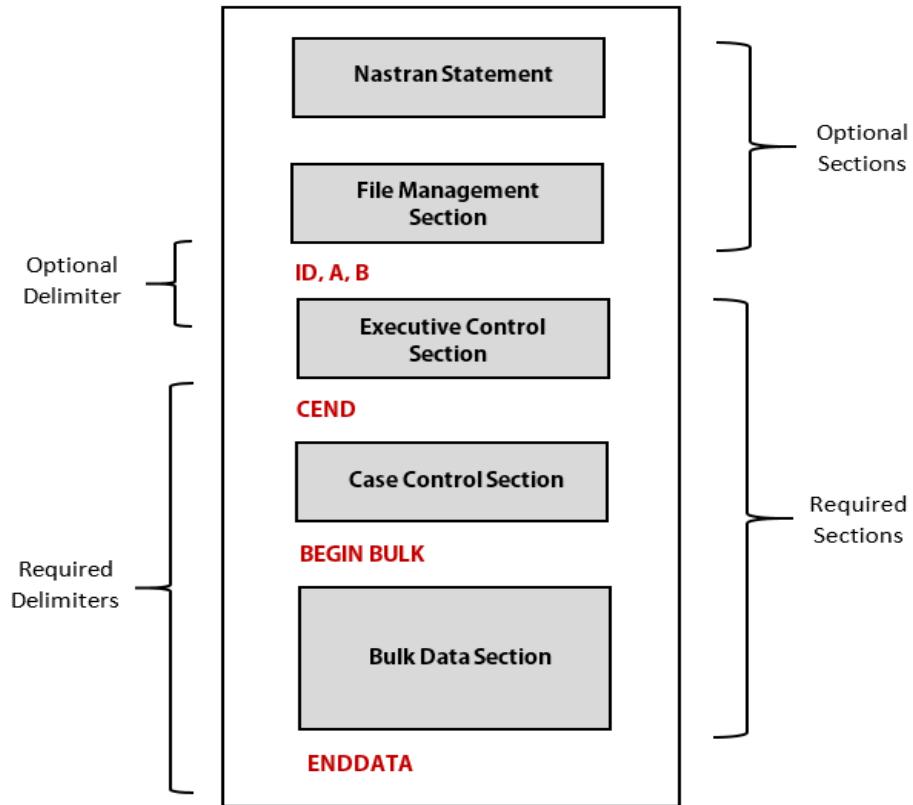
- Can be viewed and edited in any text editor
- Can include comments to document modeling assumptions and changes
- Allows the user to add entries which are not supported in Patran
- Useful in debugging a model.

The MSC Nastran input file is often referred to as the bulk data file (.bdf) or the data file (.dat) in the MSC Nastran manuals. It consists of six distinct sections—three of them are required and the other three are one-line delimiters.

Input data is organized in (optional) blocks. Key words identify the data for each optional block. This form of input enables you to specify only the data for the optional blocks that you need to define your problem. The various blocks of input are optional in the sense that many have built-in default values which can be used in the absence of any explicit input from you.

The input requires records that are 80 characters (or columns) in length. The structure of the input file is shown in the [Figure 1-1](#).





Executive Control Section Recommended _____	{ ID ROD EXAMPLE SOL 101 TIME 5 CEND
Case Control Section Recommended _____	{ LOAD=8 DISP=ALL SPCF=ALL ECHO=BOTH BEGIN BULK
Bulk Data Section Recommended _____	{ GRID,1,,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 ENDDATA

Figure 1-1 Structure of MSC Nastran Input File



The required sections of a typical MSC Nastran input file are:

- Executive Control: It describes the problem or solution type and optional file management. It is terminated by an CEND parameter.
- Case Control: It defines the load history and output requests. It is terminated by the BEGIN BULK option.
- Bulk Data: It defines a detailed model, load and constraint description. Contains a model data starting with the BEGIN BULK option is terminated by the ENDDATA. It may contain multiple Bulk Data sections when defining model partitions called Part Superelements.

Delimiters

The delimiter entries are required to separate sections of the input file:

Delimiters	Description
ID A,B	First statement in executive control section (optional).
CEND	End of executive control section, beginning of case control section.
BEGIN BULK	End of case control section, beginning of bulk data section.
ENDDATA	Last entry in all the input files.

Table 1-2 Delimiter Entries of the Input File

Nastran Statement

The MSC Nastran statement is optional and is used to modify system parameters (also called system cells) that control internal solution processing or provide specific diagnostics. Examples include aspects of working memory, datablock size, datablock parameters, machine specific issues, numerical methods, etc. It determines the overall program control for the current run.

The MSC Nastran statement if used, must be placed before Executive Control Section. Since it is used for special circumstances, it is not needed in most runs. It may also be specified in the Runtime Configuration (RC) files at the system, user, and job level.

For further information refer to [Executing MSC Nastran](#) in the *MSC Nastran Quick Reference Guide*.

File Management Section (FMS)

The File Management Section (FMS) is also optional. If required, it is also located before Executive Control Section. The goal of FMS is to make the operating system invisible to the user. It allocates files, manipulates the database, and performs restarts. It also determines overall program control for the current run. It is commonly used when solving large problems. It is used primarily to attach or initialize



MSC Nastran databases and Fortran files. The initialization of a Fortran file includes the specification of its filename, Fortran unit numbers, and Fortran attributes. MSC Nastran Statement and FMS use free field format (in columns 1-72).

For many MSC Nastran problems, FMS are not required, because a default FMS is executed at the beginning of every run. For detailed discussion of the FMS, refer to [File Management Statements](#) in the *MSC Nastran Quick Reference Guide*.

Executive Control Section

The Executive Control Section is the first required group of statements in any MSC Nastran input file. It provides overall control of the solution. It provides overall job control for the problem and sets up initial switches to control the flow of the program through the desired analysis. Entries in the Executive Control Section are called statements.

The Executive Control Section contains statements that does the following:

- Select the solution sequence to be run (e.g., statics, normal modes).
- Request various diagnostic outputs.
- Contain user-written DMAP.
- Choose a sparsesolver or domainsolver option.

Executive control section and case control section uses free field format in columns 1-72. Input may begin in any of these columns and is separated by commas or blanks. The set of the executive control section must be terminated by a CEND delimiter.

The basic functions of this section are:

- Identify the job (optional TIME statement)
- Select the type of analysis to be performed, that are required for all MSC Nastran jobs
- Set limits on the allowable CPU time (maximum time limits for execution)

Most statements are order independent. Exceptions are the COMPILE, COMPILER, ALTER, ENDALTER, and LINK statements. The LINK statement must appear after all COMPILE statements. The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. This statement sets the default print control for subsequent COMPILE statements.

For a complete description of the Executive Control Section, refer to [Executive Control Statements](#) of the *MSC Nastran Quick Reference Guide*.



Summary of Executive Control Statements

The Executive Control Statements consists of the following statements:

Executive Control Statements	
ALTER	Specifies deletion and/or insertion of the DMAP statements that follow.
APP	Specifies an approach in a solution sequence.
CEND	Designates the end of the Executive Control statements.
COMPILE	Requests compilation of specified subDMAPs or the NDDL file.
COMPILER	Specifies DMAP compilation diagnostics.
DIAG	Requests diagnostic output or modifies operational parameters.
DOMAINSOLVER	Selects domain decomposition solution methods.
ECHO	Controls the echo of Executive Control statements.
ENDALTER	Designates the end of a DMAP sequence headed by an ALTER.
GEOMCHECK	Specifies tolerance values and options for optional finite element geometry tests.
ID	Specifies a comment.
LINK	Requests the link of a main subDMAP.
MALTER	Inserts and/or deletes DMAP statements in solution sequences
MODEL_CHECK	Specifies model checkout run options.
SOL	Requests execution of a solution sequence or DMAP program.
SOL 600, ID	Creates Marc input and optionally executes Marc from SOL 600
SOL 700, ID	Executes MSC Nastran Explicit Nonlinear (SOL 700)

The following sections briefly describe three main statements i.e., ID, SOL, and CEND.

ID Statement

This optional statement is used to identify the job. If used, the ID statement must be first given in the Executive Control Section. The format is:

ID i1, i2

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



SOL Statement

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

SOL n

Where n is a positive integer which identifies the solution type or the character name of the solution procedure.

This manual describes SOL 101 linear static analysis. Many other solution types are available in MSC Nastran, explained in [Case Control Section](#).

CEND Statement

The CEND statement is a recommended statement that designates the end of the executive control section (and the beginning of the case control section). The format of the statement is:

CEND

A variety of other executive control statements are available, most notably the DIAG statement which offers you a variety of useful diagnostic information. Discussion of these statements is beyond the scope of this manual, but more information can be found in the [Executive Control Statements](#) of *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

CEND

CEND = Required entry = Last Entry in Executive Control

DIAG J = Optional statement - enables diagnostics

Case Control Section

The Case Control Section defines the subcase structure for the problem. It makes selections from the Bulk Data Section, and creates output requests. This section provides control of the individual loadings to be applied and the output requests. Entries in the case control section are called commands. The case control section always follows the Executive Control Section and precedes the Bulk Data Section and is required in every run.

The primary functions of the case control section are as follows:

- Specifies sets of bulk data input that are to be used in the analysis (e.g., loads, constraints, eigenvalue solution method, etc.)



- Specifies and control the type of analysis output required (e.g., forces, stresses, and displacements)
- Manages sets of Bulk Data input, define analysis subcases (multiple loadings in a single job execution), and select loads and boundary conditions
- Makes output selections
- Specifies printout and post-processing options
- Defines subcases/structures for the analysis (load cases).

For a summary of all output that can be requested for each solution sequence refer to [Case Control Commands](#) in *MSC Nastran Quick Reference Guide*. For Case Control Section Requirements, refer to [The Case Control Section](#).

Bulk Data items selected in the above manner include loads, constraints, and thermal fields.

Note: BULK DATA entries must be selected by CASE CONTROL commands. If not, they will be ignored in the current run.

Summary of Case Control Commands

The Case Control commands described in this guide are summarized as follows:

Table 1-3 Case Control Commands

Case Control Commands	
\$	Comment Entry
DISPLACEMENT (Case)	Displacement Output Request
ECHO (Case)	Bulk Data Echo Request
FORCE (Case)	Element Force-Output Request
LABEL (Case)	Output Label
LOAD (Case)	External Static Load Set Selection
OLOAD (Case)	Applied Load Output Request
SET (Case)	Set Definition
SPC (Case)	Single Point Constraint Set Selection
SPCFORCES (Case)	Single Point Forces of Constraint Output Request
STRESS (Case)	Element Stress Output Request
SUBCASE (Case)	Subcase Delimiter
SUBTITLE (Case)	Output Subtitle
TITLE (Case)	Output Title



Bulk Data Section

Entries in this section are used to define the model and data which may be selected in the Case Control Section at execution time.

Bulk Data entries consists of the following:

- Finite element model/geometry
- Coordinate systems
- Finite elements
- Element properties
- Loads
- Boundary conditions
- Material properties

The Bulk Data can be submitted with the entries in any order as a sort is performed prior to the execution of the input file processor.

You can get a printed copy of either the unsorted or sorted Bulk Data by selecting in the Case Control Section. Bulk Data entries can be entered in any order and they end with the recommended ENDDATA delimiter.

Note: A sorted echo is automatically provided unless specifically suppressed by you.

Format of Bulk Data Entries

The format of the Bulk Data section is based on each line as follows:

- It can have 80 possible columns
- It can be divided into 10 fields

For example the CROD element topology is defined as follows:

Format:

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

Example

CROD	12	13	21	23					
------	----	----	----	----	--	--	--	--	--

Field	Contents	Value
EID	Unique element identification number.	Integer > 0



Field	Contents	Value
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 ≠ G2

Bulk data entries consists of the following three data formats:

- Integer: Integers must be entered without a decimal point (e.g.: 1, 134, 267, etc.).
- Real: Real data has a decimal point and may have an exponent. There are several ways to represent real numbers. Following representations of the real number 123.4 are numerically equivalent and acceptable to MSC Nastran. All these represent the same number as follows:

123.4	123.4+2	123.4E2	123.4E+1
0.1234E	.1234E3		

- Character String (BCD): BCD is represented as a text string
 - It must start with a letter (A-Z)
 - It may contain numbers in the text (0-9)
 - It must be 8 or fewer characters long
 - No special characters or embedded blanks may be used
 - Examples: TEST123, X32, DUM1

Note: Integer and real data formats are used for majority of the data entries.

Summary of Bulk Data Entries

Some of the common Bulk Data entries are summarized below. See the *MSC Nastran Quick Reference Guide* for a complete description of all Bulk Data entries.

Table 1-4 Bulk Data Entries

Bulk Data Entries	
\$	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
CONM1	Concentrated Mass Element Connection, Rigid Body Form
CONROD	Rod Element Property and Connection



Table 1-4 Bulk Data Entries

Bulk Data Entries	
CORD1C	Cylindrical Coordinate System Definition, Form 1
CORD1R	Rectangular Coordinate System Definition, Form 1
CORD1S	Spherical Coordinate System Definition, Form 1
CORD2C	Cylindrical Coordinate System Definition, Form 2
CORD2R	Rectangular Coordinate System Definition, Form 2
CORD2S	Spherical Coordinate System Definition, Form 2
CPENTA	Five-Sided Solid Element Connection
CQUAD4	Quadrilateral Plate Element Connection
CROD	Rod Element Connection
CTETRA	Four-Sided Solid Element Connection
DEFORM	Static Element Deformation
FORCE	Static Force
GRAV	Acceleration or Gravity Load
GRID	Grid Point
LOAD	Static Load Combination (Superposition)
MATBV	Isotropic Material Property Definition
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



Continuation Entries

Continuation entries must be used in case of entries that require more than one line of input. These entries may be generated automatically when they are in sorted order. Continuation entries are not required when the entries are in sorted order.

Note: Field 10 and field 1 must contain unique continuation identifiers that match.

The format of the continuation entries is as follows:

1	2	3	4	5	6	7	8	9	10
MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						

The parent entry can be blank in columns 74-80 (field 10), and the continuation entry can be blank in columns 2-8 (field 1). For small field entries, the first column of the continuation entry may be blank or contain a + symbol. For large field entries, the first column of the continuation entry must contain a * symbol.

The input rules are listed as follows:

- When you use continuation mnemonics, a plus sign (+) or astrix (*) is required in column 1, field 1 of a continuation entry. The remaining contents in field 1 of a continuation entry must be identical to the entry in field 10 (columns 74 through 80) of the parent entry (or the preceding continuation entry).
- Any entry in the first column of field 10 (column 73) on the parent entry is ignored by the continuation entry.
- Small field and large field continuation entries may be used together in defining a single data item entry.

Methods for Entering a MAT8

There are two methods for entering a MAT8 entry with continuation.

Method 1:

1	2	3	4	5	6	7	8	9	10
MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						

Method 2



Method 2 is auto generated. Its format is as following:

1	2	3	4	5	6	7	8	9	10
MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						

Field Formats in Input Files

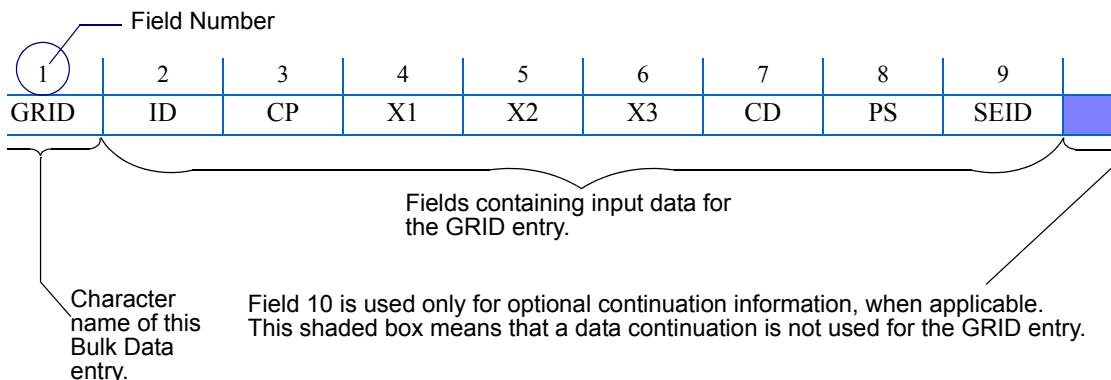
MSC Nastran has three different field formats for input data:

Different Field Formats	
Free Field Format	Input data fields are separated by commas.
Small Field Format	Ten fields of eight characters each.
Large Field Format	Ten fields each 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:



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:

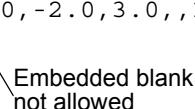
GRID, 2,,1.0,-2.0,3.0,,136

 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
- 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 as follows:

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:

1				8 character field	5	6	7	8	9
80 characters									

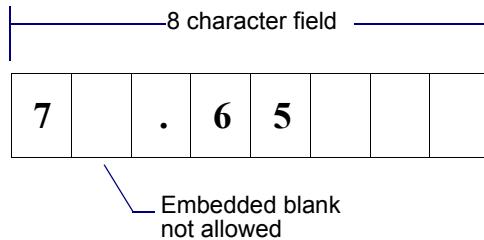


The following is an example of the GRID entry in small field format:

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 ←	→ 16 ←	→ 16 ←	→ 16 ←	→ 16 ←	→ 8 ←
Columns					



Second Line:

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

Input Format Rules

Rules for the input format are as follows:

- Errors result if data extends beyond its field into another field
- Input data items in fields 1 and 10 must be left justified
- Input data in fields 2 through 9 do not have to be left or right justified
- Input items must not have embedded blanks

G	R	I	D	,	1	0	,	,	7	.	5	,	8	,	.	6	,	9	.	0
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

Note: Since a blank is a valid separator, MSC Nastran interprets this as an integer 8 rather than the intended value, 8.6. Since real numbers are required for coordinate data, a fatal error result.

- All real numbers (including zero) must have a decimal point (this is a common error)
- Many fields have a default value, if these fields are left blank, the default value will be used (Refer to *MSC Nastran Quick Reference Guide*).

Common Errors in Entry Format

Caution: Failure to leave the proper number of fields blank when defining data values leads to either a fatal error or wrong answers.

Make sure you follow the below recommendations to avoid commonly made errors in the input format:

- Leave the proper number of blanks or include the correct number of commas to delimit data fields.
- Use the correct format for integer numbers and real numbers. See individual entry format in the *MSC Nastran Quick Reference Guide* for these specifications.
- Define all the required fields for the Bulk Data entries.
- Make sure the continuation entries follow the parent entry, sometimes the content of the continuation field may be ignored by setting system cell 357 to 1.



MSC Nastran Output Files

After an input file is submitted for execution, several output files may be generated. [Table 1-5 on page 90](#) provides description of all the potential files generated by MSC Nastran. Assuming the input file is named `run1.dat`, see `Install_dir/mscxxxx/doc/linstat/run1.dat`.

The output files that are generated depend on your input file requests and the command line options that you specify when the job is submitted. Some of the files shown will not be generated, and some will be generated and deleted automatically once the job is complete.

Table 1-5 MSC Nastran Output File and Contents

File	Contents
<code>run1.f06</code>	The main output file containing the printed output such as displacements, element forces, stresses, etc.
<code>run1.f04</code>	A history of the assigned files, disk space usage, and modules used during the analysis. Useful for debugging.
<code>run1.log</code>	A summary of the command line options used and the execution links.
<code>run1.DBALL</code>	A database containing the input file, assembled matrices, and solutions. Used also for restarting the run for additional analysis or output.
<code>run1.MASTER</code>	The file containing the master directory of the files used by the run and the physical location of the files on the system. This file is also needed for a restart run.
<code>run1.IFPDAT</code>	The file containing the bulk data entries for IFP restart processing
<code>run1.plt</code>	Contains the plot information requested by the NASTRAN plotter commands specified in input file.
<code>run1.pch</code>	Contains the punch output as requested in the input file.
<code>run1.op2</code>	Output database used for postprocessing of the results.
<code>run1.h5</code>	Results database in HDF5 format.
<code>run1.sts</code>	Contains convergence history for nonlinear problems.



4

Modeling Issues

- Units 92
- Format of the Input Data 92
- Meshes and Mesh Transitions 97
- Designing the Model 98
- Using Test Models 100



Units

Always use a consistent set of units to develop a finite element model. MSC Nastran cannot interpret physical units.

An example of inconsistency in usage of units is to specify model geometry in feet 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 occur and no user warning messages are generated.

List of consistent systems of units are listed in the [Table 1-1](#):

Table 1-1 Consistent Systems of Units

Quantity	English	SI
Input:		
Grid Point Geometry	inch	meter
Elastic Modulus	$\text{lb}_f/\text{inch}^2$	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	$\text{lb}_f/\text{inch}^2$	Newton/meter ²

Format of the Input Data

Real, Integer, and Character Input Data

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 issues an error message if the wrong class of input is used.

Real numbers may be entered in various other 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—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 as follows:

- 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.

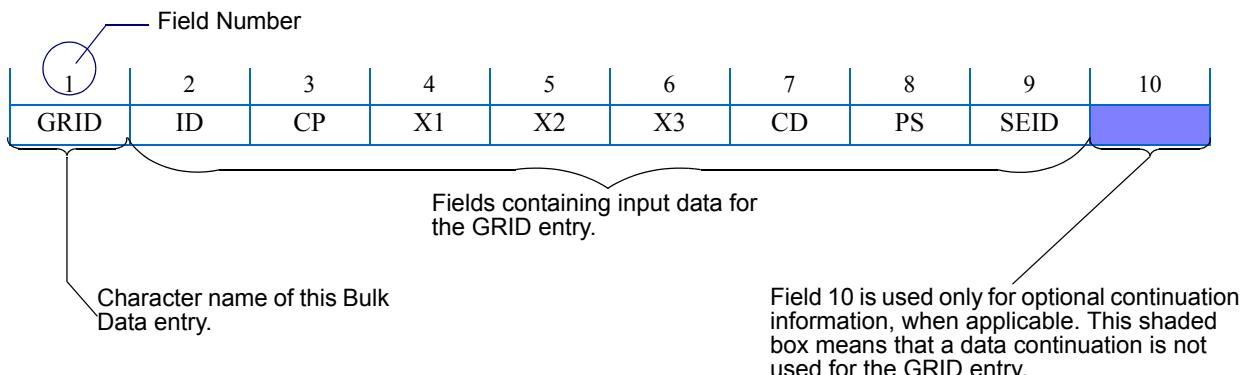
Consider the format of a typical MSC Nastran Bulk Data entry and the GRID entry which is used in MSC Nastran to describe the geometry of the structural model:

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 GRID Bulk Data entry example in free field format is shown below:

```
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:

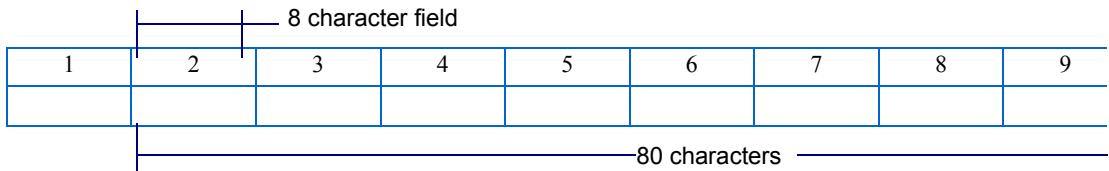
- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession.
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and 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 as follows:

```
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:



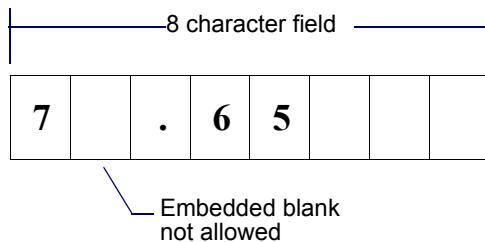


The following is an example of the GRID entry in small field format:

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

The rule for small field format is:

- 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 example of 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	► 16	► 16	► 16	► 16	► 8
Columns					



Second Line:

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

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 (what each field represents: 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	E1	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 then generates unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:**Listing 13-1**

```
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 13-2**

```
GEAR TOOTH EXAMPLE
                               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
```

For the above 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.

Meshes and Mesh Transitions

The finite element mesh is used to subdivide the CAD model into smaller domains called elements, and those Elements are connected to some grid-points called nodes. Adding more or refining elements produces a finer mesh, which can more closely represent an irregularly shaped structure.

The results are calculated by solving the relevant governing equations numerically at each of the nodes of the mesh. The pattern and relative positioning of the nodes also affect the solution, the computational efficiency and time. In general, a finer mesh is more accurate, but is also more computationally expensive.



Mesh transition regions are required in most finite element models. The purpose of mesh transitions are:

- 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 1-1](#).

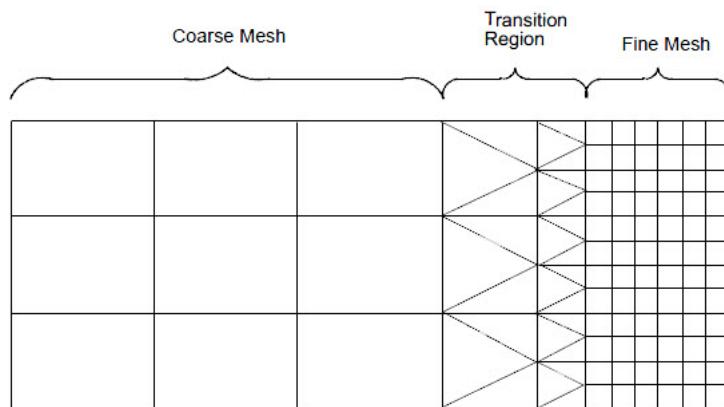


Figure 1-1 Typical Plate Model Mesh Transition

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

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. 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 large degrees (100 or more) 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.



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 the Response of your Structure

A second fundamental reason to use test models occurs when you 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.



5

Model Geometry

- Model Geometry 102
- Grid Points 103
- Coordinate Systems 105



Model Geometry

Model geometry is defined in MSC Nastran with the help of 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 over a million grids. The structure's grid points displace along with the loaded structure. Each grid point of the structural model has six possible components of displacement:

Three translations about the direction are as follows:

- x-
- y-
- z-

Three rotations about the axes are as follows:

- x-
- y-
- z.

The above components of displacement are called degrees of freedom (DOFs).

Example

In [Figure 1-1](#), on 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).

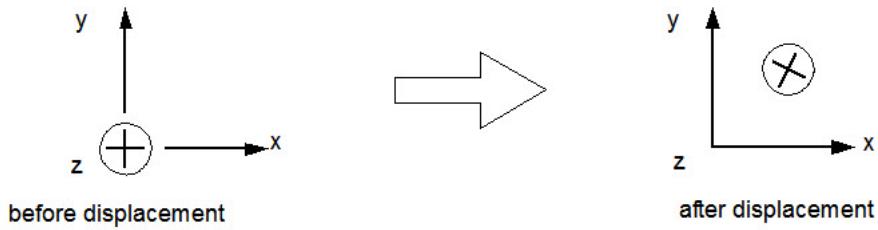
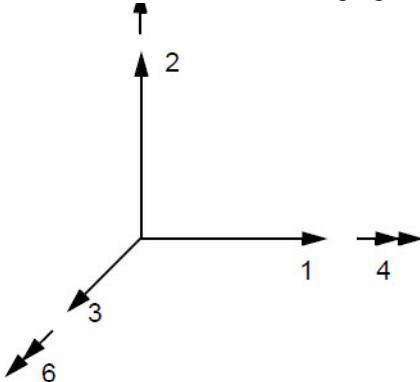


Figure 1-1 Grid Point Displacement



Grid Points

Grid points are used to define the geometry of a structure. As discussed in [Model Geometry](#), 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 the following figure.



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

- DOF 1 = T₁ = U₁ = translation in direction 1
- DOF 2 = T₂ = U₂ = translation in direction 2
- DOF 3 = T₃ = U₃ = translation in direction 3
- DOF 4 = R₁ = θ₁ = rotation about direction 1
- DOF 5 = R₂ = θ₂ = rotation about direction 2
- DOF 6 = R₃ = θ₃ = rotation about direction 3

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

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



Field	Contents	Value
ID	Grid point identification number.	$0 < \text{Integer} < 100000000$
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
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)
- 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](#) for more information. Field 9 applies only to superelement analysis (superelement analysis is described in [Superelement Analysis](#)).

Coordinate Systems

Basic Coordinate System

MSC Nastran has a built-in rectangular Cartesian system called the basic coordinate system, also called the default 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.

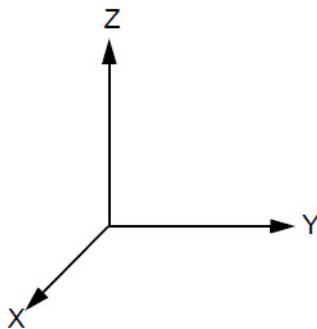


Figure 1-2 The Basic Coordinate System

Local Coordinate System

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 as follows:

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.



- The **CORD2R**, **CORD2C**, and **CORD2S** entries define a local coordinate system by specifying the location of three points.

Note: If the model is modified and any of these reference grid point locations change, the coordinate system orientation will also change.

The format of **CORD2C** Bulk Data entry 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						

Filed	Content	Value
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
A _i , B _i , C _i	Coordinates of three points in coordinate system defined in field 3.	Real

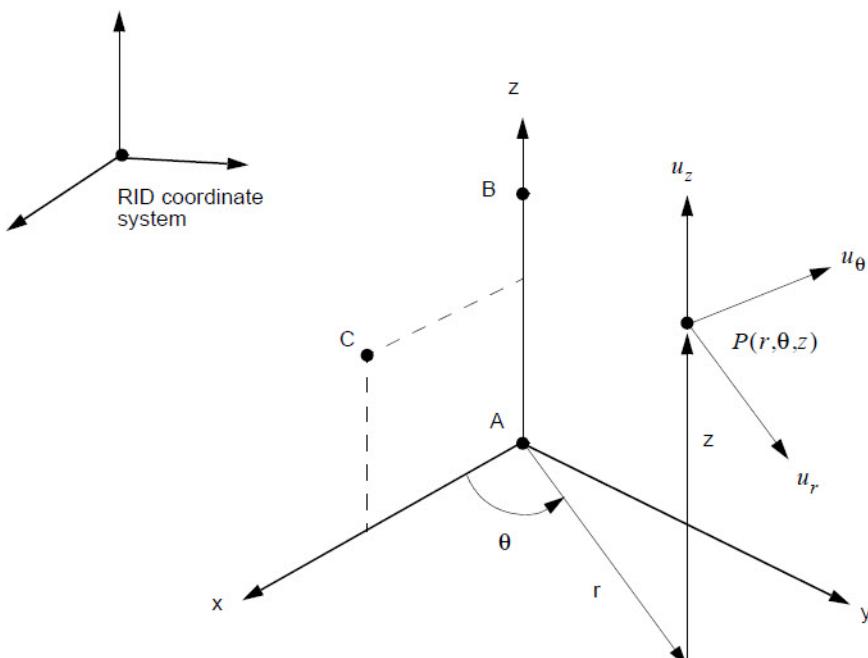


Figure 1-3 CORD2C Definition



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 non-colinear. Non-colinearity is checked by MSC Nastran's geometry processor.

- The first point (A) defines the origin
- Second point (B) defines the direction of the z-axis
- Third point (C) lies in the plane of the azimuthal origin.

Note: 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 1-3](#)) 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).

The following example illustrates the use of the [CORD2C](#) entry.

An arch with a semicircular top is shown in the [Figure 1-4](#). To facilitate grid point input, we wish to establish a local cylindrical coordinate system for grid points 3 through 7.

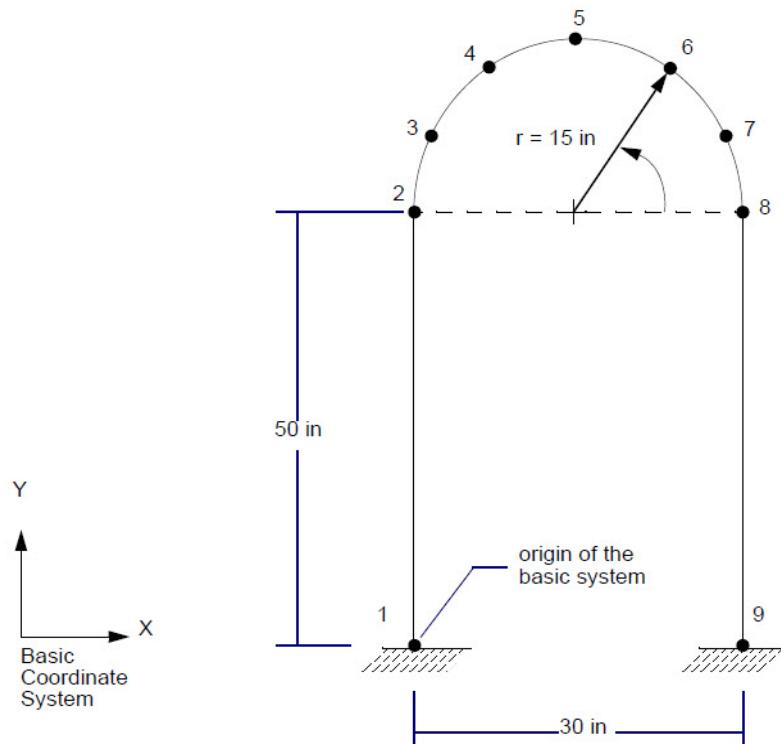


Figure 1-4 Arch Structure

A cylindrical coordinate system with an ID of 100 is defined as shown in the following figure.

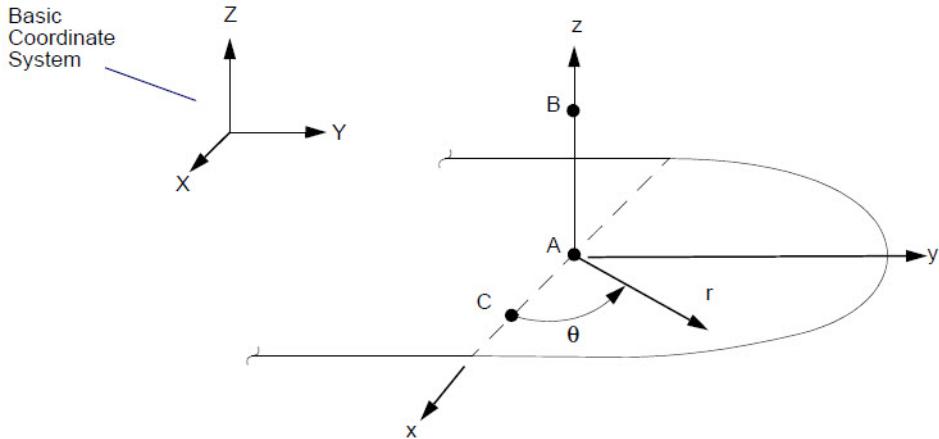


Figure 1-5 Definition of Local Cylindrical Coordinate System

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
- Point B lies on the z-axis of the new system
- Point C defines the reference axis at which $\theta = 0^\circ$

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.				
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).

Note: 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 [Bulk Data Entries](#) in [MSC Nastran Quick Reference Guide](#).



6 Basic Element Library

- Introduction 112
- Spring Element (CELAS2) 113
- Line Elements 115
- Surface Elements 127
- Solid Elements 139
- Rigid Bar Element (RBE2) 148

Introduction

MSC Nastran offers a variety of general purpose and specialty finite elements. The common and widely used elements are categorized in [Table 1-1](#):

Table 1-1 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, which are listed below:

- 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.

Additional details concerning the features and use of each of MSC Nastran's elements can be found in [Bulk Data Entries](#).



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.

Format

The format of the **CELAS2** entry is as follows:

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

Field	Contents	Value
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. (; blank or zero if scalar point)	$0 \leq \text{Integer} \leq 6$; blank or zero if scalar point.
GE	Damping coefficient.	Real.
S	Stress coefficient.	Real.

The description for the above format is as follows:

- 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.
- 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 [Figure 1-1](#). One end is fixed and the other is subjected to a 10 lb_f axial load. The axial stiffness of the spring (k) is $100 \text{ lb}_f/\text{inch}$. What is the displacement of GRID 1202?

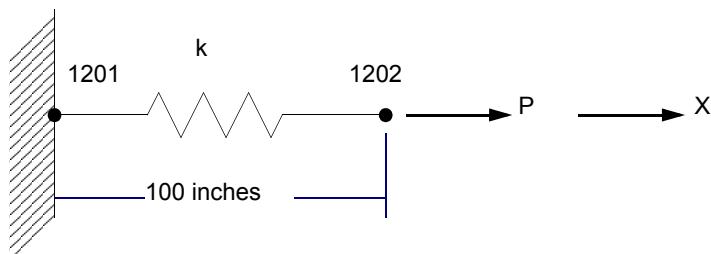


Figure 1-1 Extension of the Spring

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			
<hr/>									
GRID	1201		0.	0.	0.		123456		
GRID	1202		100.	0.	0.		23456		

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

Note: 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, where P is the applied load.

For the complete input file see `Install_dir/mscxxxx/gt06celas.dat`. The grid point displacement and element force output is shown in [Figure 1-2](#).

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

ELEMENT	FORCE	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)
ID.		ELEMENT	FORCE	ELEMENT	FORCE
		ID.		ID.	
1200	-1.000000E+01				

Figure 1-2 CELAS2 Spring Element Output

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

$$u_x = \frac{P}{k} = \frac{10.1b_f}{100.1b_f/\text{inch}} = 0.1 \text{ in.}$$

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

$$f = k(u_x^1 - u_x^2) = 100. (0. - 0.1) = -10.0 \text{ lb}_f$$

where:

u_x^1 is the displacement of G1

u_x^2 is the displacement of G2

Note: 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. Hence, be careful of how you interpret signs when using scalar elements.

Line Elements

Line elements, also called one-dimensional elements, are used to represent rod and beam behavior. A rod element does not support bending, whereas tension, compression, and axial torsion are supported. Only a beam element supports bending.

MSC Nastran makes an additional distinction between *simple* beams and *complex* beams as follows:



- 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 1-3](#) has two grid points, one at each end, and supports axial force and axial torsion. Thus, stiffness terms exist for only two degrees of freedom 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 1-3 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.

Format

The format of the CONROD entry is as follows:

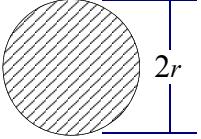
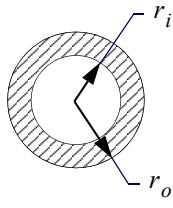
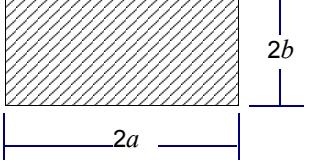
1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Field	Contents	Value
EID	Unique element identification number.	Integer > 0
G1, G2	Grid point identification numbers of connection points.	Integer > 0; G1 ≠ G2
MID	Material identification number.	Integer > 0
A	Area of the rod.	Real

Field	Contents	Value
J	Torsional constant.	Real
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 1-2](#) for a variety of cross sections.

Table 1-2 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:

$$\tau = C \frac{M_\theta}{J} \quad (6-1)$$



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 used when defining rod elements having the same properties.

Format

The format of **CROD** element is as follows:

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

Field	Contents	Value
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 ≠ G2

Rod Element Property (PROD)

PROD

The **PROD** entry supplies element properties to CROD elements.

Format

The format of **PROD** element is as follows:

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



Field	Contents	Value
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 [Sample MSC Nastran Model](#).

Simple Beam Element (CBAR)

CBAR Element Characteristics

CBAR



The CBAR element is a general purpose beam that supports the following:

- Tension and compression
- Torsion (bending in two perpendicular planes)
- Shear in two perpendicular planes
- The CBAR uses two grid points and can provide stiffness to all six degrees of freedom 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).

Note: Further description on offsets and pin flags is beyond the scope of this guide.

Refer to the following entries ([CBAR](#) and [CBEAM](#) in [MSC Nastran Quick Reference Guide](#)) or [The CBAR Element, 99](#) and [The CBEAM Element](#) (p. 120) in the *MSC Nastran Linear Static Analysis User's Guide* for further details.

CBAR Format

Two formats of the [CBAR](#) entry are available, the format 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	

Alternate Format:

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

Field	Contents	Value
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 ≠ 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; G0 ≠ GA or GB



Field	Contents	Value
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 description of the format used above is as follows

- 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

Note: 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 1-4](#).



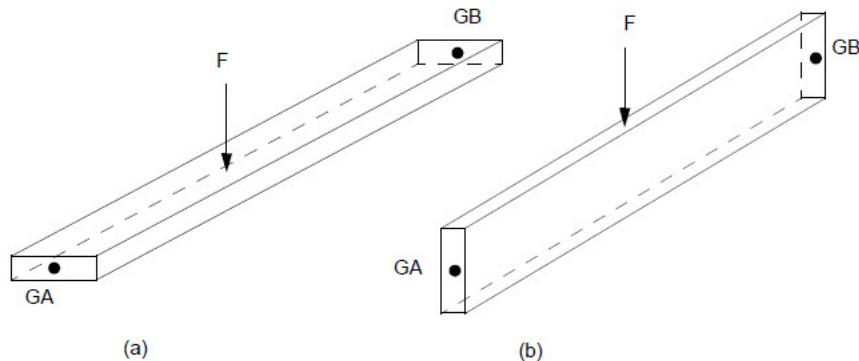
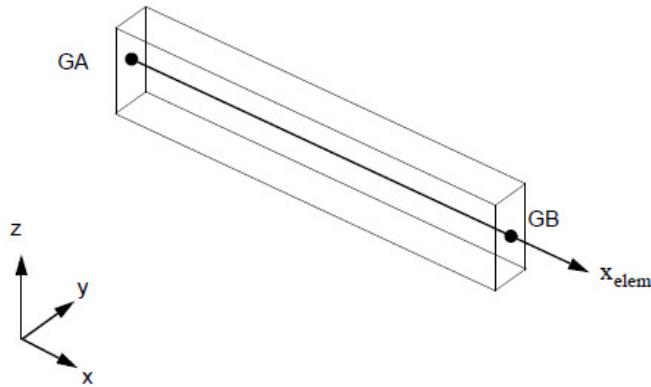


Figure 1-4 Two Orientations of the Same Beam

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 in [Figure 1-5](#).

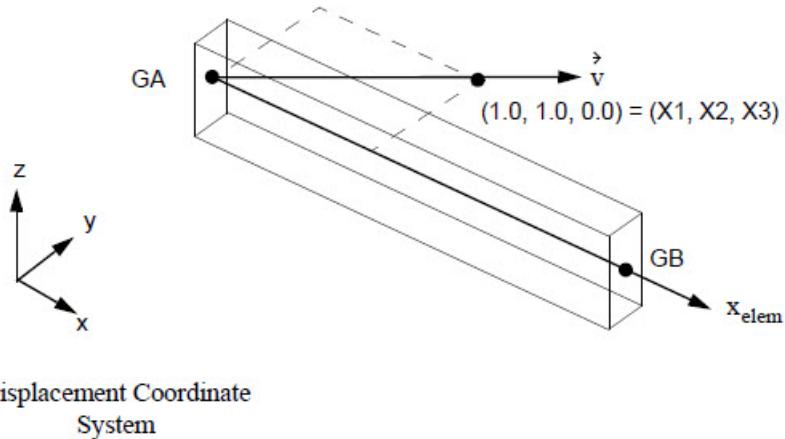
1. The element x-axis is automatically defined as the direction from GA to GB. The axis begins at GA.



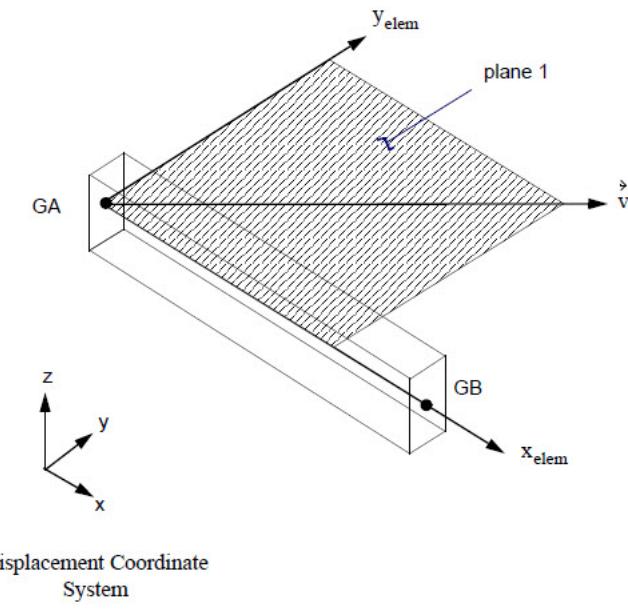
Displacement Coordinate System



2. Choose a direction for the beam's orientation vector \vec{v} . Vector \vec{v} starts at GA and contains the point (X_1, X_2, X_3) . X_1, X_2 , and X_3 , 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:



3. 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:



4. 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.

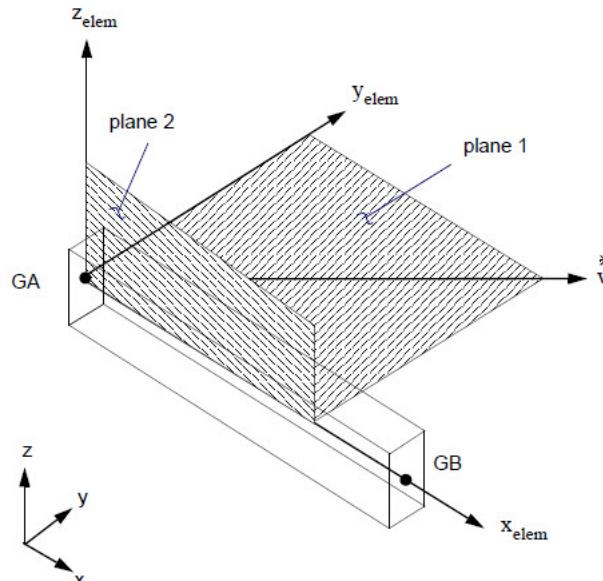


Figure 1-5 CBAR Element Coordinate System

CBAR Force and Moment Conventions

The CBAR element force and moment conventions are shown in Figures 1-6 and 1-7.

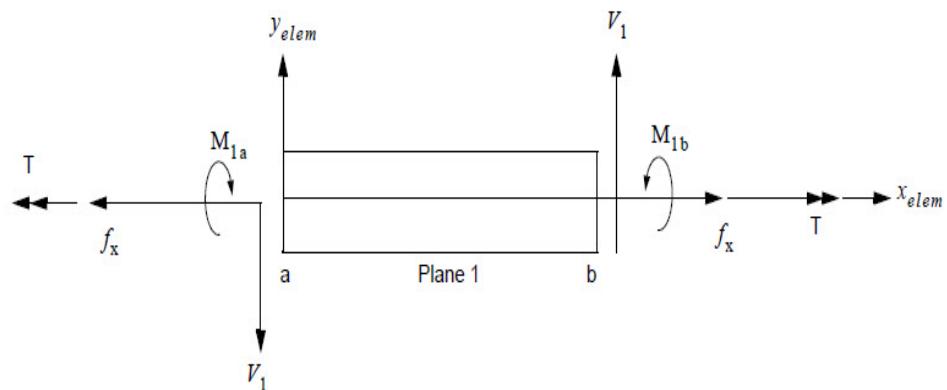


Figure 1-6 CBAR Element Internal Forces and Moments (x-y Plane)

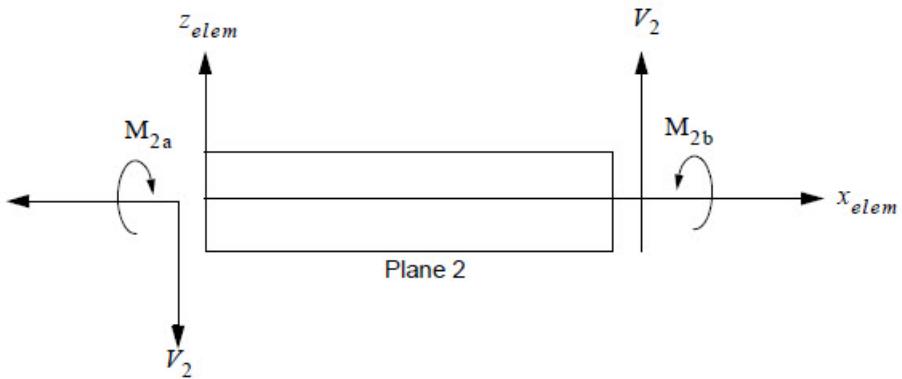


Figure 1-7 CBAR Element Internal Forces and Moments (x-z Plane)

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	Value
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 ≥ 0.0 , I2 ≥ 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

The description for the format used is as follows:

- 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 is the area moment of inertia for bending in plane 1 (same as Izz, bending about the z element axis)
 - I2 is the 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 1-2](#)).
 - K1 and K2 depend on the shape of the cross section.
 - K1 contributes to the shear resisting transverse force in plane 1.
 - K2 contributes to the shear resisting transverse force in plane 2.

Table 1-3 Area Factors for Shear

Shape of Cross Section	Value of K
Rectangular	$K_1 = K_2 = 5/6$
Solid Circular	$K_1 = K_2 = 9/10$
Thin-wall Hollow Circular	$K_1 = K_2 = 1/2$
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 (C_i, D_i, E_i, F_i) on the beam's cross section. These points are in the y-z plane of the element coordinate system as shown in [Figure 1-8](#).



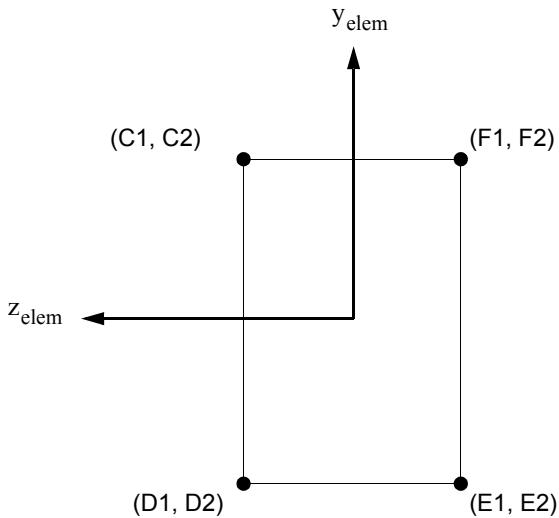


Figure 1-8 Stress Recovery Points on Beam Cross Section

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.

Surface Elements

Surface elements, also called two-dimensional elements, 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.

Note: As there is no stiffness associated to rotation about the normal axis to the plate, the rotational degree of freedom 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 1-9](#).

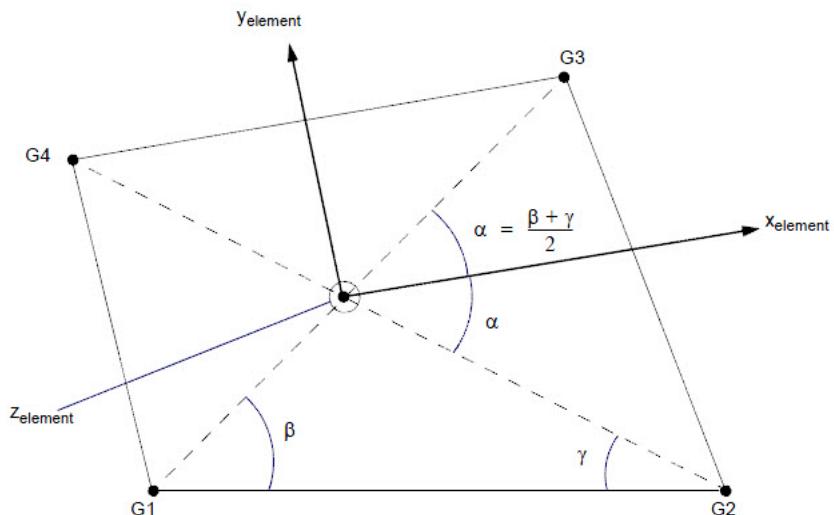


Figure 1-9 CQUAD4 Element Geometry and Element Coordinate System

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	Value
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.

The description for the format used is as follows:

- 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](#) (p. 1539) in 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. 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 1-10](#).

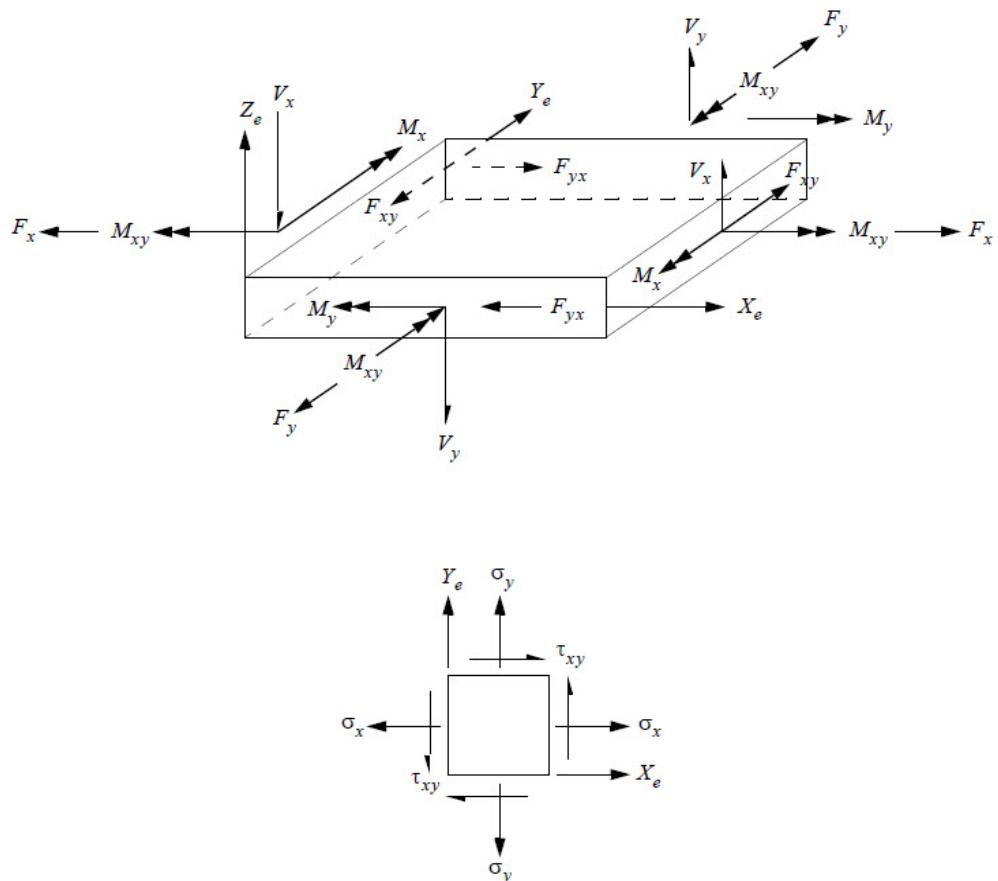


Figure 1-10 CQUAD4 Element Force, Moment, and Stress Conventions

The calculations performed are as follows:

- 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)

CTRIA3



The CTRIA3 element is a triangular plate connecting three grid points, as shown in Figure 1-11.

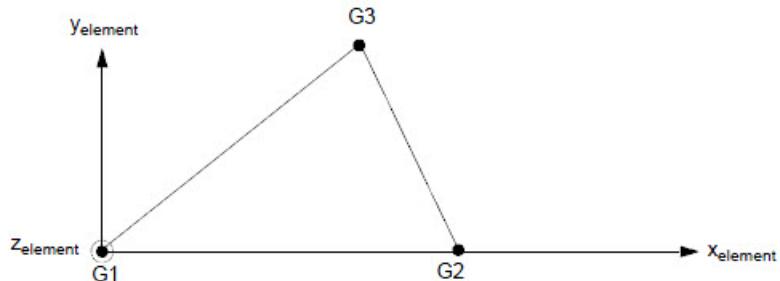


Figure 1-11 CTRIA3 Element Geometry and Element Coordinate System

Note: 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
CTRIA 3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
			T1	T2	T3				



Field	Contents	Value
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
THE TA	Material property orientation angle in degrees.	Real; Default = 0.0
MCID D	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](#) (p. 1596) in 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 1-11](#)) 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	Value
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
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 = ± 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

The description of the format used is as follows:

- 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:

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

Transverse shear flexibility 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.

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 in. 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 1-12](#).



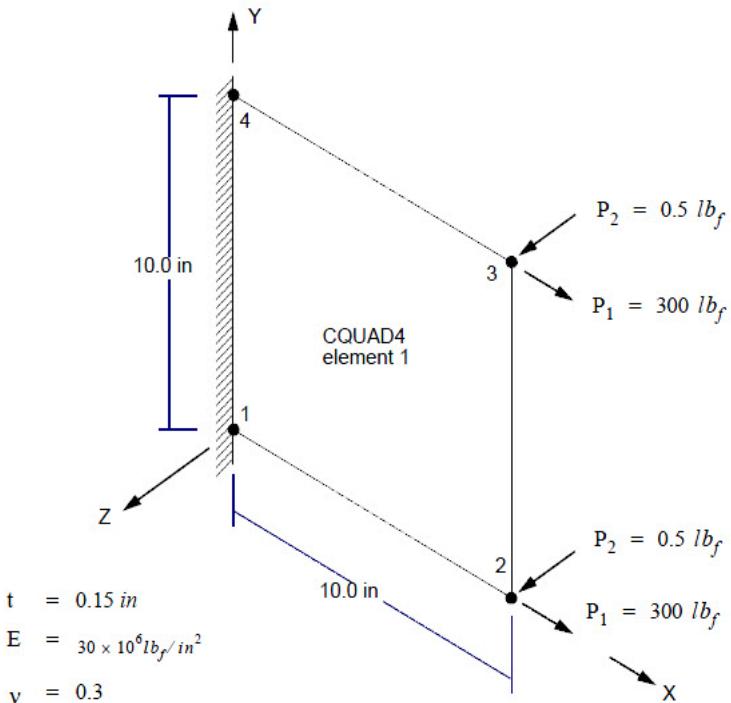


Figure 1-12 Cantilever Plate Example

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.				
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
```

For the complete input file see `Install_dir/mscxxxx/gt06quad4.dat`. The grid point displacement output is shown in [Listing 13-1](#).

DISPLACEMENT VECTOR									
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	1.303332E-04	2.999957E-05	3.795755E-03	1.333333E-04	-5.792592E-04	1.499978E-06		
3	G	1.303332E-04	-2.999957E-05	3.795755E-03	-1.333333E-04	-5.792592E-04	-1.499978E-06		
4	G	0.0	0.0	0.0	0.0	0.0	0.0		

Listing 13-1 Grid Point Displacement

Here are some checks and observations you can make in examining the deflections shown in [Listing 13-1](#):

- The maximum deflections of 3.796E-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.
- The free edge deflections are identical since the structure and loadings are symmetric.
- Grid points 1 and 4 have exactly zero displacement in all degrees of freedom, since they were constrained to be fixed in the wall.
- The lateral deflections occur in the T3 (+z) direction, which correspond with the direction of lateral loading. These displacements are reported in the displacement coordinate system, not the element coordinate system.
- The maximum lateral deflection of 3.796E-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 1-13](#).



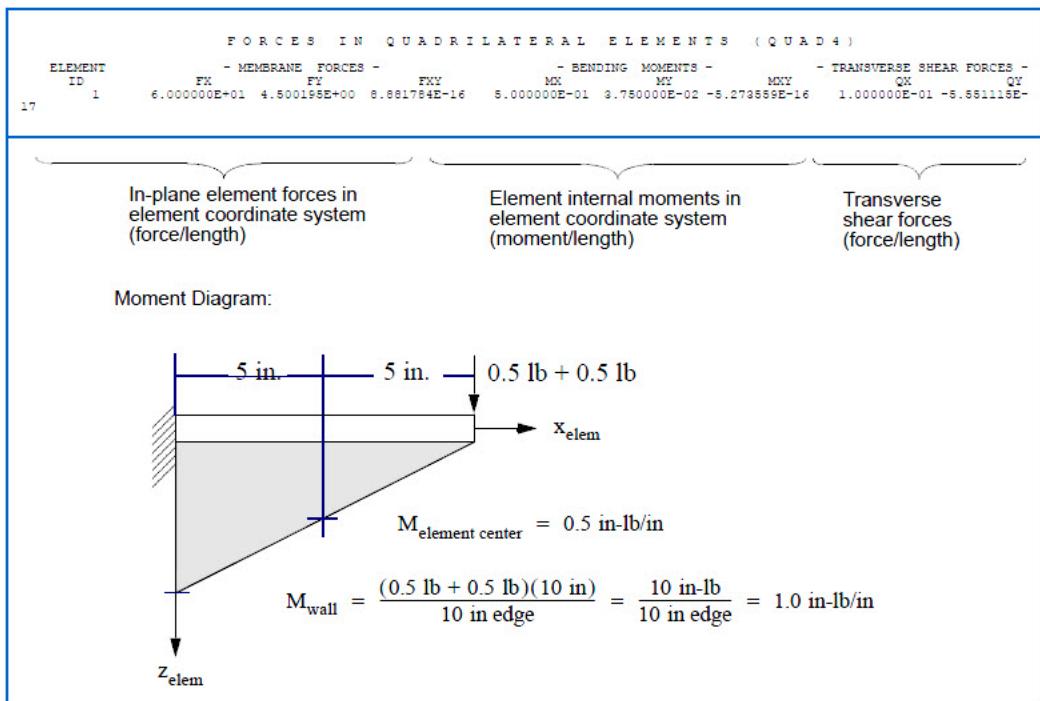


Figure 1-13 CQUAD4 Element Force and Moment Output

Note: Numbers such as -5.2735592-16 (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 1-14](#).

STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)					
ELEMENT ID.	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM	PRINCIPAL STRESSES (ZERO SHEAR)		
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE
1	-7.60000E-02	5.33333E-02	4.000180E+01	-1.34707E-13	0.0000
5.145002E+02	7.500000E-02	2.666667E-02	2.000180E+01	1.465494E-13	0.0000
2.572498E+02					
					VON MISES
					2.000180E-01

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

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

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

On the tensile surface,

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

On the compressive surface,

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

The von Mises stress on the compressive surface is given by

$$\sigma_{\text{von}}^- = \sqrt{\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2} = \sqrt{(266.67)^2 - (266.67)(20.0) + (20.0)^2 + 3(1.4E-13)^2}$$

$$\sigma_{\text{von}}^- = 257.2 \text{ lb/in}^2$$

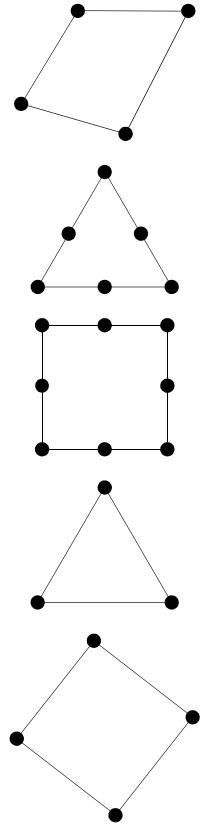
Figure 1-14 CQUAD4 Element Stress Output



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.



Solid Elements

MSC Nastran solid (three-dimensional) elements are used to represent the behavior of thick plates and solids. The principal solid elements are:

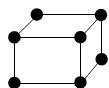
- The six-sided CHEXA
- The five-sided CPENTA
- The four-sided CTETRA

Note: 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 1-15](#).

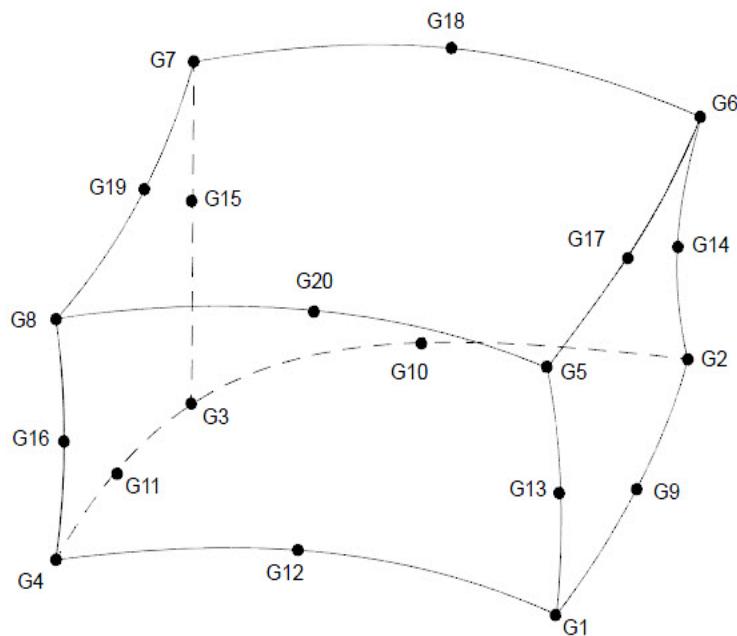


Figure 1-15 CHEXA Element Connection



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	Value
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

The description of the format used is as follows:

- 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.

Note: 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 1-16](#).



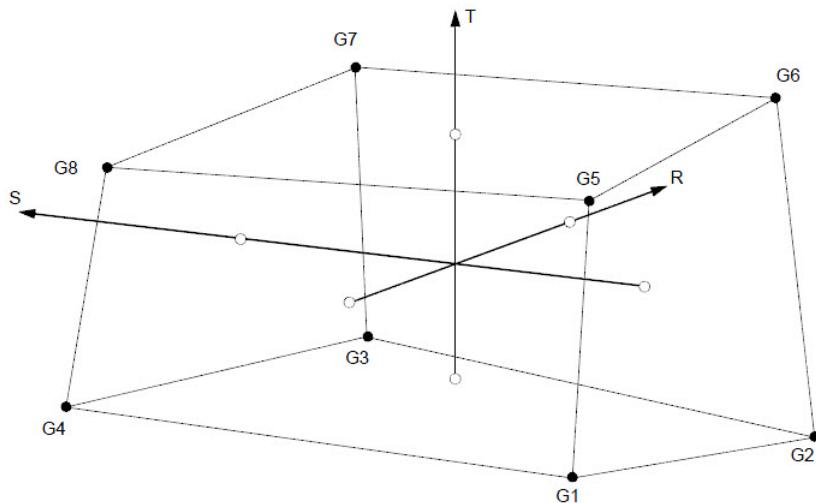


Figure 1-16 CHEXA Element Coordinate System

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 1-17.

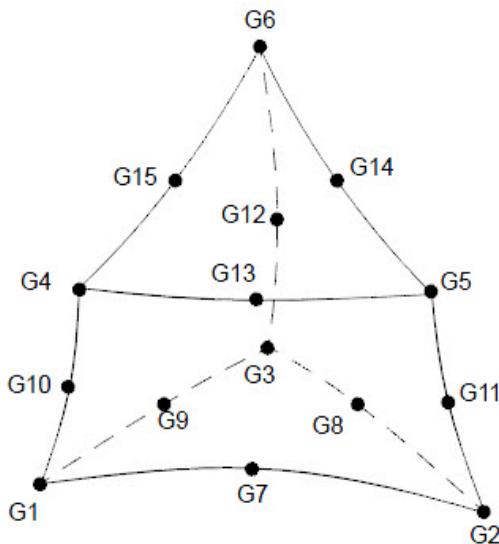


Figure 1-17 CPENTA Element Connection

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	Value
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

The description of the format used is as follows:

- 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.



Note: 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 1-18](#).

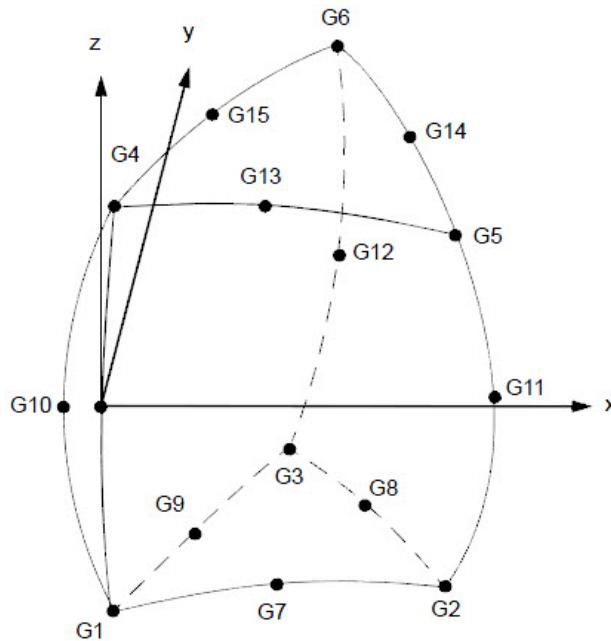


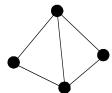
Figure 1-18 CPENTA Element Coordinate System

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 1-19](#).

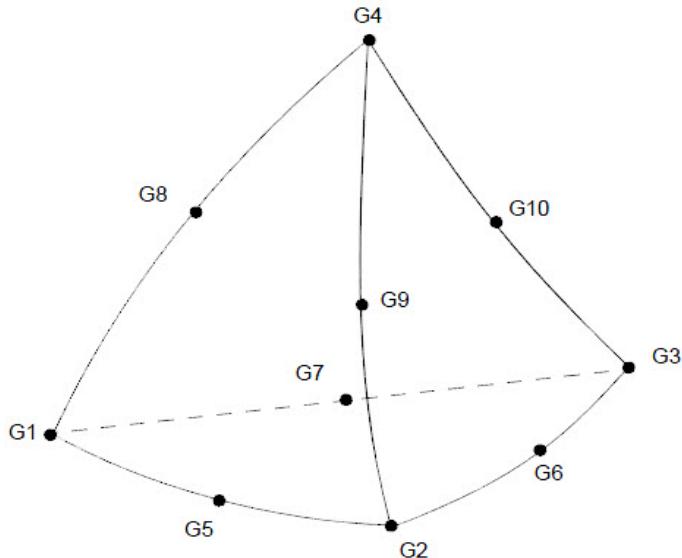


Figure 1-19 CTETRA Element Connection



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	Value
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

The description of the format used is as follows:

- Grid points G1, G2, and G3 define a triangular face.
- The midside nodes, G5 to G10, must be located as shown on [Figure 1-19](#).

Note: 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 1-20](#).

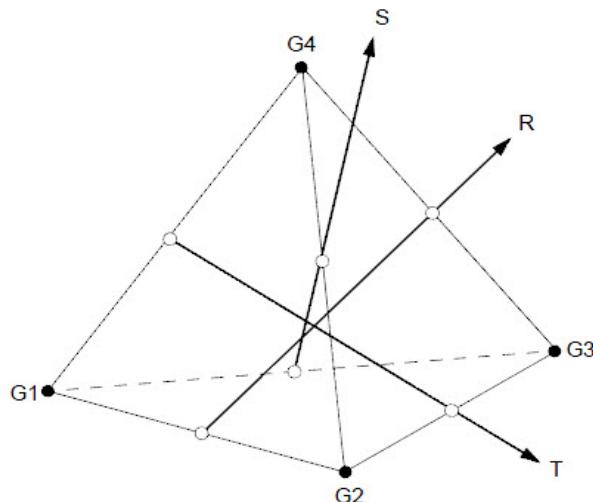


Figure 1-20 CTETRA Element Coordinate System



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	COROT	

Example:

PSOLID	2	100	6	TWO	GRID	REDUCED	PORO		
--------	---	-----	---	-----	------	---------	------	--	--

Field	Contents	Value
PID	Property identification number.	Integer > 0
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, MAT10, MATHP, MATPE1, or MATHE entry.	Integer > 0
CORDM	Identification number of the material coordinate system.	Integer; Default = 0, which is the basic coordinate system
IN	Integration network.	Integer; Character, or blank
STRESS	Location selection for stress output.	Integer; Character, or blank
ISOP	Integration scheme.	Integer; Character, or blank
FCTN	Fluid element flag.	Character: FFLUID indicates a fluid element with frequency dependent rigid absorber properties, PFLUID indicates a fluid element, SMECH indicates a structural element, PORO indicates an element of poroelastic medium; Default = SMECH.



COR OT	Corotational request. SOL 700 only.			Integer; Default = 0.			
0	Do not rotate						
1	Force local coordinate system to rotate with element						

The description of the format is as follows:

- 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.

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. For example, if a a CBAR element with extremely large values of I1 and I2 is used to simulate a rigid connection, a numerically ill-conditioned problem may occur. The RBE2 element defines a rigid body whose independent degrees of freedom are specified at a single point and the 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.-	ALPHA				

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					



7 Material Properties

- Basic Material Property Definitions 152
- Material Definition (MAT1) 153
- Other Material Types 155

Basic Material Property Definitions

The description on basic material properties 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 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 related material properties are described as follows:

Linear	Deformation is 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 has the same properties at every point; it is uniform without irregularities. The material properties are independent of location within the material.
Isotropic	Material properties do not change with the direction of the material.
Young's modulus (Modulus of elasticity) E	E is the constant of proportionality relating stress-to-strain for uniaxial behavior 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 9-1](#):

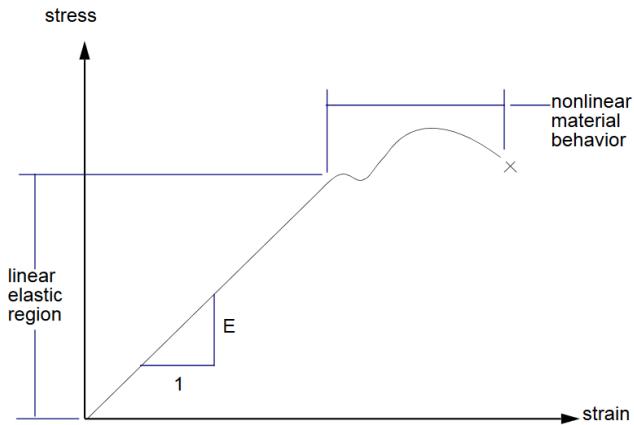


Figure 1-1 Stress-Strain Curve for Structural Steel

If the load 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 plastic (permanently) deformed state. Refer to [Nonlinear Analysis](#) for further description. Nonlinear methods are also required if the material is nonlinear in its elastic range.

Material Definition (MAT1)

Linear, elastic, homogeneous, isotropic materials are modeled in MSC Nastran with the MAT1 Bulk Data entry. The [MATBV](#) 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	Value
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. Refer to “ Parameter Descriptions ” on page 732 of the <i>MSC Nastran Quick Reference Guide</i> .	Integer ≥ 0 or blank

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

Only two of the three properties E, G, and NU, are required. The remaining value, if required, is calculated automatically according to the following relation:

$$G = \frac{E}{2(1 + v)}$$

E and G may not both be blank.

Some of the properties which are not applied in the stiffness formulation for certain elements are listed in [Table 9-1](#).



Table 1-1 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	Two of them are required. If all of them are supplied, they should satisfy $G = \frac{E}{2(1 + v)}$		

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 $E = 30 \times 10^6 \text{ lb}_f/\text{in}^2$, $v = 0.3$ and mass density $\rho_m = 7.0 \times 10^{-4} \text{ lb}_f\text{-sec}^2/\text{in}^4$.

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

In the free field format as:

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

Other Material Types

The following material types may be used for linear elastic behavior, too. Although they may be used for other type analysis, the discussion on it exceeds the scope of this book.



MAT2	Anisotropic material for two-dimensional elements.
MAT3	Orthotropic material for axisymmetric solid elements.
MAT8	Orthotropic material for two-dimensional elements.
MAT9	Anisotropic material for solid elements.
MATTORT	3-D orthotropic material behavior.
MATFTG	Fatigue behavior.
MATBV	Temperature-dependent material properties.

Material Options for Thermal Behavior

The material options for thermal behavior are:

MATBV	Thermal properties for temperature-independent isotropic materials.
MAT5	Thermal properties for temperature-independent anisotropic materials.



Material Options for Fluid Behavior

The material options for fluid behavior are:

- MAT10** Material properties for fluid elements in fluid-structure interaction analysis.
- MFLUID** Fluid volume properties for virtual mass.
- MATF1** Frequency-dependent poroelastic material.

Material Options for Structural Behavior

The material options to model nonlinear structural material behavior are:

- MATDIGI** Composite material with progressive failure.
- MATEP** Elastic-plastic material.
- MATF** Material failure models.
- MATG** Gasket material behavior.
- MATHE** General hyperelastic material behavior.
- MATHP** Generalized Mooney-Rivlin.
- MATUDS** User-defined subroutine.
- MATVE** Viscoelastic material behavior.
- MATVP** Viscoplastic and creep behavior.
- MCOHE** Cohesive zone material behavior.



8

Applying Constraints

- Description 160
- Rigid Body Motion and Mechanisms 160
- Single Point Constraints 161
- Automatic Identification and Removal of Singularities (AUTOSPC) 162
- Boundary Condition Examples 163

Description

A constraint is the application of a given displacement (i.e., component of translation or rotation) on a grid point or points.

Two basic types of constraints in MSC Nastran are:

- Single point constraints (SPCs): A single point constraint is applied to an individual grid point. Single point constraints can enforce either zero displacement or nonzero displacement
- Multipoint constraints (MPCs): A multipoint constraint is a mathematical constraint relationship between one grid point and another grid point (or set of grid points).

The boundary conditions of a static structure (fixed, hinged, roller support, etc.) normally involve various degrees of freedom to 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 recovered at grid points constrained by SPCs and are listed in the .f06 output file.

Rigid body motion and mechanisms describe how the boundary conditions are used to prevent rigid body motion. The presence of rigid body motion in MSC Nastran static analysis will cause the run to fail.

Single Point Constraints describe several ways to apply SPCs to grid points. Automatic Identification and Removal of Singularities (AUTOSPC) describes how MSC Nastran automatically detects and constrains singularities using a feature called AUTOSPC. Boundary condition examples gives several examples of modeling simple boundary conditions.

Rigid Body Motion and Mechanisms

The equation $\{F\} = [K]\{u\}$ requires $[K]$, 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 cannot proceed. Rigid body motion occurs when the structure moves freely in one or more directions of displacement, (displacement without strain). Rigid body motion causes singularities and must therefore be constrained.

Consider the two beams shown in Figure 10-1:

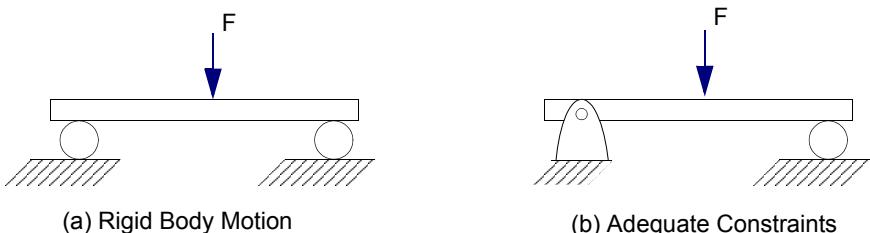


Figure 1-1 Rigid Body Motion vs. Adequate Constraints

Static Equilibrium

The beam (a) shown in the [Figure 1-1](#) is in static equilibrium. Displacement in the axial direction of the beam does not occur because there are no actual forces acting 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.

Mechanisms

A subclass of rigid body motion occurs when part of an otherwise constrained structure is capable of rigid body motion, 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. Details on inertia relief is beyond the scope of this book; for further information, refer to “[Chapter 11: Inertia Relief](#)” on page 477 of the *MSC Nastran Linear Static Analysis User’s Guide*.

Single Point Constraints

Single point constraints (SPCs) are applied to the displacements of individual grid points. In this section we shall discuss about applying 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:

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

Field	Contents	Value
SID	Identification number of the single point constraint set.	Integer > 0
Gi	Grid point identification number.	Integer > 0
Ci	Component number.	0 < 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

Automatic Identification and Removal of Singularities (AUTOSPC)

MSC Nastran automatically identifies and constrains singularities in the stiffness matrix with the use of a parameter called AUTOSPC. A grid point singularity table is printed in the output (.f06) file showing which degrees of freedom were constrained. If this table is present in the output, it is critical that you inspect the results carefully and understand which degrees of freedom were singular and why. It is possible that singular degrees of freedom are due to modeling errors.

A sample grid point singularity table is shown in [Figure 1-2](#). This table is from the single element cantilever rod model in a simple MSC Nastran model.



GRID POINT SINGULARITY TABLE							
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE UNION	NEW USET EXCLUSIVE UNION	SB	SB
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 1-2 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 degrees of freedom. Degree of freedom 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 degree of freedom was detected and constrained by AUTOSPC.

For further information on singularities and the grid point singularity table, see [Chapter 19: Grid Point Weight Generator](#) in the *Linear Static Analysis User's Guide*.

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

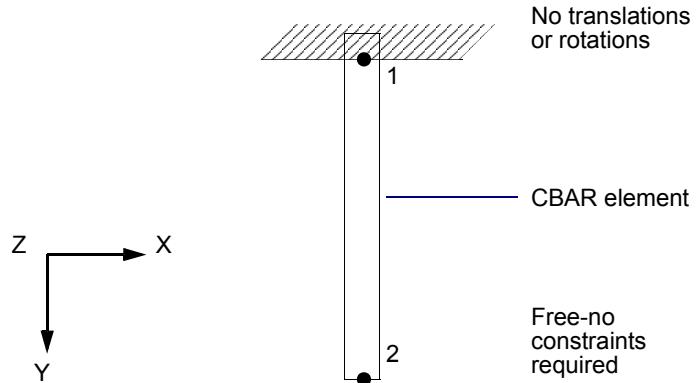


Figure 1-3 Cantilever Beam



Grid 1 must be constrained in all six degrees of freedom. 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		

In the free field format as:

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

Note: We do not need to specify a constraint set in the Case Control Section.

Example 2: Fixed Plate

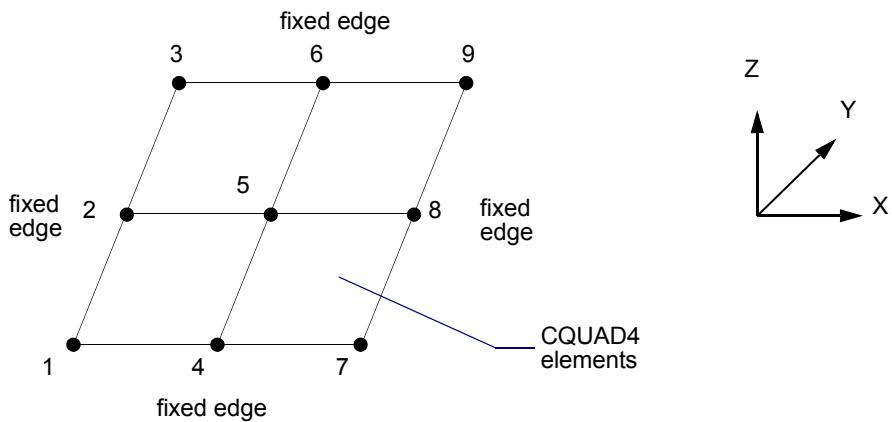


Figure 1-4 Fixed Plate

Grids 1, 2, 3, 4, 6, 7, 8, and 9 must be constrained in all six degrees of freedom. 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							



In the free field format as:

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

Note: “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

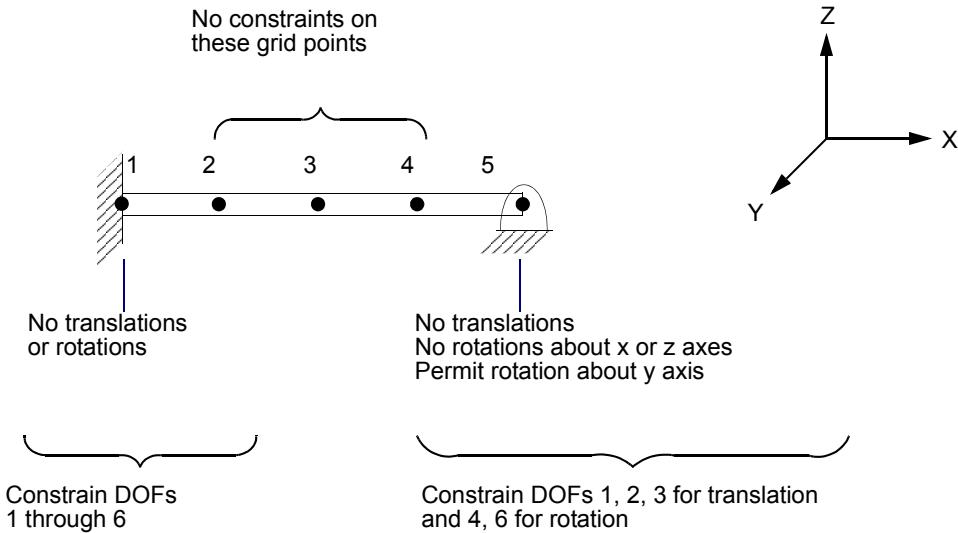


Figure 1-5 Fixed-Hinged Beam

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			

In the free field format as:



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

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. Be cautious while you model boundary conditions as accurately as possible.



9

Applying Static Loads

- Overview of Basic Static Loads 168
- Concentrated Loads 173
- Distributed Load on a 1-D Element (PLOAD1) 177
- Pressure Loads 182
- Acceleration Loads (GRAV) 186
- Enforced Displacements 189
- Combining Loads (LOAD) 190
- Using Subcases (SUBCASE, SUBCOM, SUBSEQ, SUBSEQ1) 192

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 1-1](#):



Table 1-1 Basic Static Loads

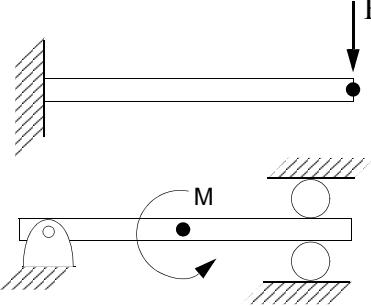
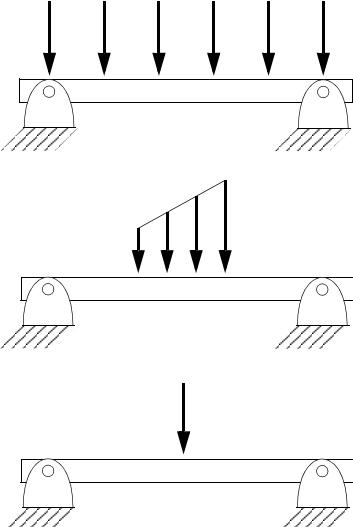
Description	Examples	Bulk Data Entry	Reference
Concentrated loads applied to grid points.		FORCE MOMENT	Concentrated Loads
Distributed or concentrated loads on line elements.		PLOAD1	Distributed Load on a 1-D Element (PLOAD1)



Table 1-1 Basic Static Loads (continued)

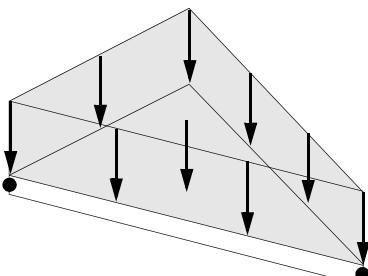
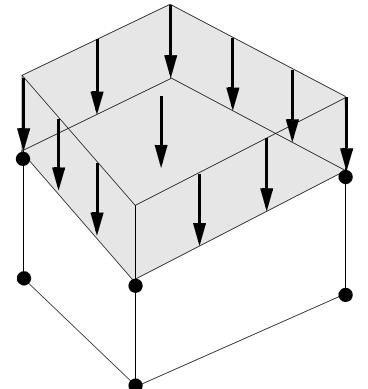
Description	Examples	Bulk Data Entry	Reference
Normal uniform pressure load on a triangular or quadrilateral surface (by defining grid points). This is for either 2-D or 3-D elements.	 	PLOAD	Pressure Loads



Table 1-1 Basic Static Loads (continued)

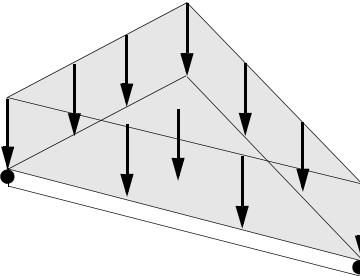
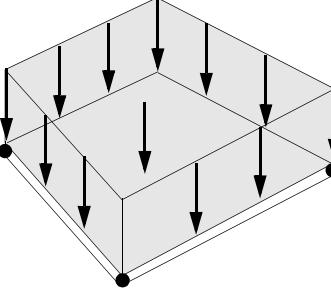
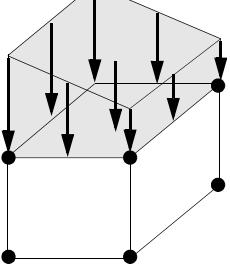
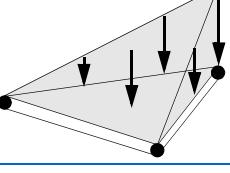
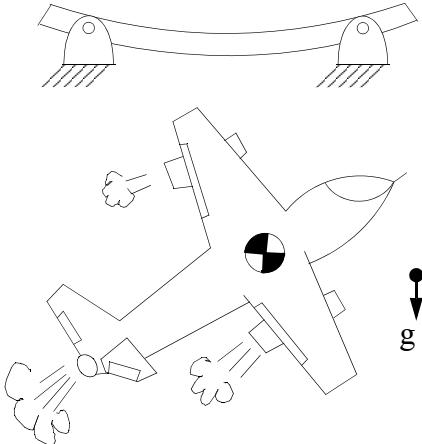
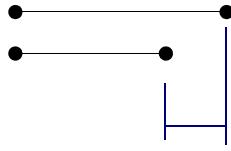
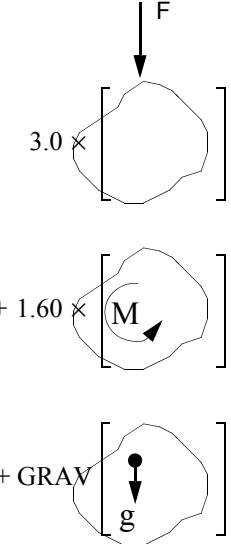
Description	Examples	Bulk Data Entry	Reference
Normal uniform pressure load on a 2-D element (by defining element IDs).	 	PLOAD2	Pressure Loads
Normal or traction pressure load on the face of a 2-D or 3-D element (by specifying element IDs). Pressure can be uniform or linearly varying.	 	PLOAD4	Pressure Loads



Table 1-1 Basic Static Loads (continued)

Description	Examples	Bulk Data Entry	Reference
Gravity or acceleration loads		GRAV	Acceleration Loads (GRAV)
Axial deformation of one-dimensional elements		DEFORM	Enforced Displacements
Linear combinations of loads		LOAD	Combining Loads (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 usage 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
<code>LOAD (Case)</code>	FORCE, FORCE1, FORCE2 MOMENT, MOMENT1, MOMENT2 PLOAD, PLOAD1, PLOAD2, PLOAD4 GRAV* LOAD* SPCD*
<code>DEFORM (Case)</code>	DEFORM

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

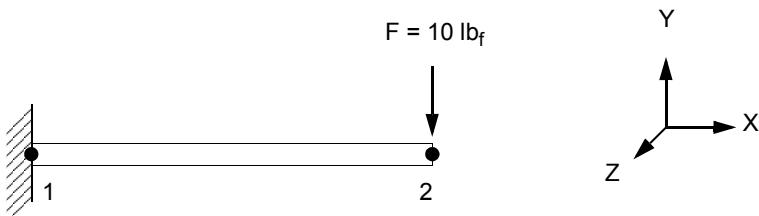
Concentrated Loads

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	Value
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 $N_i \neq 0.0$

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

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

And, in the free field format as:

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

$$\vec{f} = F \vec{N} \quad (9-1)$$

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:

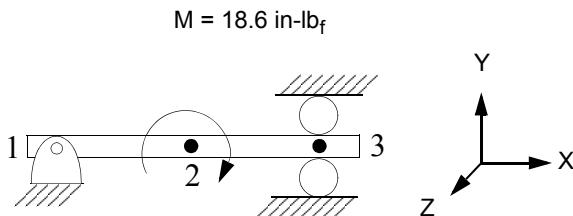
FORCE1 Uses two grid points. To define load direction it need not be the same as the loaded grid point.

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	Value
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.		

In the free field format as:

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



The applied moment \vec{m} is given by

$$\vec{m} = M\vec{N} \quad (9-2)$$

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. 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).

The choice of which entry to use is a matter of preference.

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 1-1](#).

Format:

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



Field	Contents	Value
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

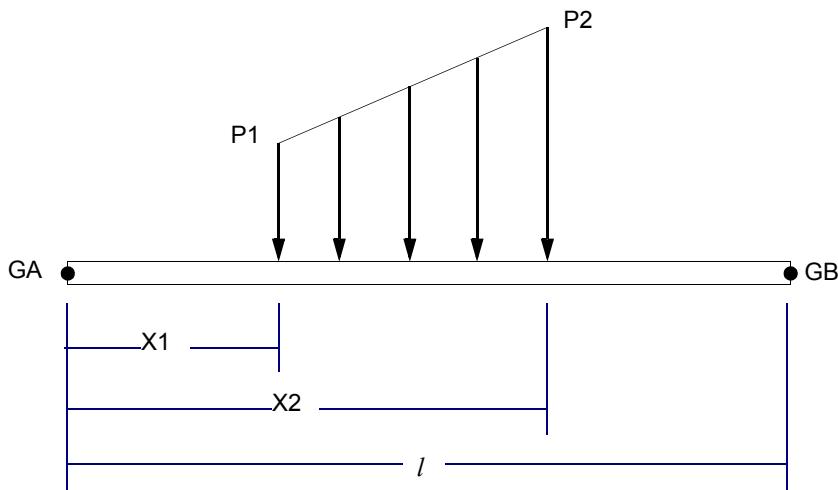


Figure 1-1 PLOAD1 Convention in CBAR or CBEAM Elements

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.



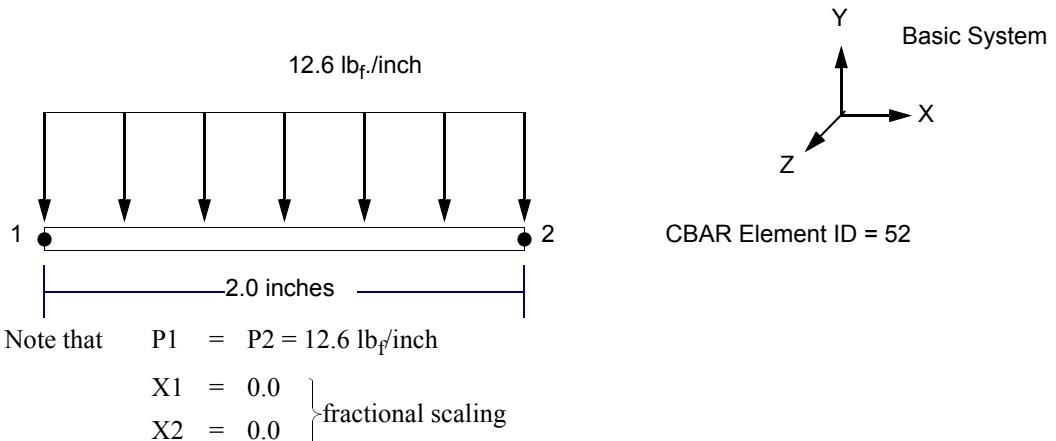
The SCALE factors mean the following:

LE	LENGTH	The values of X_i are the actual length of the element.
FR	FRACTIONAL	The values of X_i are fractional distances (fraction of length of element) along the length of the element starting from end A.
LEPR	LENGTH PROJECTED	The X_i 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 X_i 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.

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.

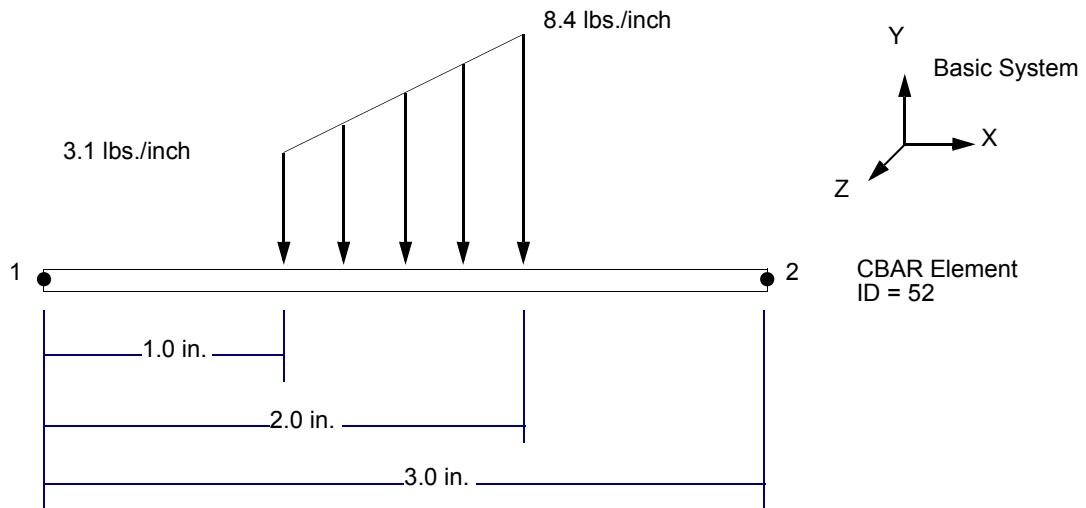


1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	FR	0.0	-12.6	1.0	-12.6	

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

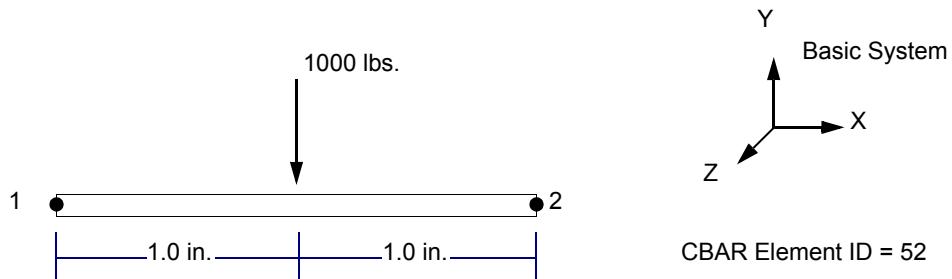
1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	LE	0.0	-12.6	2.0	-12.6	

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.			

Pressure Loads

Uniform Normal Pressure Load on Surface (PLOAD)

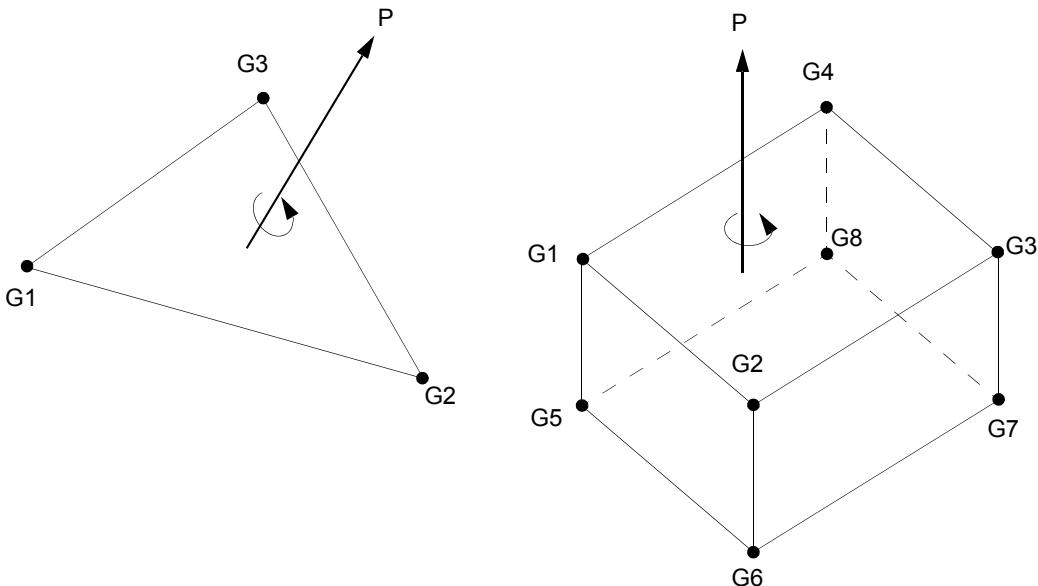
The PLOAD Bulk Data entry is used to define a uniform normal static pressure load on triangular or quadrilateral surfaces by using grid points. The **PLOAD** entry may be applied to 2-D (surface) or 3-D (solid) elements, and has the following form:

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

Field	Contents	Value
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	Value
SID	Load set identification number.	Integer > 0
P	Pressure value.	Real
EIDi	Element identification number.	Integer ≥ 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

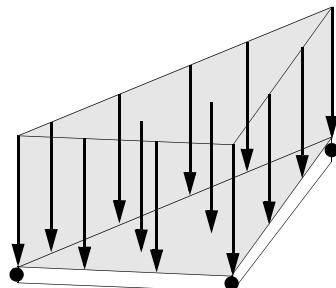


entry (i.e., with respect to the positive element z axis). In addition, when using the THRU option, all referenced elements must actually exist.

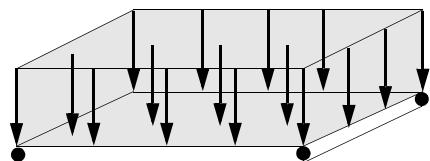
Normal or Traction Pressure Load on a Face (PLOAD4)

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:

Surface Elements:

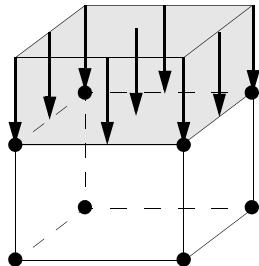


CTRIA3

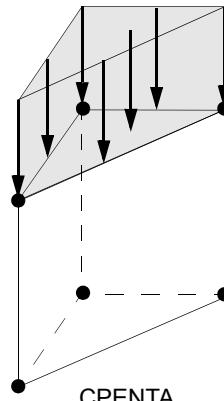


CQUAD4

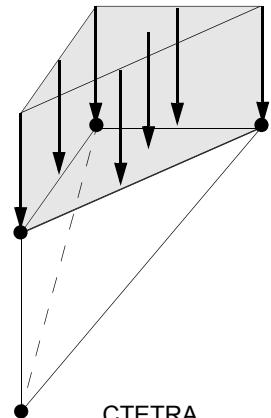
Solid Elements:



CHEXA



CPENTA



CTETRA

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 (for surface elements):

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

Field	Contents	Value
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

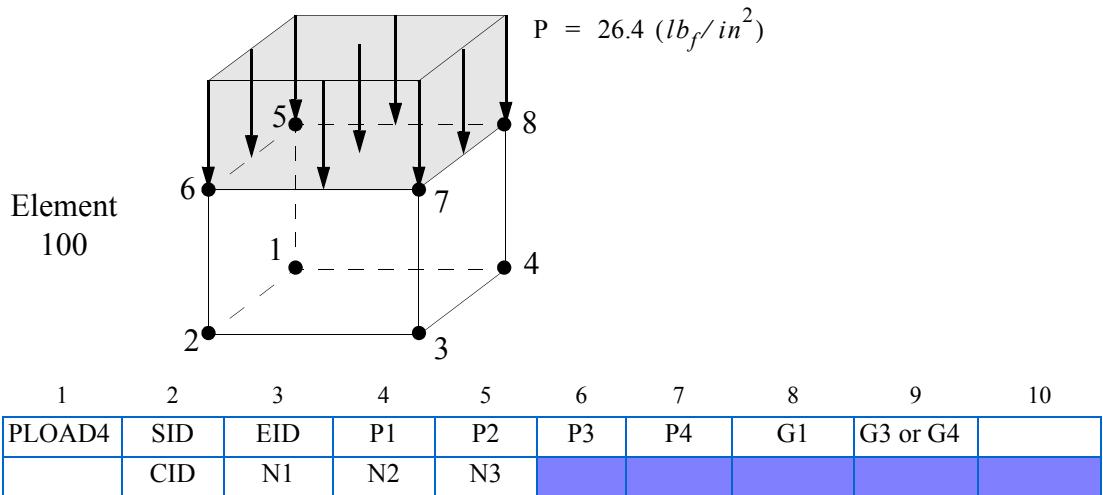
The load set ID (SID) is selected by the Case Control command LOAD = SID. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1; P4 is left blank for a triangular face. In addition, for pressure that acts normal to the face, the continuation entry is not used. For details about how to apply pressure loads in directions other than normal to the surface (traction loads) see the description of “[PLOAD4](#)” on page 2748 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:



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 26.4 lb/in².

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:

GRAV	SID	CID	A	N1	N2	N3	MB		
------	-----	-----	---	----	----	----	----	--	--



Field	Contents	Value
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 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: While entering the values, unit consistency must be preserved. For example, if the value of A is in in/sec² other specifications of the model that involve length units (element length, moments of inertia, modulus of elasticity, etc.) must also be in inches.

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 190 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:

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

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

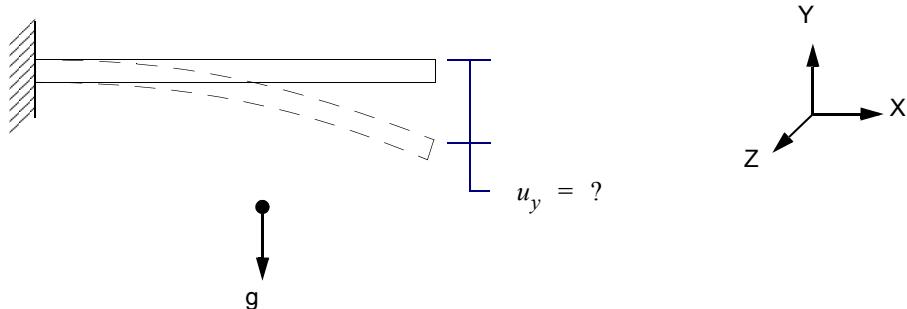
1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	12	30.E6		0.3	7.0E-4				

See [Material Definition \(MAT1\)](#), 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:

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

The acceleration due to gravity on the Earth's surface is approximately 32.2 ft/sec^2 (386.4 in/sec^2) or 9.8 m/sec^2 . Recall that length units must be consistent throughout the model (this example uses inches).

Hence, the **GRAV** entry in the tabular format is:

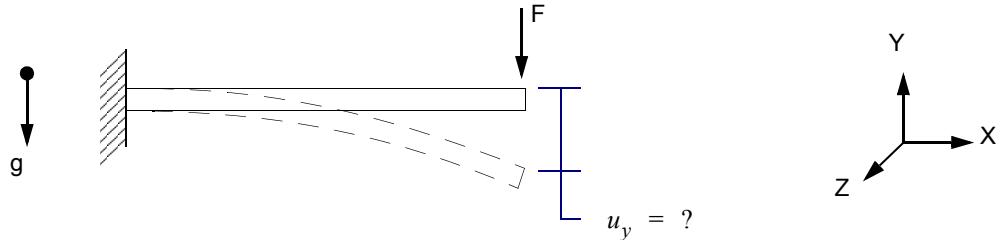
1	2	3	4	5	6	7	8	9	10
GRAV	15		386.4	0.	-1.	0.			

And, in the 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.

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

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

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

0.3 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 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.			

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	Values
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	Value
SID	Identification number of a static load set.	Integer > 0
Gi	Grid or scalar point identification number. (Integer > 0).	Integer > 0
Ci	Component numbers.	0 ≤ 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.

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	Value
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:

$$\text{LOAD} = S \sum_i Si \cdot \{Li\} \quad (9-4)$$

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. Hence,

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

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.			



Using Subcases (SUBCASE, SUBCOM, SUBSEQ, SUBSEQ1)

MSC Nastran allows you to efficiently analyze multiple load cases in one run using the **SUBCASE** (Case) 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** (Case) command. Coefficients defining the linear combination of a SUBCOM are specified on a **SUBSEQ** (Case) or a **SUBSTEP** (Case) command. A SUBSEQ or a SUBSEQ1 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 13-1.

Listing 13-1 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.

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

```
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
```

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:

$$\begin{aligned}\text{SUBSEQ} &= 1.0 \text{ (SUBCASE 1 Load)} + 1.0 \text{ (SUBCASE 2 Load)} + 0.0 \text{ (SUBCASE 3 Load)} \\ &= 1.0 \text{ (LOAD SET 10)} + 1.0 \text{ (LOAD SET 20)} \\ &= 1.0 \cdot \text{Dead Load} + 1.0 \cdot \text{NW Wind Load}\end{aligned}$$

Note: 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. However, the 0.0 placeholder must still be present.

In SUBCOM 20:

$$\begin{aligned}\text{SUBSEQ} &= 1.0 \text{ (SUBCASE 1 Load)} + 0.0 \text{ (SUBCASE 2 Load)} - 1.5 \text{ (SUBCASE 3 Load)} \\ &= 1.0 \text{ (LOAD SET 10)} - 1.5 \text{ (LOAD SET 30)} \\ &= 1.0 \cdot \text{Dead Load} - 1.5 \cdot \text{SW Wind Load}\end{aligned}$$

SUBSEQ1

An alternative method of performing load combinations is to use the SUBSEQ1 option. This is very benefit when a large number of loadcases/SUBCASE and only a few SUBCASES are combined. Using this technique, the alternative format for SUBCOM20 would have been:

```
SUBCOM20
SUBTITLE = Load Combination 2
Label = same
SUBSEQ1 = 1.0, 1.0, 1, -1.5, 3
```



10

Controlling the Analysis Output

- Printing the Input File (ECHO) 196
- Output Titles (TITLE, SUBTITLE, LABEL) 198
- Case Control Sets (SET) 199
- Requesting Analysis Results 200

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 [ECHO \(Case\)](#) Case Control command produces 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 appears 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 13-1](#) and [Listing 13-2](#).

Listing 13-1 Unsorted Listings

```
NASTRAN EXECUTIVE CONTROL ECHO
$ CHAPTER 12 EXAMPLE 1
SOL 101
CEND
HINGED BEAM
WITH CONCENTRATED FORCE

CASE      C O N T R O L      E C H O
COMMAND
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
HINGED BEAM
WITH CONCENTRATED FORCE
I N P U T      B U L K      D A T A      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
,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 ENTRY COUNT =      25
```



Listing 13-2 Sorted Listing

```
HINGED BEAM
WITH CONCENTRATED FORCE

          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-     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-    SPC1   100   12345   1     4
          ENDDATA
TOTAL COUNT=          13
```

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.

Output Titles (TITLE, SUBTITLE, LABEL)

Up to three title lines per page of MSC Nastran output are available by using the [TITLE \(Case\)](#), [SUBCASE \(Case\)](#), and [LABEL \(Case\)](#) 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

The Case Control commands listed below produce the following headings on each page of output:

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

SIMPLY SUPPORTED BEAM
WITH CONCENTRATED FORCE
MSC Nastran GETTING STARTED USER'S GUIDE

R	POINT ID.	TYPE	T1	T2	T3	R1	R2	V E C T O
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								

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 \(Case\)](#) command according to the following formats:

SET n = ALL

SET n = i₁, i₂, i₃, ..., i₁₂ THRU i₂₈, i₃₅, ...

where n is the set identification number and i₁, i₂, i₃, ..., 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.



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):

DISPLACEMENT (Case)=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.
SPCFORCES (Case)=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 (Case)=ALL or OLOAD=n	Requests the set of applied loads in static analysis.
GPFORCE (Case)=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 (Case)=ALL or STRESS=n	Requests element stresses for a set of structural elements.
FORCE (Case)=ALL or FORCE=n	Requests element forces to be calculated for a set of structural elements.
STRAIN (Case)=ALL or STRAIN=n	Requests the strains for a set of plate elements.
ESE (Case) =ALL or ESE=n	Requests the strain energy for a set of elements.

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*.



11

Additional Considerations

- MSC Nastran User and System Messages 204
- Epsilon-A Measure of Numerical Behavior 205
- Element Distortion and Accuracy 206
- The Current Error List 209
- The Comment (\$) Entry 209

MSC Nastran User and System Messages

MSC Nastran provides a variety of messages which indicate the status of your finite element model and the machine. Six types of basic messages are as follows:

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:

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

where

id: It is a unique message identification number.

text: It is the message displayed in capital letters for each of the diagnostic messages.

A series of asterisks (****) in the text indicates specific information that will be filled in the message by MSC Nastran, 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 corrective action.

User Messages

User messages are related to the finite element model (e.g., an incorrect input file).

- UIM: Describes the general information which may not necessarily refer to a problem.
- UWM: Indicates that a typical situation has been detected. You must go through the warning to determine whether the problem exists or not. 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.
- UFM: Describes errors that are severe enough which can lead to termination of MSC Nastran. The UFM usually appears at the end of the MSC Nastran .f06 output file.
- SIM/SWM or SFM: These messages are issued when system encounters a logical or operational condition with its Executive System, DMAP or accessing the Operating System (OS) operations.



System Messages

System Messages refer to diagnostics related to program errors. A short excerpt from the message list is shown in [Figure 1-1](#).

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 1-1 Excerpt from System and User Message List

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 file. Epsilon is based on a strain energy error ratio, and gives 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 mechanisms.

Epsilon should be one of the first items you must 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}) indicates numerical ill-conditioning and requires further investigation.



The possible causes of a large epsilon include the following:

- Rigid body motion
- The presence of a mechanism
- Unreasonably stiff elements
- 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 message shown in [Figure 1-2](#).

```
*** 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 1-2 EPSILON Printout

Element Distortion and Accuracy

The accuracy of a finite element degrades as its shape is distorted. You 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
- Taper

CQUAD4 Aspect Ratio

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

$$\text{ASPECT RATIO} = \frac{a}{b}$$

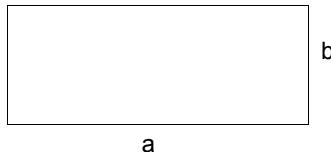


Figure 1-3 CQUAD4 Aspect Ratio



An element's aspect ratio a/b should be less than 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.

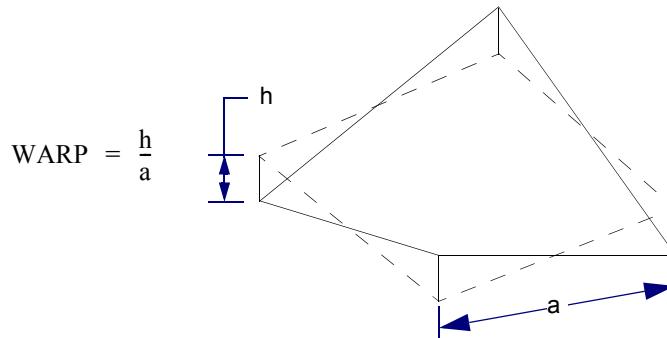


Figure 1-4 CQUAD4 Warp

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

CQUAD4 Skew

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

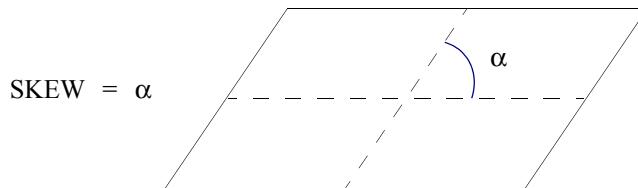


Figure 1-5 CQUAD4 Skew

For an element with no skew, $\alpha = 90^\circ$. When the skew angle is less than 30 degree, UIM 5491 is shown. 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 shown. As a rule, taper angles should not exceed 30 degree in most of the applications.



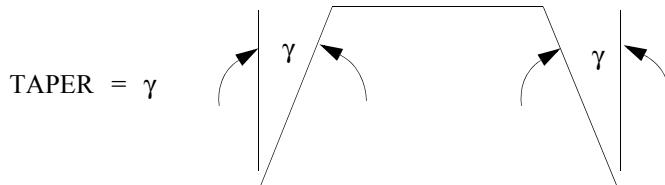


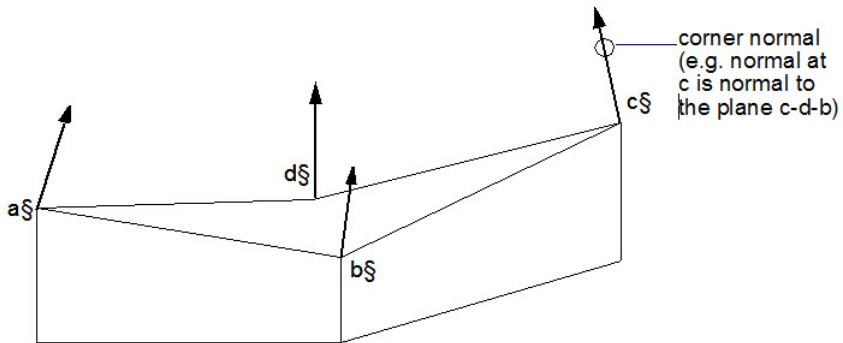
Figure 1-6 CQUAD4 Taper

CHEXA Solid Element Distortion

MSC Nastran makes two element geometry checks for CHEXA elements: aspect ratio and face warping. These checks are done because, the solution accuracy in regions of nonconstant 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 shown for that element.

Warping indicates that a face of a CHEXA is not planar, shown with User Information Message 4656, which indicates that a face is considerably out of plane. This message is shown if the corner normals at the opposite corners of a face deviate from each other by more than 45 degree, as shown in [Figure 1-7](#) (midside nodes are neglected in the calculation).



“Out of Plane” message is shown if the angle between normals at b and d > 45 degrees

“Out of Plane” message is shown if the angle between normals at a and c > 45 degrees

Figure 1-7 CHEXA Warping



The Current Error List

MSC Software maintains a comprehensive Current Error List (CEL) for recent versions of MSC Nastran and recommends that all users go through each edition of the Current Error List. The list is delivered with the MSC Nastran media and is also available online at:

<http://simcompanion.mscsoftware.com/infocenter/index?page=content&id=KI8008006>. The online version requires a login ID to the MSC Software store.

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.

An example of typical comment entries in a Bulk Data Section is shown in [Listing 13-1](#).

Listing 13-1 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
```



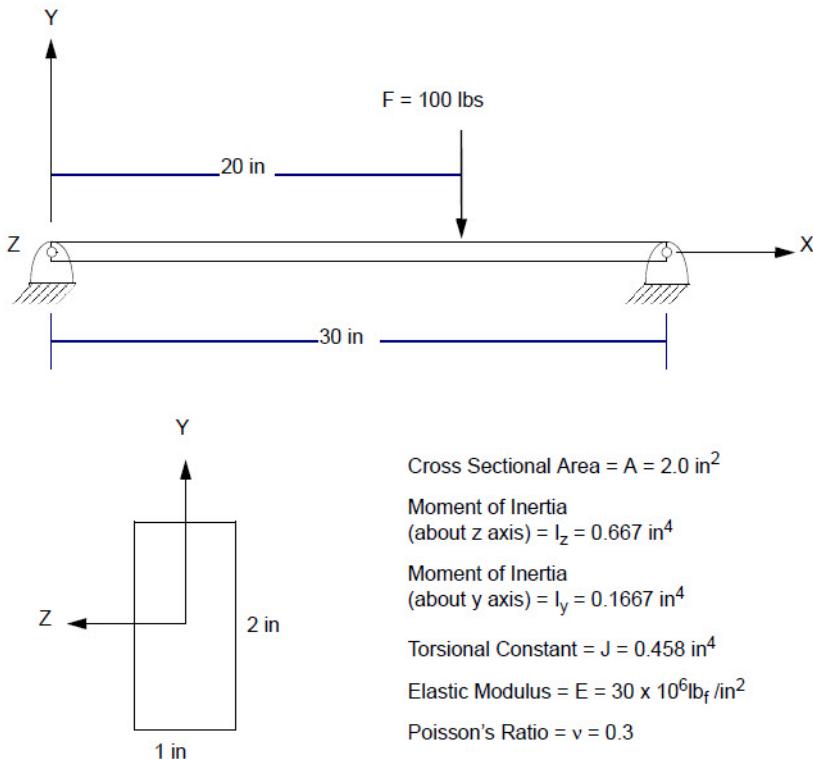
12 Performing Analysis

- Defining the Problem 212
- Specifying the Type of Analysis 212
- Designing the Model 213
- Creating the Model Geometry 214
- Defining the Finite Elements 216
- Specifying Material Properties 220
- Representing Boundary Conditions 221
- Applying the Loads 223
- Controlling the Analysis Output 224
- Completing the Input File and Running the Model 225
- MSC Nastran Output 226
- Reviewing the Result 230
- Comparing the Results with Theory 235



Defining the Problem

In this chapter, we perform a complete MSC Nastran analysis. Consider the hinged steel beam shown in Figure 1-1. 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.



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
```

Both the TIME and ID statements are optional. The default value of TIME, however, is too small for all but the most trivial problems. The format of the ID entry (ID i1,i2) must be adheared to or a fatal error will result.

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
```

```
CEND
```

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 1-2](#).

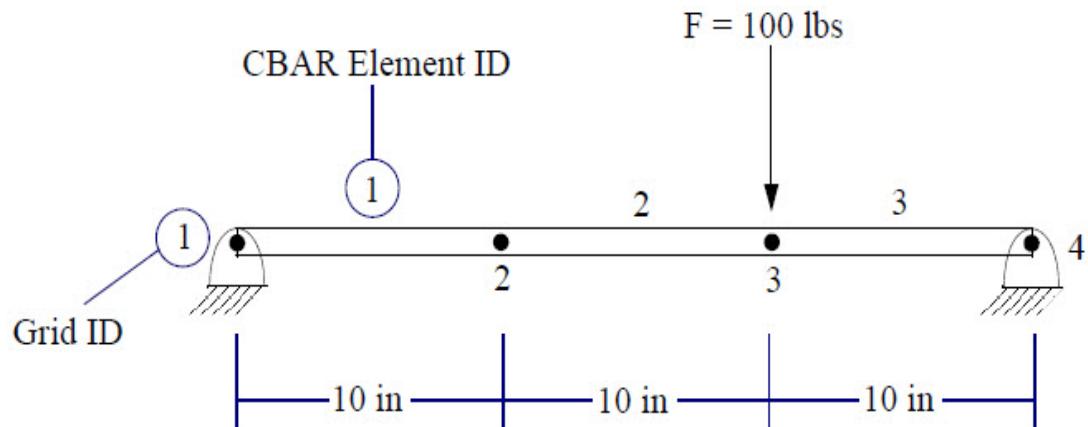


Figure 1-2 The Finite Element Model

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



Creating the Model Geometry

Coordinate System

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 1-3. 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.

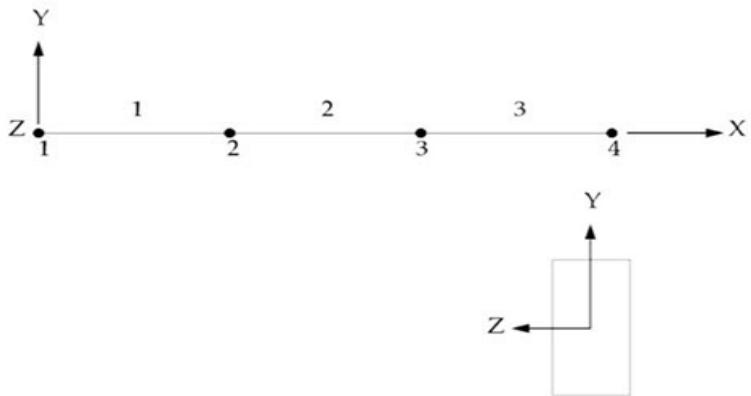


Figure 1-3 Model Coordinate System

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	Content	Value
ID	Grid point identification number.	$0 < \text{Integer} < 1000000$
CP	Identification number of coordinate system in which the location of the grid point is defined.	$\text{Integer} \geq 0$ or blank
X1, X2, X3	Location of the grid point in coordinate system CP.	Real; Default = 0.0

Field	Content	Value
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
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.
3	20.	0.	0.
4	30.	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.
```



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:

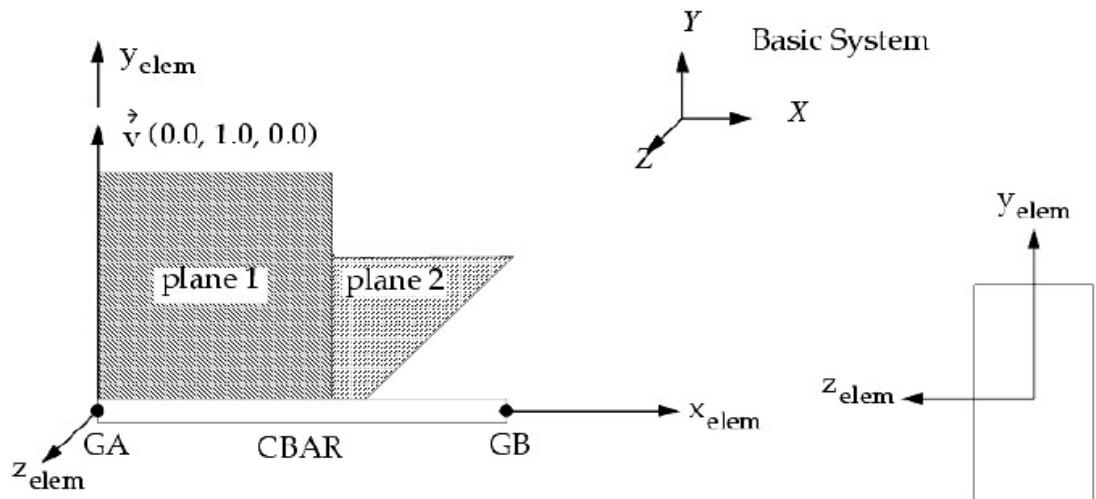
	1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3			
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B		

Field	Integer	Value
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 ≠ GB
X1, X2, X3	Components of orientation vector \mathbf{V} , from GA, in the displacement coordinate system at GA.	Real
G0	Alternate method to supply the orientation vector \mathbf{V} using grid point G0. Direction of \mathbf{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 \mathbf{w}_a and \mathbf{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 \mathbf{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 \mathbf{V} is shown in the following figure.



\mathbf{V} and x_{elem} defines Plane 1 and the y_{elem} Axis

Plane 1 is thus formed by \mathbf{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		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Field	Contents	Value
PID	Property identification number.	Integer > 0
MID	Material identification number.	Integer > 0
A	Area of bar cross section.	Real
I1, I2, I3	Area moments of inertia.	Real; $I1 \geq 0.0, I2 \geq 0.0, I1 - I2 > I12^2$
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 you will specify I1 and I2 in fields 5 and 6. Recall that the choice of orientation vector **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,

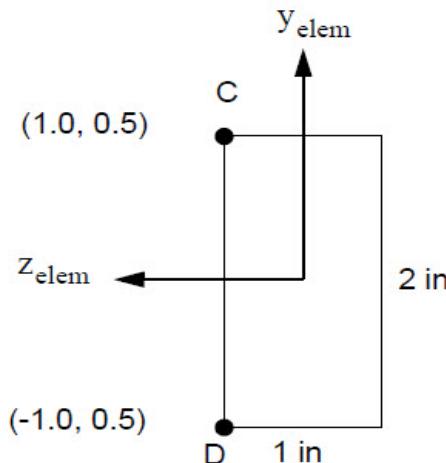
$$I1 = I_z = 0.667 \text{ in}^4, \text{ and } I2 = I_y = 0.1667 \text{ in}^4.$$

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).

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



Specifying Material Properties

The fields used and their properties are as follows:

Field	Contents	Value
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.

The beam's material is steel, with an elastic modulus of 0×10^6 lb/in. 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	Value
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

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:

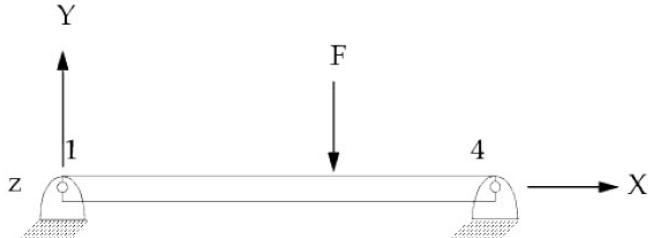
1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Field	Contents	Value
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

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 in the following figure:



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

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	Value
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.



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(PLOT)=ALL (to reduce output and time)
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
```



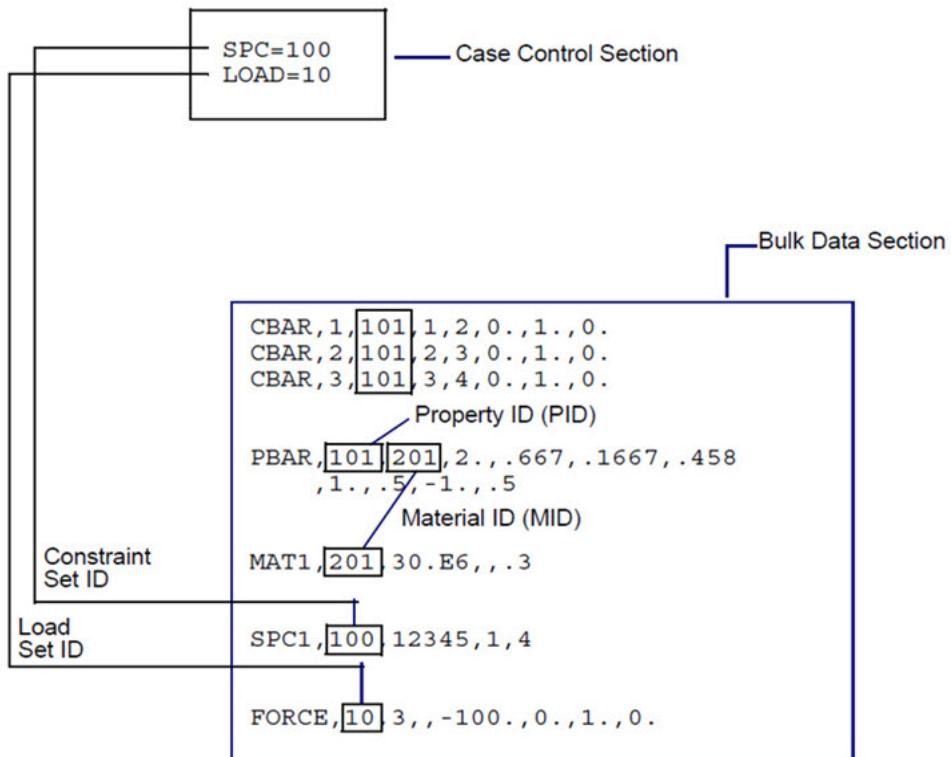
Completing the Input File and Running the Model

The completed input file (model without shear deflection) is called BASICEX1.DAT, and is shown as follows:

```
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 following diagram:



The job is submitted to MSC Nastran with a system command similar to the following:

```
~/nast20180 basicex1 scr=yes
```

(Travis, your comment also says “Do we tell them why we put scr=yes here?” Please suggest the info which can be added here

The details of the command are unique to your system; for more information refer the MSC Nastran Installation and Operations Guide.

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 13-1](#). Dates are not shown on the pages.



Listing 13-1 Complete .f06 Results File

Warning: This computer program is protected by copyright law and international treaties.
 Unauthorized use, reproduction or distribution of this computer program, or any portion of it, may
 result in severe civil and criminal penalties.
 Copyright (C) 2017 MSC.Software Corporation and its licensors. All rights reserved.

```
*****  

*          MSC Software  

*          CORP  

*  

*          M S C     N a s t r a n  

*  

*          Version 2017 1.0- CL479603  

*  

*          APR5, 2017  

*  

*          x86-64 Based System  

*          MODEL Intel 2600 MHz (subpcsr01)  

*          Linux 3.10.0-229.el7.x86_64  

*          Compiled for x86_64 (SINGLE Mode)  

*****
```

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
 CEND

1 HINGED BEAM	PAGE	3		
WITH CONCENTRATED FORCE	CASE	CONTROL	DECK	ECHO
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				



228 | MSC Nastran Getting Started Guide
MSC Nastran Output

Listing 13-1 Complete .f06 Results File (continued)

1 HINGED BEAM	PAGE	4	
WITH CONCENTRATED FORCE			
<pre> 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 +PB1,1.,.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 </pre>			
1 HINGED BEAM	PAGE	5	
WITH CONCENTRATED FORCE			
<pre> C A R D S O R T E D B U L K D A T A E C H O C O U N T . 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 .. 1- CBAR 1 101 1 2 0. 1. 0. 1. 0. 2- CBAR 2 101 2 3 0. 1. 0. 1. 0. 3- CBAR 3 101 3 4 0. 1. 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 11- +PB1 1. .5 -1. .5 12- SPC1 100 12345 1 4 ENDDATA TOTAL COUNT= 13 </pre>	+PB1		
1 HINGED BEAM	PAGE	6	
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.			
DAREA ID TYPE T1 T2 T3 R1 R2 R3	OLOAD	RESULTANT	
TOTALS 0.000000E+00 -1.000000E+02 0.000000E+00 0.000000E+00 0.000000E+00 -2.000000E+03			
*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)			
THE DECOMPOSITION OF KLL YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF		7.111111E+00	



Listing 13-1 Complete .f06 Results File (continued)

```

1 HINGED BEAM PAGE 7
WITH CONCENTRATED FORCE

0
*** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO. EPSILON EXTERNAL WORK EPSILONS LARGER THAN 0.001 ARE FLAGGED
WITH ASTERISKS
1 -4.7606364E-16 1.1105558E-01

1 HINGED BEAM PAGE 8
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

DAREA ID TYPE T1 T2 T3 R1 R2 R3
TOTALS 0.000000E+00 1.000000E+02 0.000000E+00 0.000000E+00 0.000000E+00 2.000000E+03

1 HINGED BEAM PAGE 9
WITH CONCENTRATED FORCE
DISPLACEMENT VECTOR

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

1 HINGED BEAM PAGE 10
WITH CONCENTRATED FORCE
FORCES OF SINGLE-POINT CONSTRAINT

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

1 HINGED BEAM PAGE 12
WITH CONCENTRATED FORCE
FORCES IN BAR ELEMENTS (CBAR)

ELEMENT BEND-MOMENT END-A BEND-MOMENT END-B - SHEAR - AXIAL
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 6.666666E+02 0.0 -3.333333E+01 0.0 0.0
0.0 3 6.666667E+02 0.0 7.958079E-13 0.0 6.666666E+01 0.0 0.0
0.0

1 HINGED BEAM PAGE 13
13 WITH CONCENTRATED FORCE

```



Listing 13-1 Complete .f06 Results File (continued)

1 HINGED BEAM WITH CONCENTRATED FORCE										PAGE	14
ELEMENT ID.	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)		
	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C			
1	0.0 -4.997502E+02	0.0 4.997502E+02	0.0 0.0	0.0 0.0	0.0	0.0 4.997502E+02	0.0 -4.997502E+02				
2	-4.997502E+02 -9.995003E+02	4.997502E+02 9.995003E+02	0.0 0.0	0.0 0.0	0.0	4.997502E+02 9.995003E+02	-4.997502E+02 -9.995003E+02				
3	-9.995003E+02 -1.193115E-12	9.995003E+02 1.193115E-12	0.0 0.0	0.0 0.0	0.0	9.995003E+02 1.193115E-12	-9.995003E+02 -1.193115E-12				

1 HINGED BEAM WITH CONCENTRATED FORCE										PAGE	15	
0 * * * * D B D I C T P R I N T * * * * SUBDMAP = PRTSUM , DMAP STATEMENT NO. 30												
* * * * 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												
----- F 0 0 1 1 ' ' T T T T F F T 0 F -1 0												

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 Result

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 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.

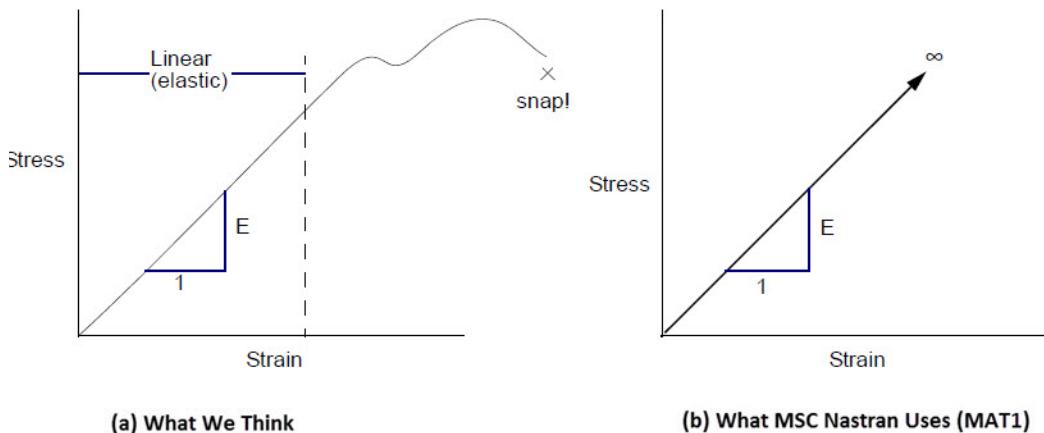


Figure 1-4 Reality Vs Modeling

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.

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 1-6). These values match the forces of single point constraint reported on page 10 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 1-7(b). 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.

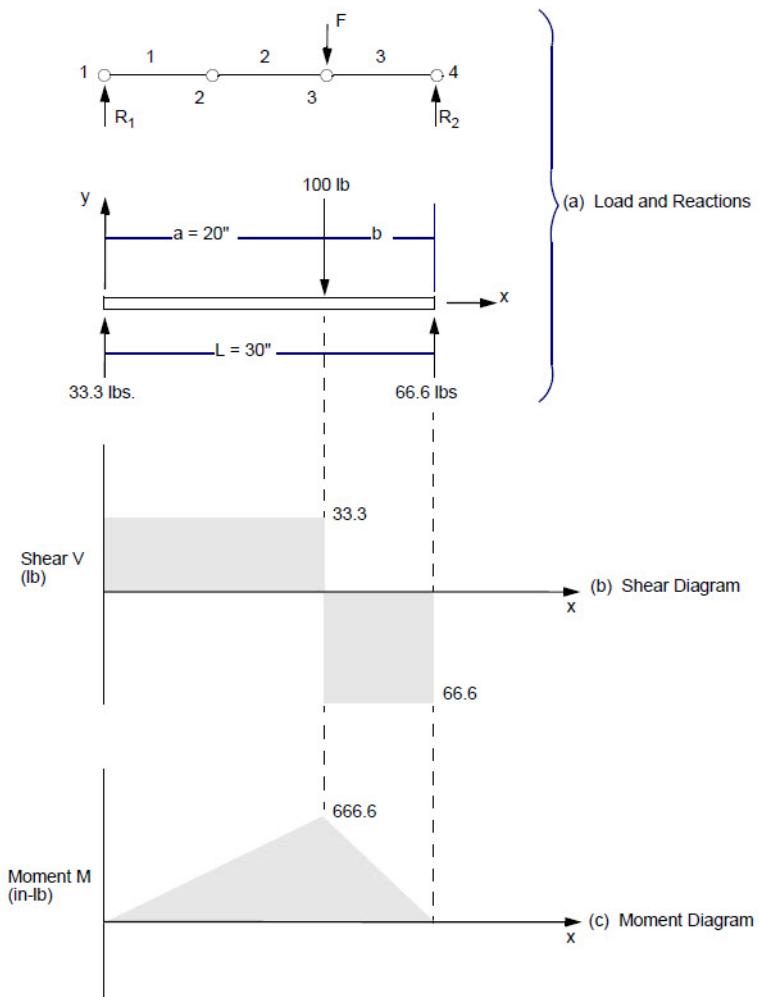


Figure 1-5 Beam Reaction Forces, Shear Diagram, and Moment Diagram

Note: Shear occurs only in plane 1 (the plane of the applied force).



The sign convention for CBAR element internal shear forces in Plane 1 ($x_{\text{elem}} - y_{\text{elem}}$ plane) is shown in the following figure:

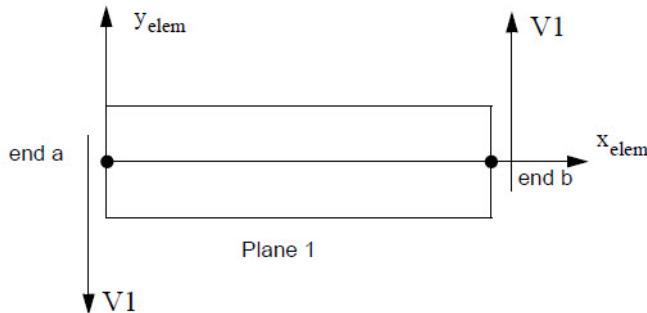


Figure 1-6 CBAR Element Shear Convention (Plane 1)

Thus, the signs make sense with respect to the applied load.

Displacement and Stress Results

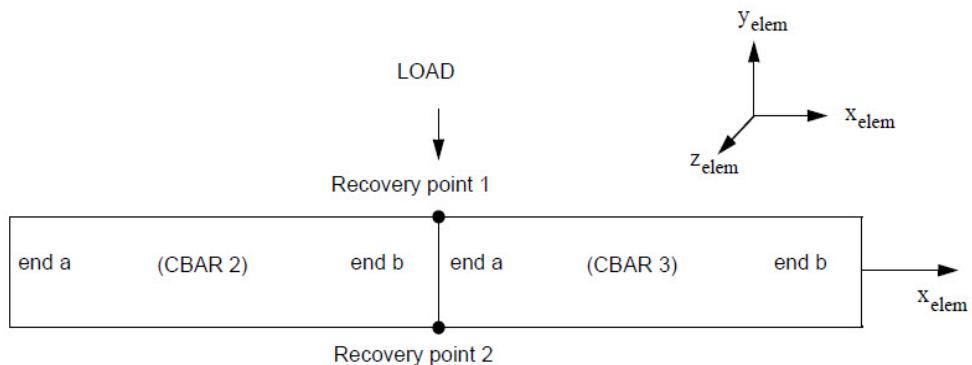
The displacement at the point of application of the load (GRID 3) is shown on page 9 of the results:

$$u_y^3 = -2.221112E-3 \text{ inch}$$

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 the following figure:



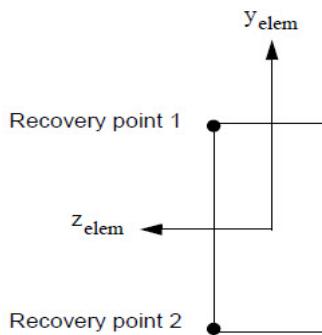


Figure 1-7 Bar Element Output Nomenclature

The MSC Nastran CBAR element stress output (Figures 6-7) is interpreted as shown in the following figure.

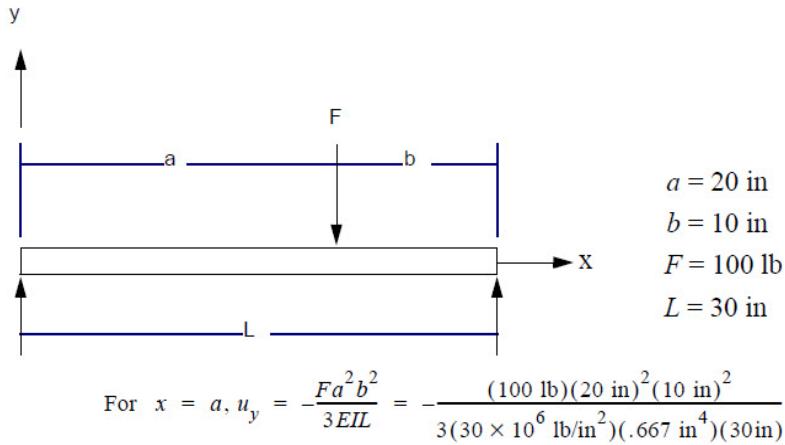
CBAR 2 End b, point 1		CBAR 2 End b, point 2	
HINGED BEAM WITH CONCENTRATED FORCE			
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3
1	0.0 -4.997502E+02	0.0 4.997502E+02	0.0 0.0
2	-4.997502E+02	4.997502E+02	0.0 0.0
3	-9.995003E+02 -9.995003E+02 -1.193115E-12	9.995003E+02 9.995003E+02 1.193115E-12	0.0 0.0 0.0
S T R E S S E S			
CBAR 3 End a, point 1 -999.5 lb/in ²		CBAR 3 End a, point 2 999.5 lb/in ²	

Figure 1-8 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 999.5 lb/in^2 (tension).

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 the MSC Nastran Output. The deflection due to bending only is calculated as follows:



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 Listing 1-2. See [MSC_DOC_DIR/doc/gt12ex1S.dat](#).

```

1 HINGED BEAM
WITH CONCENTRATED FORCE
CARD          S O R T E D   B U L K   D A T A   E C H O
COUNT
1-    . 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 ..
2-    CBAR   1      101   1      2      0.     1.     0.
3-    CBAR   2      101   2      3      0.     1.     0.
4-    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
11-   +0000011.   .5     -1.     .5
12-   +0000002  12345  12345  1      Shear Factor K:
13-   SPC1   100    12345  1      K1 = K2 = 5/6 = .8333 for rectangular
ENDDATA
TOTAL COUNT= 14

```

Figure 1-9 Shear Factor K on PBAR Entry



Shear Factor K: $K_1 = K_2 = 5/6 = .8333$ for rectangular sections

The deflection results are given in the output:

1 HINGED BEAM							PAGE	
WITH CONCENTRATED FORCE								
POINT ID.	TYPE	T1	DISPLACEMENT			R1	R2	R3
			T2	T3	R1			
1	G	0.0	0.0	0.0	0.0	-2.221112E-04		
2	G	0.0	-1.960807E-03	0.0	0.0	-1.388195E-04		
3	G	0.0	-2.255780E-03	0.0	0.0	1.110556E-04		
4	G	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 1-6\(c\)](#), 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.



13

Executing MSC Nastran

- Run MSC Nastran 238
- Using nastran Command 238
- Using Pre-processor to Execute MSC Nastran 240
- Checking the Analysis 242
- Interpreting the .f04 File 243
- Memory and Disk Usage Statistics 246
- Database Usage Statistics 247

Run MSC Nastran

You can execute MSC Nastran several ways, but the two recommended here are as follows:

1. Using a pre-processor
 - SimXpert
 - Patran
2. Direct nastran submission
 - Using Windows
 - Using Linux

Using nastran Command

MSC Nastran is executed from the command line using the command `nast20180`. This command sets environment variables in order for MSC Nastran to correctly execute on your system and it also has capabilities to predict memory and solvers (as of MSC Nastran 2018.0 with `solve=auto`). The basic format of the `nast20180` command is:

```
nast20180 input_data_file keywords
```

```
nast20180 input_data_file [keyword1=value1 keyword2=value2 ...]
```

where `input_data_file` is the name of the file containing the input data and `keyword=valuei` is one or more optional keyword assignment arguments. For example, to run an MSC Nastran job using the data file `example1.dat`, enter the following command:

```
nast20180 example1.dat
```

Various options to the `nast20180` command are available using keywords. Keyword assignments consist of a keyword, following by an equal sign, followed by a keyword value. For example,

```
nast20180 example1.dat memorymax=16gb
```

Note: In Windows you can use a hash mark # instead of the equal sign. This is useful if `nast20180 example1.dat memorymax=16gb` command is placed in .bat file.

```
nast20180 example1.dat memorymax=16gb
```

The details of submitting an MSC Nastran job are specific to your machine. Contact your IT personnel or refer to the MSC Nastran Installation Guide for further information. Keyword assignment can be specified in the command line or included in RC files.

The following sets of Run time Configuration (RC) files are controlled by you:

- Files that define parameters applicable to all MSC Nastran jobs that you run.



- Files that define parameters specific to all MSC Nastran jobs that reside in the input data files directory. They are located in the same directory as the input data file. If the `rcf` keyword is used, this local RC file is ignored.

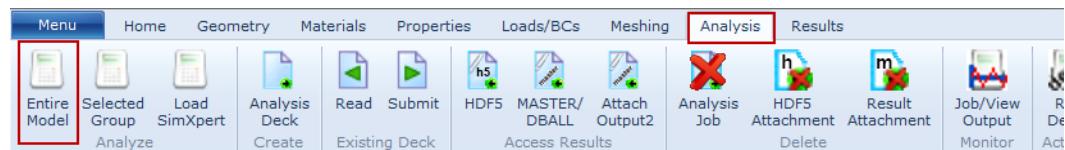
The location and names of these RC files are described in [App. A: Configuring the Runtime Environment](#) of [MSC Nastran Installation and Operations Guide](#).



Using Pre-processor to Execute MSC Nastran

Nastran can be executed using MSC Software's pre-processor Patran. Do the following steps to execute MSC Nastran using Patran:

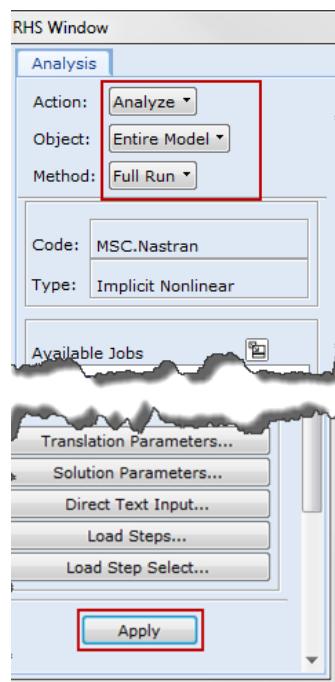
1. Select Analysis tab and select Entire Model.



The RHS Window is displayed.

2. In the RHS Window select the following:

- Action: Analyze
- Object: Entire Model
- Method: Full Run

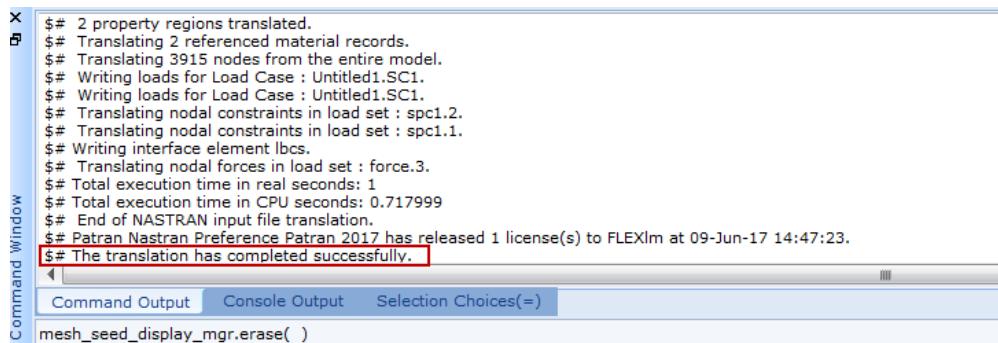


3. Click Apply.

This will execute the program in the command window.



4. The command window displays the status of translation.



```

Command Window
X $# 2 property regions translated.
$# Translating 2 referenced material records.
$# Translating 3915 nodes from the entire model.
$# Writing loads for Load Case : Untitled1.SC1.
$# Writing loads for Load Case : Untitled1.SC1.
$# Translating nodal constraints in load set : spc1.2.
$# Translating nodal constraints in load set : spc1.1.
$# Writing interface element lbc.
$# Translating nodal forces in load set : force.3.
$# Total execution time in real seconds: 1
$# Total execution time in CPU seconds: 0.717999
$# End of NASTRAN input file translation.
$# Patran Nastran Preference Patran 2017 has released 1 license(s) to FLEXlm at 09-Jun-17 14:47:23.
$# The translation has completed successfully.

Command Output Console Output Selection Choices(=)
mesh_seed_display_mgr.erase( )

```

5. After the problem is executed in Nastran, click Results.



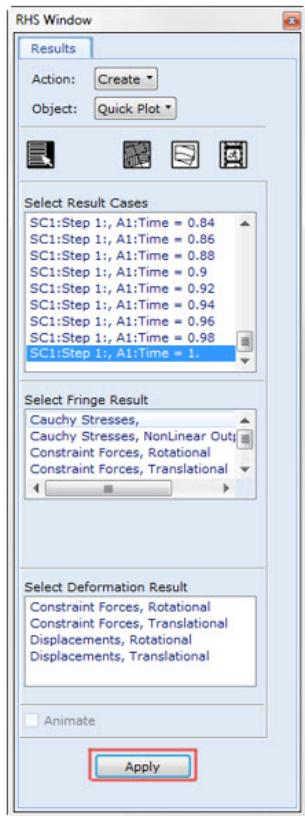
6. Under Results tab, select:

- Action: Create
- Object: Quick Plot

7. Select Result Cases and Result Type.



8. Click Apply to see the results.



Checking the Analysis

Once you submit the Nastran run, multiple files such as l0g, f04 and f06 gets created in same folder where your input file is located. These files are helpful to check progress as well as error messages. All these files are ascii files and can be opened in notepad or notepad++.

The Nastran files are text files and can be accessed through Notepad or Notepad++. The Nastran Input files are generated at the location where the input file is stored. The following statements will help you in checking if the analysis is done or not:

- In .log file one can check for NSEXIT = EXIT(0) to check if run completed successfully.



```

15:56:45 Analysis started.
15:56:45 Geometry access/verification to CAD part initiated (if needed).
15:56:45 Geometry access/verification to CAD part successfully completed (if needed).
15:56:45 Finite element model generation started.
15:56:45 Finite element model generated 3960 degrees of freedom.
15:56:45 Finite element model generation successfully completed.
15:56:45 Application of Loads and Boundary Conditions to the finite element model started.
15:56:45 Application of Loads and Boundary Conditions to the finite element model successfully completed.
15:56:56 27 records read from JTD file "e:/nastran_demo/ch_49/video_demo/ch49b.bdf"
15:56:56 NSEXIT: EXIT(0)
15:56:56 Analysis complete 0
Real:      13.801 seconds ( 0:00:13.801)
User:      11.076 seconds ( 0:00:11.076)
Sys:       0.811 seconds ( 0:00:00.811)
=====

```

- If it exits with other number, there is an error in the analysis, then check .f06 file for FATAL

```

*** USER INFORMATION MESSAGE 4158---STATISTICS FOR SYMMETRIC DECOMPOSITION OF DATA BLOCK
      NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL =      1
      MAXIMUM RATIO OF MATRIX DIAGONAL TO FACTOR DIAGONAL = 7.2E+15 AT ROW NUMBER
*** USER WARNING MESSAGE 4698.  STATISTICS FOR DECOMPOSITION OF MATRIX KLL

THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN 1.00000E+05 OR
NEGATIVE TERMS ON THE FACTOR DIAGONAL.
      GRID POINT ID      DEGREE OF FREEDOM      MATRIX/FACTOR DIAGONAL RATIO      MATR
      6714                  T1          -7.19297E+15           6.

^^^ DMAF FATAL MESSAGE 9050 (SEKRS) - RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS
IN MATRIX KLL. USER PARAMETER BAILOUT MAY BE USED TO CONTINUE THE RUN.

```

Note: If there is a USER FATAL MESSAGE or a SYSTEM FATAL MESSAGE occurrence, then the job did not complete successfully.

Interpreting the .f04 File

MSC Nastran writes information to the .f04 file that helps you to examine and tune the performance of your job. An overview of the complete .f04 file can be found in [Chapter 7: Solution Sequences](#) of the [MSC Nastran Reference Manual](#). This section describes detailed explanations of selected portions of the .f04 file.

Physical File Information

This summary table describes the physical files used for the DBsets. A sample of this table, located near the top of the .f04 file, is shown below.



SUMMARY OF PHYSICAL FILE INFORMATION			
ASSIGNED PHYSICAL FILE NAME	RECL (BYTES)	MODE	FLAGS
--	--	--	--
/tmp/65872_57.SCRATCH	8192	R/W	L
/tmp/65872_57.OBJSCR	8192	R/W	L
/tmp/65872_57.MASTER	8192	R/W	L
/tmp/65872_57.DBALL	8192	R/W	L
/tmp/65872_57.DBALL2	8192	R/W	L
/tmp/65872_57.SCR300	8192	R/W	L
/MSC(msc20170/linux64/SSS.MASTERA	8192	R/O	L
/MSC(msc20170/linux64/SSS.MSCOBJ	8192	R/O	L

FLAG VALUES ARE --

F	FFIO INTERFACE USED TO PROCESS FILE
H	HPIO INTERFACE USED TO PROCESS FILE
L	FILE HAS BEEN LOCKED
M	FILE MAPPING USED TO PROCESS FILE
R	FILE BEING ACCESSED IN 'RAW' MODE

** PHYSICAL FILES LARGER THAN 2GB FILES ARE NOT SUPPORTED ON THIS PLATFORM

From the above format, the file names describe the following :

ASSIGNED PHYSICAL FILENAME:	It is the physical filename with any symbols translated
RECL:	It is the record length in bytes
MODE:	It is the file access mode
R/W:	It is read-write mode
R/O:	It is read-only mode
FLAGS:	The column contains various letters which depends on the capabilities of the platform and user requests, the text below the table indicates flag values that are possible on the specific platform

In the above format, an INIT statement was used to create the DBALL DBset with two files using the logical names DBALL and DBALL2.

The following summary is a message which if large files (see “[Using Databases](#)” on page 73 of *MSC Nastran Installation and Operations Guide*.

Memory Map

Immediately following the summary of physical file information is a map showing the allocation of memory. This map is also described in “[Managing Memory](#)” on page 97 of *MSC Nastran Installation and Operations Guide*.



```
** MASTER DIRECTORIES ARE LOADED IN MEMORY.
USER OPENCORE (HICORE)      =     291 MB
EXECUTIVE SYSTEM WORK AREA   =      8 MB
MASTER (RAM)                 =      1 MB
SCRATCH(MEM) AREA            =     50 MB (100 BUFFERS)
BUFFER POOL AREA (BPOOL4)    = 7838 MB (15676 BUFFERS)

TOTAL MSC NASTRAN MEMORY LIMIT =     8192 MB
```

From the above summary, the master directories describe the following:

- USER OPENCORE: It is the amount of memory available to the module for computation purposes
- EXECUTIVE SYSTEM WORK AREA: It is the space reserved for the executive system
- MASTER (RAM): It is the space reserved to cache datablocks from the MASTER DBset
- SCRATCH (MEM) AREA: It is the space reserved to cache datablocks from the SCRATCH and SCR300 DBsets
- BUFFER POOL AREA: It is the space reserved for the buffer pool
- TOTAL MSC NASTRAN MEMORY LIMIT: It is the total space allocated to MSC Nastran's during the simulation

Day Log

The Day Log portion of the .f04 is a DMAP execution summary. This log, in table format, contains the vast majority of the information in the .f04. The beginning of the Day Log format is shown below:

DAY	TIME	ELAPSED	I/O MB	DEL_MB	CPU SEC	DEL_CPU	SUB_DMAP/DMAP_MODULE	MESSAGES
10:32:16		0:16	13.6	.3	.8	.0	SESTATIC	20 IFPL BEGN
		0:16	13.7	.1	.8	.0	IFPL	29 IFP1 BEGN
		0:16	13.7	.0	.8	.0	IFPL	39 XSORT BEGN

The format of the Day Log is as follows:

- DAY TIME: It is the time of day of the entry
- ELAPSED: It is the elapsed time since the start of the job
- I/O MB: It is the megabytes of I/O to the databases since the start of the job
- DEL_MB: It is the delta I/O since the previous entry
- CPU SEC: It is the total CPU seconds since the start of the job
- DEL_CPU: It is the delta CPU since the previous entry



SUB_DMAP/DMAP_MODUL E: It indicates the DMAP statement being executed

MESSAGES: These are any messages issued by the module

BEGN: It is the start of the module

END: It is the end

Note:

- The I/O MB value is computed by multiplying SYSTEM(85), which is incremented by one for each GINO I/O, by BUFFSIZE. This value will lose accuracy if the DBsets do not have the same BUFFSIZE.
- If SYSTEM (84) is set to 0, the I/O MB column will be the number of GINO I/Os.
- The I/O MB column will be scaled by gigabytes and a G will be appended after each number if the value is greater than or equal to 100 000.

Memory and Disk Usage Statistics

These tables are written after the job has completed, and indicate the maximum memory used by any sparse numerical module and the maximum disk used by any module during the job. The format is shown below.

SPARSE SOLUTION MODULES				MAXIMUM DISK USAGE			
HIWATER (WORDS)	DAY_TIME	SUB_DMAP NAME	DMAP MODULE	HIWATER (MB)	DAY_TIME	SUB_DMAP NAME	DMAP MODULE
517786	04:35:44	SEKRRS	18 DCMP	15.625	04:35:48	SESTATIC	186 EXIT

In the above table format, modules describe the following:

HIWATER WORDS: It is the maximum amount of open core used by certain sparse numerical modules

DAY_TIME: It is the time of day the module ran

SUB_DMAP NAME: It is the name of the SUBDmap

DMAP MODULE: It indicates the line number and module name that made the maximum request, Similarly, in the right hand table

HIWATER (MB): It is the maximum amount of disk space used by any module

DAY_TIME: It is the time of day the module ran

SUB_DMAP NAME: It is the name of the SUBDmap

DMAP MODULE: It indicates the line number and module name that made the maximum request



Database Usage Statistics

These statistics shown in the following format, summarize the I/O activity for the DBsets.

*** DATABASE USAGE STATISTICS ***										
DBSET TRANSFERRED	LOGICAL DBSETS			USED			DBSET FILES			I/O
	ALLOCATED (BLOCKS)	ALLOCATED (WORDS)	BLOCKSIZE (BLOCKS)	USED %	FILE	ALLOCATED (BLOCKS)	ALLOCATED (GB)	HIWATER (BLOCKS)	HIWATER (GB)	
MASTER	5000	65536	52	1.04	MASTER	5000	2.44	152	0.074	
1.449										
DBALL	2000000	65536	3	0.00	DBALL	2000000	976.56	3	0.001	
0.003										
OBJSCR	5000	8192	374	7.48	OBJSCR	5000	0.31	374	0.023	
0.529										
SCRATCH (0.000)	4000100	65536	291	0.01	(MEMFILE	100	0.05	100	0.049	
209.438					SCRATCH	2000000	976.56	28297	13.817	
24.080					SCR300	2000000	976.56	1992	00.973	
										TOTAL:
										235.498

The above statistical format contains two parallel tables. The LOGICAL DBSETS table lists each DBset while DBSET FILES tables lists the component files of the DBset. The description of the files is:

- | | |
|------------------------|--|
| DBSET: | It is the name of the DBset |
| ALLOCATED: | It is the MSC Nastran DBset size limit in blocks |
| BLOCKSIZE: | It is the BUFSIZE of the DBset minus one |
| USED(BLOCKS) and USED% | are the number of blocks and percent of the DBset used. SCRATCH is the amount used during cleanup. Refer to HIWATER for total use. |
| DBSET FILES | |
| FILE: | It is the file's logical name associated with the DBset to the left |
| ALLOCATED: | It is the number of BLOCKS/Gb allocated by MSC Nastran for the file |
| HIWATER: | It is the number of BLOCKS/Gb actually used in the file |
| I/O TRANSFERRED: | It is the amount if I/O to the file |

In this example, the MASTER and OBJSCR DBsets are each composed of one file. The MASTER, DBALL and OBJSCR are each composed of one file. The SCRATCH DBset has three components such as, MEMFILE, SCRATCH, and SCR300.

The above format can be used to determine if the DBsets and files are appropriately sized and the amount of I/O activity associated with each file. Best elapsed time performance can be obtained if the files with the greatest activity are on different physical devices (and better yet, separate I/O controllers or busses).





14 Performance Considerations

- Geometric Reduction Methods 250
- Memory and I/O Performance 253
- Parallel Performance 254



Introduction

MSC Nastran can solve equations of virtually unlimited size—problems have been solved in excess of 10,000,000 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.

Geometric Reduction Methods

The geometric reduction methods are:

- Superelement analysis
- Axisymmetric analysis
- Cyclic symmetry analysis

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 as follows:

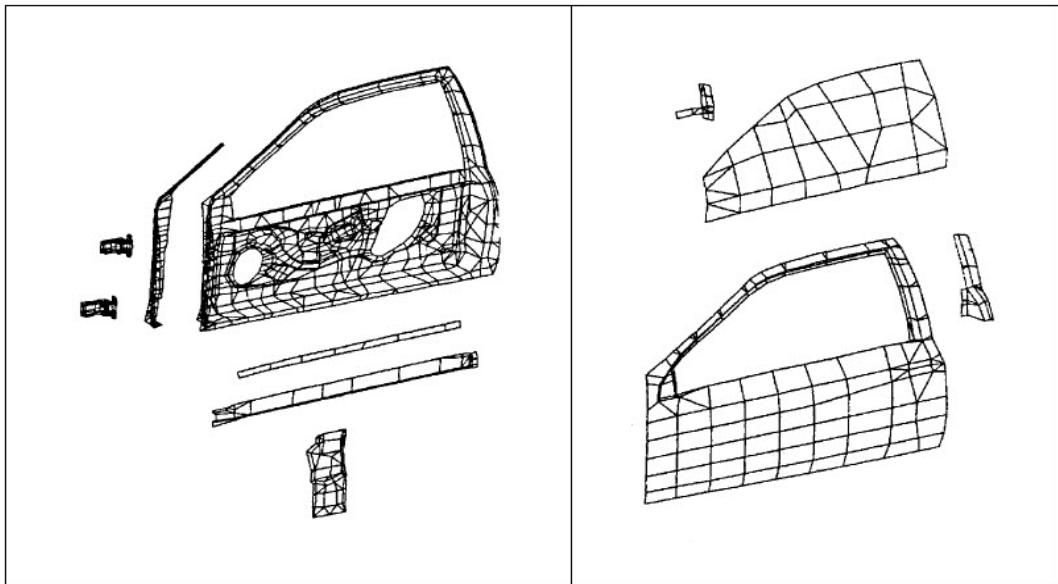


Figure 1-1 Superelements Used to Model a Car Door

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.

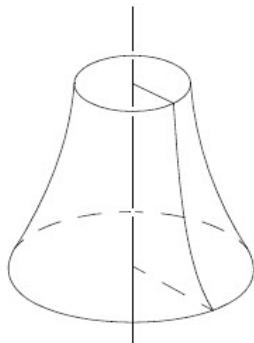
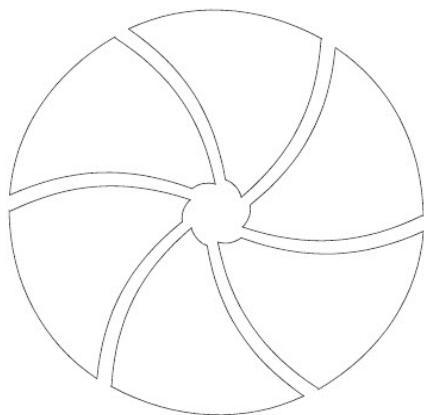


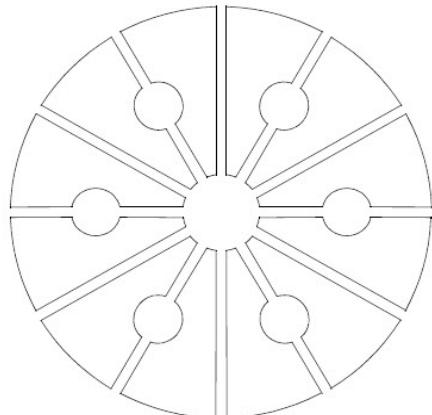
Figure 1-2 Axisymmetric Structure

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 [Figure 1-3](#). Cyclic symmetry analysis can be performed for statics, normal modes, buckling, and frequency response analysis.



Rotational Symmetry



Dihedral Symmetry

Figure 1-3 Cyclic Symmetry Analysis



Memory and I/O Performance

Computer system memory (i.e., RAM Memory) is a critical component of the performance of MSC Nastran. MSC Nastran was designed to run in a minimal memory environment, but nevertheless, insufficient memory can lead to suboptimal performance. If there is insufficient memory, MSC Nastran stores parts of the model during the simulation on the disk, which requires reading to and writing from the disk, referred to as I/O. This increased I/O cost can decrease the performance of MSC Nastran.

Default Memory Settings

MSC Nastran can use 4-byte integers by specifying `mode=i4` as an optional keyword or 8-byte integers by specifying `mode=i8` as an optional keyword. In MSC Nastran 2014.0, the default is 8-byte integers. For 8-byte integers, the maximum RAM memory that can be used is virtually unlimited, whereas for 4-byte integers, the maximum RAM memory available is 8 Gigabytes. A typical laptop can have anywhere from 2 Gigabytes to 16 Gigabytes of RAM memory with high performance computing systems reaching 256 Gigabytes of RAM memory and even up to 1 Terabyte of RAM memory.

MSC Nastran uses 50% of RAM memory for 8-byte integers or the minimum of 50% of RAM memory and 8 Gigabytes for 4-byte integers. You can use a Linux tool like *top* or a Windows tool like *Task Manager* to see the amount of memory actually used by MSC Nastran for the given simulation. MSC Nastran uses 50% of computer system memory to leave the other 50% to the system for I/O caching or for standard computer system processes. MSC Nastran can be used more than 50% of computer system memory various other methods that are described in the *MSC Nastran Quick Reference Guide*, referred to as the *memorymax* keyword.

I/O Performance

As stated in the previous section, when there is insufficient memory, MSC Nastran uses the disk to store parts of the model and perform I/O, to allow the user to increase the problem size that they can run on their computing system. A typical simulation can require a few hundred Megabytes of I/O, or in extreme cases, a few Terabytes of I/O. A typical Hard Disk Drive (HDD) is defined by the speed at which it spins with typical amounts being 5400 RPM (Revolutions per Minute) or up to 15,000 RPM.

The faster the disk spins, the faster the computing system can perform I/O operations. For example, a modest hard drive might read and write at 50 Megabytes per second, whereas a higher end hard drive can read and write at 100 Megabytes per second. The higher end hard drive can deliver 2x faster performance for the I/O part of MSC Nastran. A Solid State Device (SSD) is another popular disk storage medium that has higher read and write speeds. A typical SSD can read and write at 200 Megabytes per second or even as high as 500 Megabytes per second. For MSC Nastran, the higher end SSDs results in faster performance in the typical simulation.

Buffer Pool Usage

MSC Nastran uses a part of the memory it allocates for buffering I/O. As stated earlier, MSC Nastran is designed to run large problems that do not fit entirely in memory so I/O is performed to run these large



problems. By leaving some of memory for buffering, we can accelerate the performance of the I/O in MSC Nastran and, thus, the overall performance. MSC Nastran has been designed to use the optimal buffer pool amount by default so you need not change the buffer pool amounts except for large and complex problems. Further details of buffer pool usage are described in the *MSC Nastran Quick Reference Guide*.

Parallel Performance

Most modern desktop computers or laptop computers have multi-core CPUs with anywhere from 2 to 16 cores per CPU. However, the typical laptop computer generally has no more than 4 cores available for work. Each core can be utilized by MSC Nastran to speed-up simulations via the means of parallel processing. If the algorithm is perfect parallelizable, then 2 cores can reduce the wall time of the simulation by a factor of 2. In practice, this is rarely the case but instead 2 cores may reduce the wall time by anywhere from 25-40% depending on the model.

In MSC Nastran, there are two ways of obtaining parallelism to accelerate your simulation:

- Using Distributed Memory Processing (DMP).
- Using Shared Memory Processing (SMP).

These are explained in detail in the MSC Nastran Quick Reference Guide. In this section, we consider SMP parallelism since we assume that the reader of the *Getting Started Guide* is starting with MSC Nastran on a laptop. It is difficult to get good performance on a laptop with DMP because there are only 4 cores and generally slower hard drive speeds.

SMP parallelism utilizes parallelization of underlying math kernels to get parallelism. For example, internal to MSC Nastran sparse matrix calculations, there are a number of large dense matrix operations like matrix-matrix multiply. For this, it is possible to get very good acceleration using more than one core from the computing system. You can request more cores by specifying `smp=N` as an optional keyword, where N is a number less than the total number of cores on the computing system.



15

Example Problems

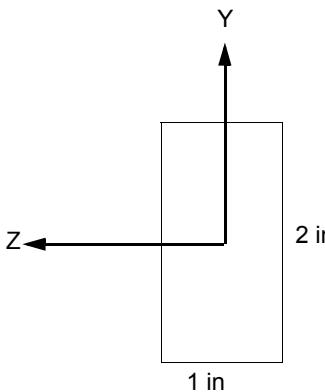
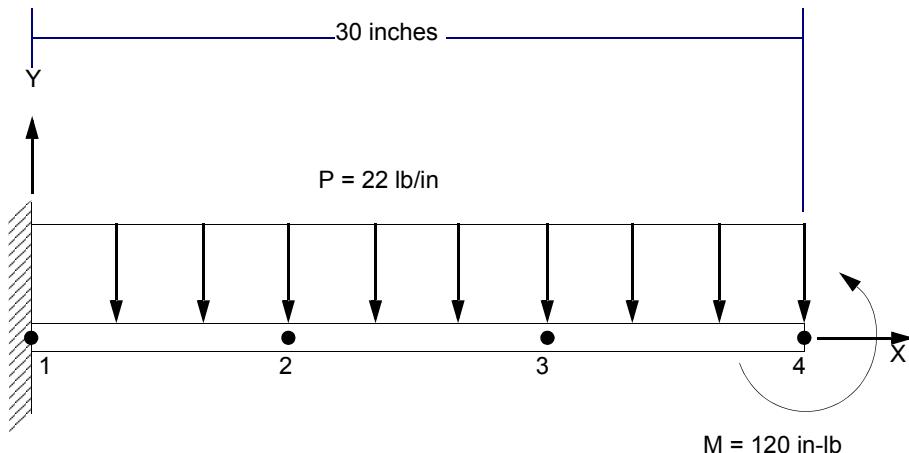
- Beam with a Distributed Load and a Moment 256
- Rectangular Plate with a Uniform Pressure Load 264
- Gear Tooth with Solid Elements 274

Beam with a Distributed Load and a Moment

This problem uses the same beam as the problem in Chapter 12: Performing Analysis (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 1-1.



$$\begin{aligned} I_1 &= I_z = 0.667 \text{ in}^4 \\ I_2 &= I_y = 0.1667 \text{ in}^4 \\ J &= 0.458 \text{ in}^4 \\ E &= 30 \times 10^6 \text{ lb/in}^2 \\ v &= 0.3 \end{aligned}$$

Figure 1-1 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, $P_1 = P_2 = 22.0 \text{ lb/in}$.

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 degrees of freedom (1, 2, 3, 4, 5, 6) 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](#). See `Install_dir/mscxxxx/gt13ex1.dat`.

Listing 13-1

```
$ CHAPTER 13 EXAMPLE 1
SOL 101
CEND
ECHO=BOTH
DISP=ALL
SPCF=ALL
LOAD=10
TITLE=EXAMPLE 1
SUBTITLE=CONTILEVER 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

```
$ CHAPTER 13 EXAMPLE 1
NASTRAN EXECUTIVE CONTROL ECHO

SOL 101
CEND
EXAMPLE 1
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT
CASE CONTROL ECHO
COMMAND
COUNT
1 ECHO=BOTH
2 DISP=ALL
3 SPCF=ALL
4 LOAD=10
5 TITLE=EXAMPLE 1
6 SUBTITLE=CONTILEVER BEAM
7 LABEL=DISTRIBUTED LOAD AND END MOMENT
8 $
9 BEGIN BULK
```

PAGE 3



Listing 13-2 (continued)

```

EXAMPLE 1 PAGE
4
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT
      I N P U T   B U L K   D A T A   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 ENTRY COUNT =      27

EXAMPLE 1 PAGE
5
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT
      S O R T E D   B U L K   D A T A   E C H O
      ENTRY      . 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 .
      COUNT
      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.   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
      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

      M O D E L   S U M M A R Y
      NUMBER OF GRID      POINTS =      4
      NUMBER OF CBAR      ELEMENTS =      3

EXAMPLE 1 PAGE
6
CONTILEVER BEAM
*** USER INFORMATION MESSAGE 7310 (VECPRN)
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
SUBCASE/ LOAD
DAREA ID  TYPE    T1      T2      T3      R1      R2      R3
      1  FX  0.000000E+00  ----  ----  0.000000E+00  0.000000E+00
      FX  0.000000E+00  ----  ----  0.000000E+00  0.000000E+00
      FY  0.000000E+00  -6.600000E+02  ----  0.000000E+00  0.000000E+00
      FZ  0.000000E+00  ----  0.000000E+00  0.000000E+00  0.000000E+00
      MX  0.000000E+00  ----  ----  0.000000E+00  ----  ----
      MY  0.000000E+00  ----  ----  0.000000E+00  ----  ----
      MZ  0.000000E+00  ----  ----  0.000000E+00  1.200000E+02
      TOTALS 0.000000E+00  -6.600000E+02  0.000000E+00  0.000000E+00  0.000000E+00  -9.780000E+03

```



Listing 13-2 (continued)

```

*** SYSTEM INFORMATION MESSAGE 41159 (DFMSA)
    THE DECOMPOSITION OF KLL      YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF     8.000000E+00
EXAMPLE 1                                         PAGE
7
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT
*** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO.          EPSILON           EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH
ASTERISKS
1                 1.5924801E-15        1.4106205E+01
8
EXAMPLE 1                                         PAGE
8
CONTILEVER BEAM

*** USER INFORMATION MESSAGE 7310 (VECPRN)
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
SUBCASE/ LOAD
DAREA ID   TYPE    T1      T2      T3      R1      R2      R3
1   FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
    FY  ----  6.600000E+02  ----  0.000000E+00  ----  0.000000E+00
    FZ  ----  ----  0.000000E+00  0.000000E+00  0.000000E+00  ----
    MX  ----  ----  ----  0.000000E+00  ----  ----
    MY  ----  ----  ----  ----  0.000000E+00  ----
    MZ  ----  ----  ----  ----  ----  9.780000E+03
    TOTALS 0.000000E+00  6.600000E+02  0.000000E+00  0.000000E+00  0.000000E+00  9.780000E+03
EXAMPLE 1                                         PAGE
9
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT

DISPLACEMENT VECTOR
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 1                                         PAGE
10
CONTILEVER BEAM
DISTRIBUTED LOAD AND END MOMENT

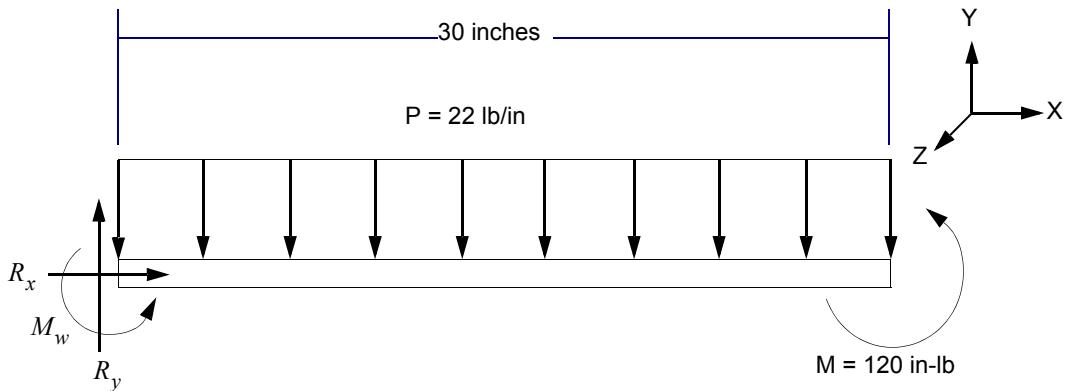
FORCES OF SINGLE-POINT CONSTRAINT
POINT ID.  TYPE    T1      T2      T3      R1      R2      R3
1   G     0.0  6.600000E+02      0.0      0.0      0.0  9.780000E+03

```

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 7 of the output. Its value of 1.5924801E-15 is indeed very small, showing no evidence of numerical difficulties. Finally, we review the reaction forces (forces of single point constraint) at the wall. A free body diagram of the structure is used to solve reaction forces as follows:





Solving for the reactions at the wall, we obtain:

Forces in x: $\sum F_x = 0 = R_x$

$$R_x = 0$$

Forces in y: $\sum F_y = 0 = R_y - (22 \text{ lb/in})(30 \text{ in})$

$$R_y = 660 \text{ lbs}$$

Moment at wall: $\sum M_{wall} = M_w + 120 \text{ in-lb} - (660 \text{ lbs})(15 \text{ in})$

$$M_w = 9780 \text{ in-lb}$$

The SPC forces are listed on page 10 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 sure that the loads were applied correctly, and at least the static equilibrium of the problem makes sense.

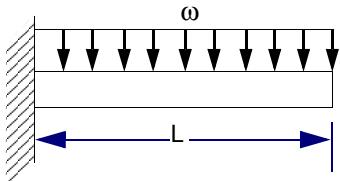
The displacement results are shown on page 9 of the .f06 file. All displacements at the wall (GRID 1) are exactly zero. The free end deflection in the y direction (T2 of GRID 4) is -1.086207E-1 in.

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.

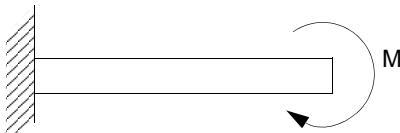


Comparison with Theory

The theoretical 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 MSC Nastran](#)), and is by no means typical of real world problems.

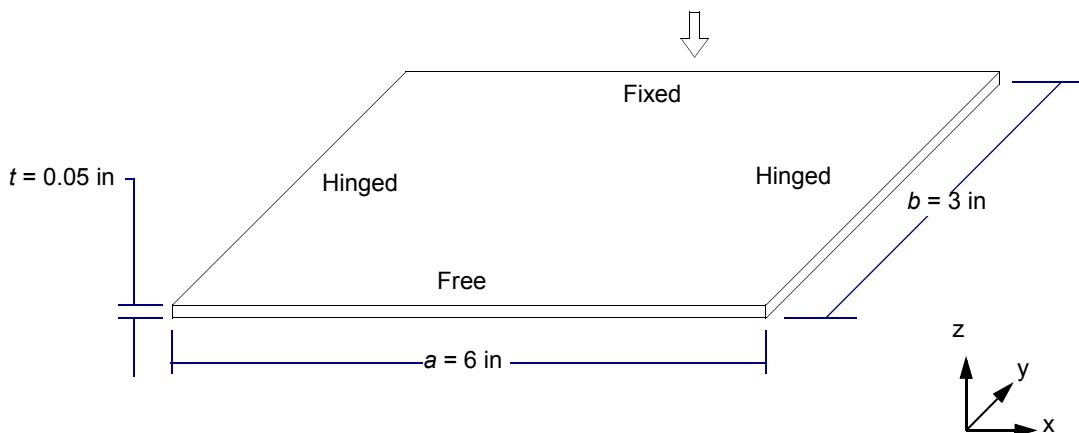


Rectangular Plate with a Uniform Pressure Load

Problem Statement

Create an MSC Nastran model to analyze the thin rectangular plate shown in [Figure 1-2](#). The plate is subject to a uniform pressure load of 0.25 lb/in^2 in the $-z$ direction. Find the maximum deflection of the plate.

$$P = \text{uniform pressure} = 0.25 \text{ lb/in}^2$$



$$E = 30 \times 10^6 \text{ lb/in}^2$$

$$\nu = 0.3$$

[Figure 1-2](#) 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 can do the job. In our case, doing the job means good displacement accuracy. The model shown in [Figure 1-3](#) 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.



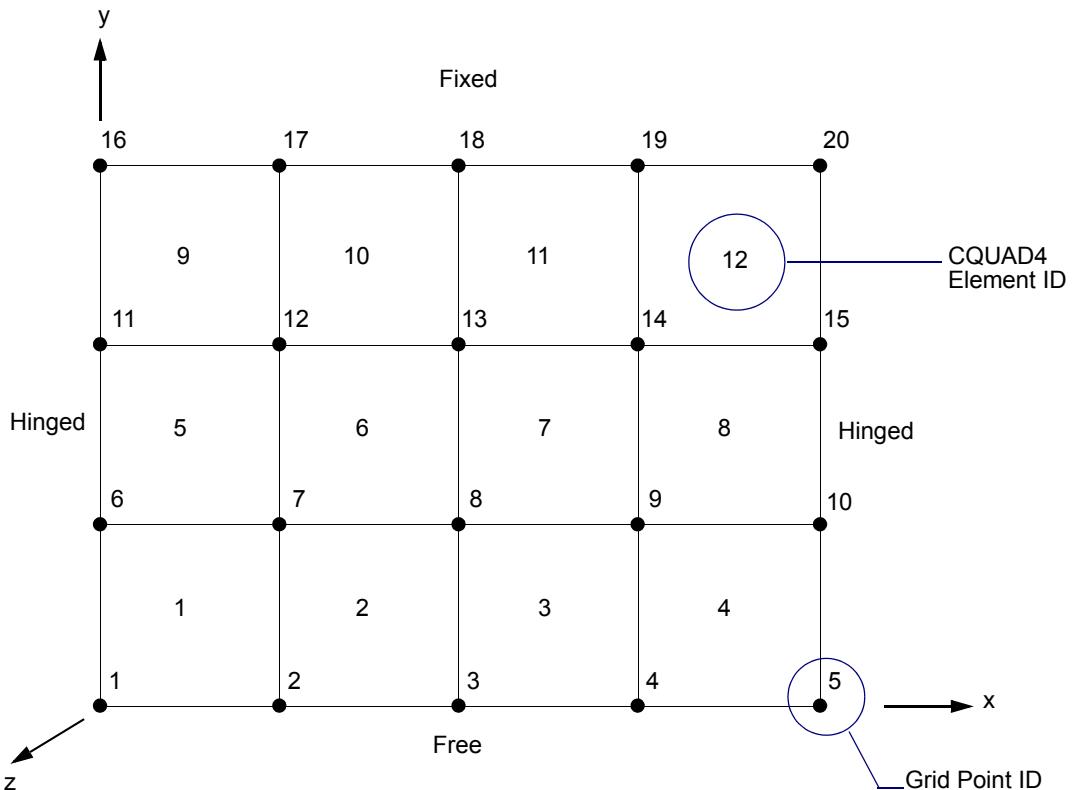


Figure 1-3 Plate Finite Element Model

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 1-4.



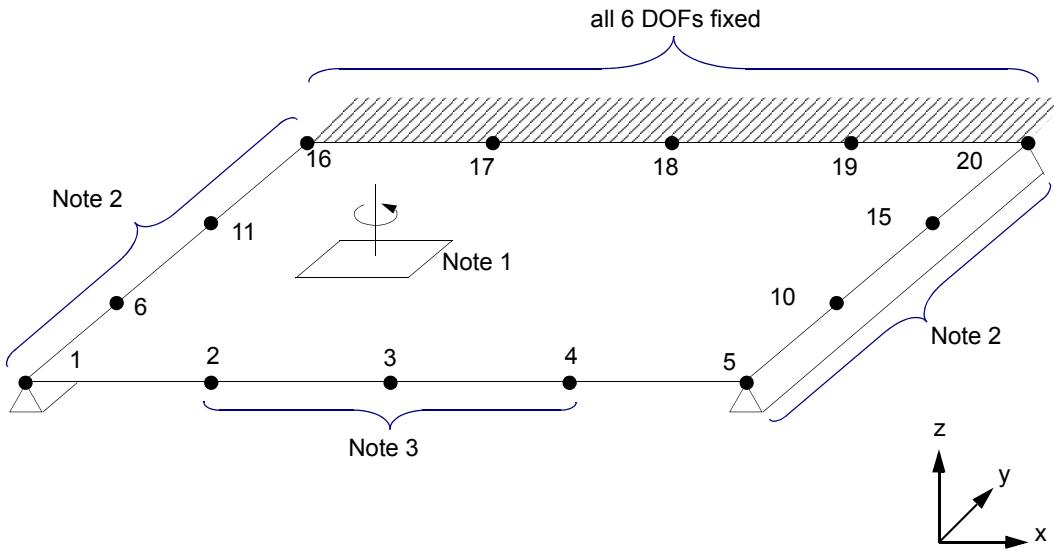


Figure 1-4 Constraints on the Plate Structure

Note:

- The out-of-plane rotational 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.
- 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 (degree of freedom 5)
 - Displacements Not Allowed: Rotation about x-axis (degree of freedom 4)
 - Translation in x, y, or z (degrees of freedom 1, 2, 3)
- The noncorner 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: Some constraints are redundantly specified. For example, GRID 17 is constrained in all 6 degrees of freedom with the fixed edge SPC1, and again in degree of freedom 6 with the out-of-plane rotational constraint. This is acceptable and allows for easy reference to the book.

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](#). See `Install_dir/mscxxxx/gt13ex2.dat`.

Listing 13-3

```
$ CHAPTER 13 EXAMPLE 2
SOL 101
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

```

$ CHAPTER 13 EXAMPLE 2
N A S T R A N   E X E C U T I V E   C O N T R O L   E C H O

SOL 101
CEND
PLATE EXAMPLE
FIXED-HINGED-HINGED-FREE
UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
C A S E   C O N T R O L   E C H O
COMMAND
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
PAGE      3

PLATE EXAMPLE
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   E C H O
.   1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 ..
$      DEFINE GRID POINTS
GRID,1,,0,0,0,0.
GRID,2,,1.5,0,0,0.
GRID,3,,3.0,0,0,0.
GRID,4,,4.5,0,0,0.
GRID,5,,6.0,0,0,0.
GRID,6,,0,1,0,0.
GRID,7,,1.5,1,0,0.
GRID,8,,3.0,1,0,0.
GRID,9,,4.5,1,0,0.
GRID,10,,6.0,1,0,0.
GRID,11,,0,2,0,0.
GRID,12,,1.5,2,0,0.
GRID,13,,3.0,2,0,0.
GRID,14,,4.5,2,0,0.
GRID,15,,6.0,2,0,0.
GRID,16,,0,3,0,0.
GRID,17,,1.5,3,0,0.
GRID,18,,3.0,3,0,0.
GRID,19,,4.5,3,0,0.
GRID,20,,6.0,3,0,0.
$      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 ENTRY COUNT =      51

```



270 | MSC Nastran Getting Started Guide
Rectangular Plate with a Uniform Pressure Load

Listing 13-4 (continued)

PLATE EXAMPLE FIXED-HINGED-HINGED-FREE UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)										PAGE	5				
ENTRY COUNT	.	1 ..	2 ..	3 ..	4 ..	5 ..	6 ..	7 ..	8 ..	9 ..	10 ..	S O R T E D	B U L K	D A T A	E C H O
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												



Listing 13-4 (continued)

```

MODEL SUMMARY
    NUMBER OF GRID     POINTS =      20
    NUMBER OF CQUAD4   ELEMENTS =     12

PLATE EXAMPLE
    FIXED-HINGED-HINGED-FREE
    UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)                                PAGE 6
PLATE EXAMPLE
    FIXED-HINGED-HINGED-FREE                                                       PAGE 7

*** USER INFORMATION MESSAGE 7310 (VECPRN)
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

SUBCASE/ LOAD
DAREA ID  TYPE    T1      T2      T3      R1      R2      R3
          FX  0.000000E+00  ----  ----  0.000000E+00  0.000000E+00
          FY  ----  0.000000E+00  ----  0.000000E+00  ----  0.000000E+00
          FZ  ----  ----  -4.500000E+00 -6.750000E+00  1.350000E+01  ----
          MX  ----  ----  ----  0.000000E+00  ----  ----
          MY  ----  ----  ----  ----  0.000000E+00  ----
          MZ  ----  ----  ----  ----  ----  0.000000E+00
TOTALS  0.000000E+00  0.000000E+00 -4.500000E+00 -6.750000E+00  1.350000E+01  0.000000E+00

*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)
THE DECOMPOSITION OF KLL      YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF 6.603744E+01                                PAGE 8
PLATE EXAMPLE
    FIXED-HINGED-HINGED-FREE
    UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)
*** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO.      EPSILON           EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERisks
          1      8.5787226E-15      2.1618363E-03
PLATE EXAMPLE
    FIXED-HINGED-HINGED-FREE                                                       PAGE 9

```



Listing 13-4 (continued)

```
*** USER INFORMATION MESSAGE 7310 (VCPRN)
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
```

SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	0.000000E+00	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	4.500000E+00	8.752237E+00	-1.350000E+01	----
	MX	----	----	----	-2.002237E+00	----	----
	MY	----	----	----	----	-4.662937E-15	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	4.500000E+00	6.750000E+00	-1.350000E+01	0.000000E+00

PLATE EXAMPLE
FIXED-HINGED-HINGED-FREE
UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)

PAGE 10

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	2.015773E-03	0.0
2	G	0.0	0.0	-2.634148E-03	1.128535E-03	1.379105E-03	0.0
3	G	0.0	0.0	-3.687613E-03	1.587460E-03	-1.003887E-17	0.0
4	G	0.0	0.0	-2.634148E-03	1.128535E-03	-1.379105E-03	0.0
5	G	0.0	0.0	0.0	0.0	-2.015773E-03	0.0
6	G	0.0	0.0	0.0	0.0	1.191799E-03	0.0
7	G	0.0	0.0	-1.548584E-03	1.039289E-03	7.968513E-04	0.0
8	G	0.0	0.0	-2.154623E-03	1.473592E-03	3.607174E-18	0.0
9	G	0.0	0.0	-1.548584E-03	1.039289E-03	-7.968513E-04	0.0
10	G	0.0	0.0	0.0	0.0	-1.191799E-03	0.0
11	G	0.0	0.0	0.0	0.0	4.142349E-04	0.0
12	G	0.0	0.0	-5.330283E-04	9.344256E-04	2.678075E-04	0.0
13	G	0.0	0.0	-7.339914E-04	1.285929E-03	8.922717E-20	0.0
14	G	0.0	0.0	-5.330283E-04	9.344256E-04	-2.678075E-04	0.0
15	G	0.0	0.0	0.0	0.0	-4.142349E-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
FIXED-HINGED-HINGED-FREE
UNIFORM LATERAL PRESSURE LOAD (0.25 LB/IN**2)

PAGE 11

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	3.573215E-01	-9.876604E-02	0.0	0.0
5	G	0.0	0.0	3.573215E-01	-9.876604E-02	0.0	0.0
6	G	0.0	0.0	4.665502E-01	-4.985499E-02	0.0	0.0
10	G	0.0	0.0	4.665502E-01	-4.985499E-02	0.0	0.0
11	G	0.0	0.0	3.688164E-01	2.798499E-01	0.0	0.0
15	G	0.0	0.0	3.688164E-01	2.798499E-01	0.0	0.0
16	G	0.0	0.0	-4.810770E-01	9.887268E-02	2.685457E-02	0.0
17	G	0.0	0.0	9.978767E-01	-7.305090E-01	-5.310716E-03	0.0
18	G	0.0	0.0	1.081024E+00	-1.001422E+00	-1.221245E-15	0.0
19	G	0.0	0.0	9.978767E-01	-7.305090E-01	5.310716E-03	0.0
20	G	0.0	0.0	-4.810770E-01	9.887268E-02	-2.685457E-02	0.0

Reviewing the Results

The value of epsilon, listed on page 8 of the output, is small, indicating a numerically well-behaved problem. A plot of the deformed plate is shown in [Figure 1-5](#). As expected, the maximum displacement (-3.687613E-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 9 of the output. Note that the SPCFORCE is positive, and the applied load is in the negative z (-T3) direction.



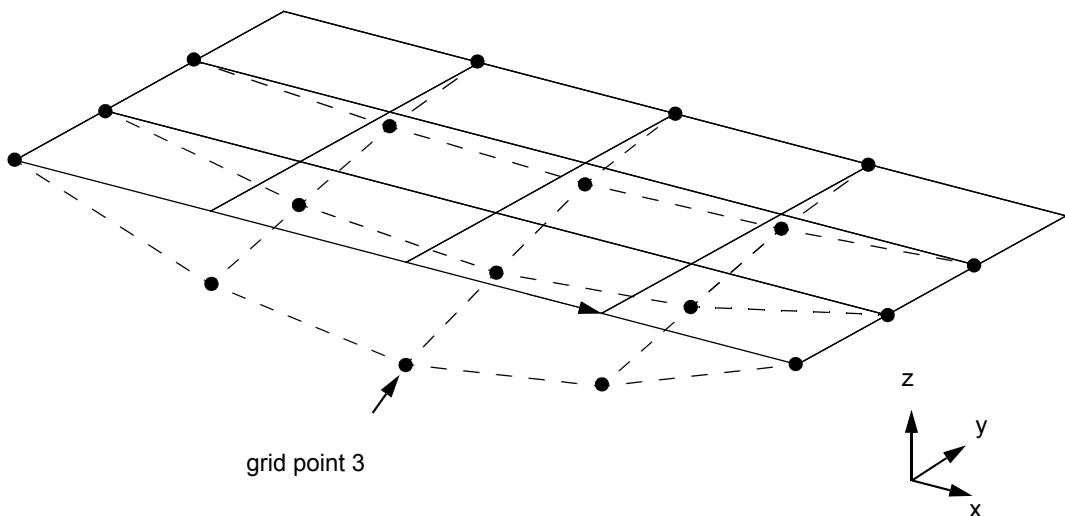


Figure 1-5 Deformed Shape

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:

$$W_{max} = (0.0582 q b^4 / D) \quad \left(\text{for } b/a = \frac{1}{2} \right)$$

Where

$$q = \text{lateral pressure} = 0.25 \text{ lb/in}^2$$

$$D = \frac{Et^3}{12(1 - v^2)} = \frac{(30 \times 10^6 \text{ lb/in}^2)(0.5 \text{ in})^3}{12(1 - 0.3^2)} = 343.407$$

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:



$$W_{max} = 3.687013E-3 \text{ in}$$

The MSC Nastran result (which includes transverse shear) is 7.4% 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 7 of the PSHELL entry) gives a deflection of

$$W_{max} (\text{no shear}) = 3.673454E-3 \text{ in}$$

Thus, for this thin plate, adding shear deflection results in less than half a percent difference in the total deflection.

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 1-6](#). 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:

$$E = 2.0 \times 10^5 \text{ MPa}$$

$$\nu = 0.3$$

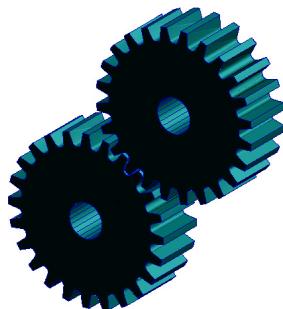


Figure 1-6 Spur Gears

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:



$$\sigma_{von} = \frac{1}{\sqrt{2}} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{yz})^2 + 6(\tau_{zx})^2 + 6(\tau_{xy})^2]^{1/2} \quad (15-1)$$

The Finite Element Model

A single gear tooth is modeled using two CHEXA solid elements with midside grid points as shown in [Figure 1-7](#). Midside grid points are useful when the shape of a structure is complex or when bending effects are important.

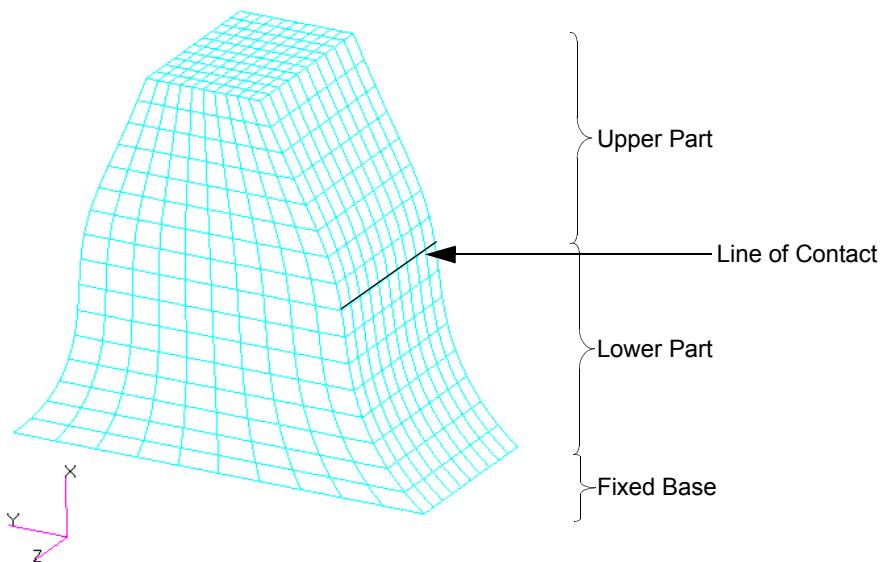


Figure 1-7 Finite Element Model of a Single Gear Tooth Meshed with 20-noded CHEXA Elements

Applying the Loads

Subcase 1 represents aligned gear teeth and uses the distributed load shown in [Figure 1-8](#). The total applied load is given by:

$$\begin{aligned} \text{Total Load} &= \text{Distributed Load} \cdot \text{Width of Gear Tooth} \\ &= 600 \times 10 \text{ mm} = 6000 \text{ N} \end{aligned}$$

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 150 N two corner grids of each element and 300 N on the nineteen intermediate grids. A load set identification number of 41 (arbitrarily chosen) is given to the FORCE Bulk Data entries of subcase 1.



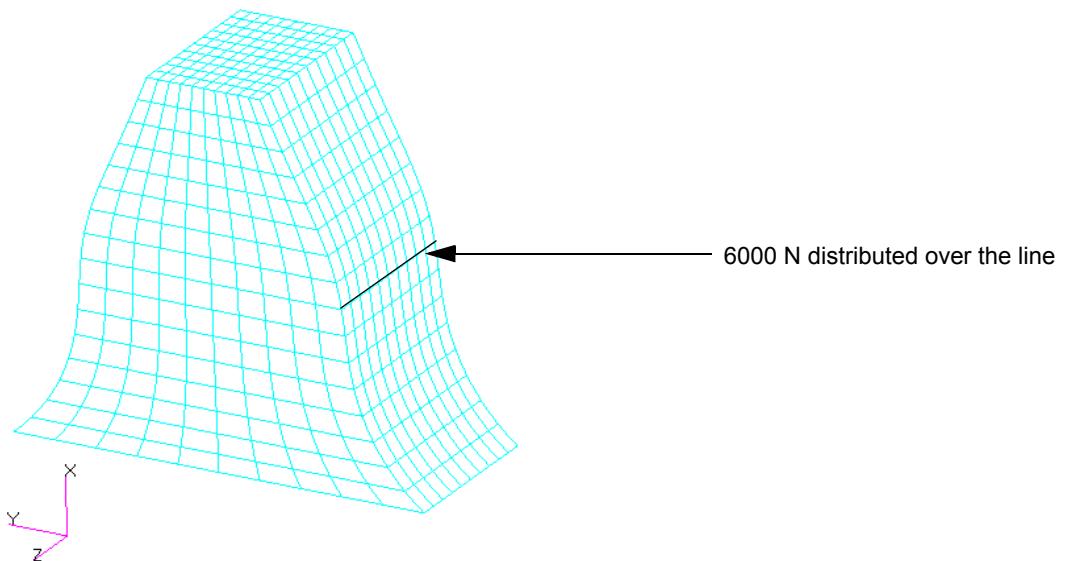


Figure 1-8 Subcase 1 - Gears in Alignment (Distributed Load)

Subcase 2 represents misaligned gear teeth and uses a single concentrated force of 6000 N as shown in [Figure 1-9](#). 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.

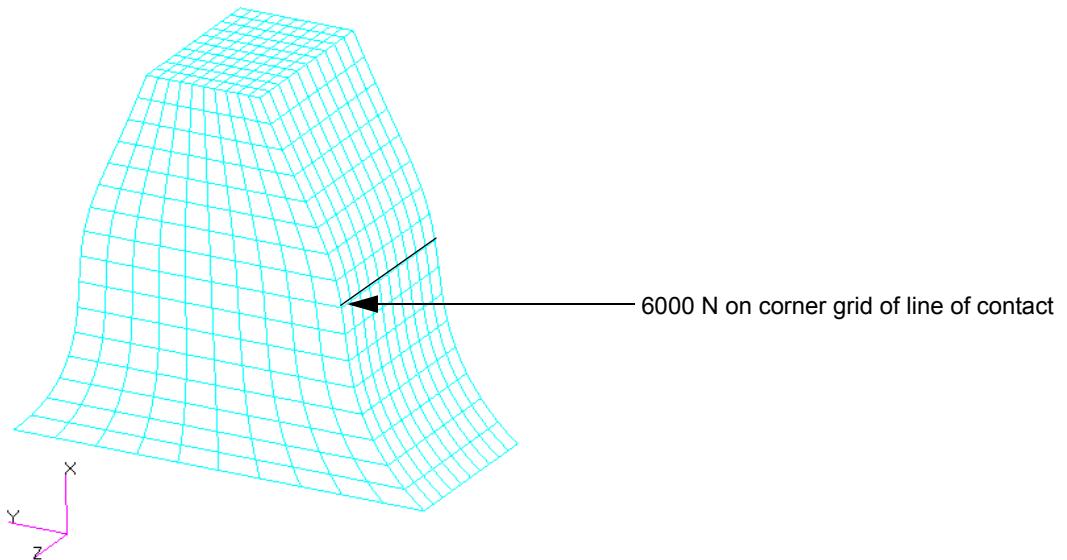


Figure 1-9 Subcase 2 - Gears Misaligned (Concentrated Load)



Applying the Constraints

The base of the tooth is assumed to be fixed as shown in [Figure 1-7](#). Consequently, grid points at the bottom face of the tooth are constrained to zero displacement in their translational degrees of freedom (1, 2, and 3). Recall that solid elements have only translational degrees of freedom and no rotational degrees of freedom. Since each grid point starts out with all six degrees of freedom, the remaining “unattached” rotational degrees of freedom must be constrained to prevent numerical singularities. Thus, all grid points in the model (1 through 7733) are constrained in degrees of freedom 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.

The Input File

The complete input file is shown in [Listing 13-5](#). See `Install_dir/mscxxxx/gt13ex3.dat`.

Listing 13-5 Gear Tooth Input File

```
$ CHAPTER 13 EXAMPLE 3
SOL 101
CEND
TITLE = GEAR TOOTH EXAMPLE
ECHO = NONE
SUBCASE 1
    TITLE=GEAR TOOTH EXAMPLE
    SUBTITLE=GEAR TOOTH UNDER 600 N/MM LINE LOAD
    SPC = 2
    LOAD = 44
    STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
SUBCASE 2
    TITLE=GEAR TOOTH EXAMPLE
    SUBTITLE=GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
    SPC = 2
    LOAD = 46
    STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM POST      0
PARAM PRTMAXIM YES
$ Elements and Element Properties for region : psolid.10
PSOLID 10      20      0
$ Pset: "psolid.10" will be imported as: "psolid.10"
CHEXA   1        10      52      50      7733      2        540      533
        535      538      51      33      1        34        541      534
        536      539      544      537      543      542
CHEXA   2        10      54      52      2        4        547      540
        538      545      53      34      3        35        548      541
        539      546      551      542      550      549
CHEXA   3        10      56      54      4        6        554      547
.
.
.
```



```

.
$ Referenced Material Records
$ Material Record : mat1.20
$ Description of Material :
MAT1      20      200000. 76923.1 .3
$ Nodes of the Entire Model
GRID      1          87.0372 12.1025 5.
GRID      2          87.6121 11.5425 5.
GRID      3          88.2241 11.0206 5.
GRID      4          88.8675 10.5418 5.
.
.
.
$ Loads for Load Case : _GEAR_TOOTH_UNDER_600_N/MM_LINE_LOAD.SC1
SPCADD   2          30
LOAD     44      1.      1.      41
$ Displacement Constraints of Load Set : spc1.30
SPC1    30      123      33      50      83      100      133      150
      183      200      233      250      283      300      333      350
      383      400      433      450      483      500      533      534
      535      536      537      650      651      652      717      718
.
.
.
6879      6880      6881      6946      6947      6948      7013      7014
7015      7016      7017      7130      7131      7132      7197      7198
7199      7264      7265      7266      7331      7332      7333      7398
7399      7400      7465      7466      7467      7532      7533      7534
7599      7600      7601      7666      7667      7668      7733
SPC1, 30, 456, 1, THRU, 7733
$ Loads for Load Case :
_GEAR_TOOTH_UNDER_6000_N_CONCENTRATED_LOAD.SC2
LOAD     46      1.      1.      42
$ Nodal Forces of Load Set : force.41
FORCE    41      518      0      150.0      0.      1.      0.
FORCE    41     1221      0      300.0      0.      1.      0.
FORCE    41     1222      0      300.0      0.      1.      0.
FORCE    41     1941      0      300.0      0.      1.      0.
FORCE    41     1942      0      300.0      0.      1.      0.
FORCE    41     2661      0      300.0      0.      1.      0.
FORCE    41     2662      0      300.0      0.      1.      0.
FORCE    41     3381      0      300.0      0.      1.      0.
FORCE    41     3382      0      300.0      0.      1.      0.
FORCE    41     4101      0      300.0      0.      1.      0.
FORCE    41     4102      0      300.0      0.      1.      0.
FORCE    41     4821      0      300.0      0.      1.      0.
FORCE    41     4822      0      300.0      0.      1.      0.
FORCE    41     5541      0      300.0      0.      1.      0.
FORCE    41     5542      0      300.0      0.      1.      0.
FORCE    41     6261      0      300.0      0.      1.      0.
FORCE    41     6262      0      300.0      0.      1.      0.
FORCE    41     6981      0      300.0      0.      1.      0.
FORCE    41     6982      0      300.0      0.      1.      0.

```



```
FORCE    41      7702      0      300.0      0.      1.      0.  
FORCE    41      7701      0      150.0      0.      1.      0.  
$ Nodal Forces of Load Set : force.42  
FORCE    42      518       0      6000.      0.      1.      0.  
$ Referenced Coordinate Frames  
ENDDATA
```

MSC Nastran Results

The .f06 Results File

The MSC Nastran results are shown in [Listing 13-6](#).

Listing 13-6 Gear Tooth Results



280 | MSC Nastran Getting Started Guide
Gear Tooth with Solid Elements

```

SOL 101
CEND
GEAR TOOTH EXAMPLE

PAGE 3

COMMAND
COUNT
1   TITLE = GEAR TOOTH EXAMPLE
2   ECHO = NONE
3   SUBCASE 1
4       TITLE=GEAR TOOTH EXAMPLE
5       SUBTITLE=GEAR TOOTH UNDER 600 N/MM LINE LOAD
6       SPC = 2
7       LOAD = 44
8       STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
9   SUBCASE 2
10      TITLE=GEAR TOOTH EXAMPLE
11      SUBTITLE=GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD
12      SPC = 2
13      LOAD = 46
14      STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
15   BEGIN BULK

GEAR TOOTH EXAMPLE

PAGE 4

M O D E L   S U M M A R Y
ENTRY NAME   NUMBER OF ENTRIES      BULK = 0
-----
CHEXA        1600
FORCE         12
GRID          7733
LOAD           2
MAT1           1
PARAM          2
PSOLID          1
SPC1            2
SPCADD          1

>>> IFF OPERATIONS COMPLETE <<<

```

```

GEAR TOOTH EXAMPLE

PAGE 5

*** USER INFORMATION MESSAGE 7310 (VECPRN)
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

SUBCASE/ LOAD
DAREA ID  TYPE    T1      T2      T3      R1      R2      R3
1   FX     0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY     ----  6.000000E+03  ----  0.000000E+00  ----  5.977536E+05
      FZ     ----  ----  0.000000E+00  0.000000E+00  0.000000E+00  ----
      MX     ----  ----  ----  0.000000E+00  ----  ----
      MY     ----  ----  ----  ----  0.000000E+00  ----
      MZ     ----  ----  ----  ----  ----  0.000000E+00
      TOTALS 0.000000E+00 6.000000E+03 0.000000E+00 0.000000E+00 0.000000E+00 5.977536E+05
2   FX     0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY     ----  6.000000E+03  ---- -3.000000E+04  ----  5.977536E+05
      FZ     ----  ----  0.000000E+00  0.000000E+00  0.000000E+00  ----
      MX     ----  ----  ----  0.000000E+00  ----  ----
      MY     ----  ----  ----  ----  0.000000E+00  ----
      MZ     ----  ----  ----  ----  ----  0.000000E+00
      TOTALS 0.000000E+00 6.000000E+03 0.000000E+00 -3.000000E+04 0.000000E+00 5.977536E+05

```



GEAR TOOTH EXAMPLE

PAGE 6

SUBCASE 1

*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)
 THE DECOMPOSITION OF KLL YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF 1.379802E+01
 *** USER INFORMATION MESSAGE 5293 (SSG3A)
 FOR DATA BLOCK KLL

LOAD SEQ. NO.	EPSILON	EXTERNAL WORK	EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1	-2.7894550E-15	4.6314602E+01	
2	2.6902868E-15	5.3673444E+02	

GEAR TOOTH EXAMPLE

PAGE 7

MAXIMUM DISPLACEMENTS

SUBCASE/

DAREA ID	T1	T2	T3	R1	R2	R3
1	4.7030803E-03	1.6682748E-02	2.7827199E-03	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	9.2060585E-03	1.7891149E-01	8.0067538E-02	0.0000000E+00	0.0000000E+00	0.0000000E+00

GEAR TOOTH EXAMPLE

PAGE 8

MAXIMUM APPLIED LOADS

SUBCASE/

DAREA ID	T1	T2	T3	R1	R2	R3
1	0.0000000E+00	3.0000000E+02	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	0.0000000E+00	6.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00



GEAR TOOTH EXAMPLE										PAGE	319
GEAR TOOTH UNDER 600 N/MM LINE LOAD										SUBCASE 1	
ELEMENT-ID	STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)									VON MISES	
	CORNER	-----CENTER AND CORNER POINT STRESSES-----				DIR.	COSINES		MEAN		
	GRID-ID	NORMAL	SHEAR	PRINCIPAL		-A-	-B-	-C-	PRESSURE		
1	OGRID CS 20 GP										
	CENTER	X -5.724210E+01	XY 3.101264E+01	A -1.884504E+00	LX 0.47	0.86	0.20		2.946234E+01	7.111794E+01	
		Y -2.278653E+01	YZ -1.244307E+00	B -7.665063E+01	LY 0.67	-0.50	0.55				
		Z -8.358395E+00	ZX 9.579105E+00	C -9.851895E+00	LZ 0.57	-0.13	-0.81				
	52	X -6.667654E+01	XY 2.772095E+01	A 8.977235E+00	LX -0.28	0.94	-0.21		2.051783E+01	8.526604E+01	
		Y -2.476887E+00	YZ 9.117184E-01	B -7.734818E+01	LY -0.63	-0.35	0.70				
		Z 7.599939E+00	ZX -5.549281E+00	C 6.817450E+00	LZ 0.73	0.06	-0.69				
	50	X -1.133700E+02	XY 3.478274E+01	A -2.388185E+01	LX 0.47	0.88	0.00		7.039645E+01	1.055047E+02	
		Y -4.908337E+01	YZ -9.416055E-01	B -1.393452E+02	LY 0.63	-0.34	0.70				
		Z -4.873601E+01	ZX 3.338587E+01	C -4.796230E+01	LZ 0.61	-0.33	-0.72				
	7733	X -2.087174E+01	XY 2.728729E+01	A 1.440324E+01	LX 0.63	0.78	-0.01		1.276217E+01	5.106662E+01	
		Y -8.579416E+00	YZ -4.163059E-01	B -4.411555E+01	LY 0.74	-0.60	0.30				
		Z -8.835346E+00	ZX 8.669305E+00	C -8.574189E+00	LZ 0.22	-0.20	-0.95				
	2	X -4.077037E+01	XY 3.599668E+01	A 4.222226E+00	LX -0.40	0.77	-0.50		2.157796E+01	7.358482E+01	
		Y -2.700311E+01	YZ -2.491610E+00	B -7.061212E+01	LY -0.52	-0.64	0.57				
		Z 3.039619E+00	ZX 1.017926E+00	C 1.656029E+00	LZ 0.75	0.03	-0.66				
	540	X -7.798841E+01	XY 2.979019E+01	A -2.414664E+00	LX 0.39	0.92	0.05		3.671485E+01	8.398258E+01	
		Y -1.770375E+01	YZ 5.063885E-01	B -9.218788E+01	LY 0.79	-0.37	0.50				
		Z -1.445239E+01	ZX 1.353924E+01	C -1.554200E+01	LZ 0.48	-0.16	-0.86				
	533	X -5.563333E+01	XY 3.267775E+01	A -1.017957E+00	LX 0.54	0.84	0.00		3.441689E+01	6.898850E+01	
		Y -2.379025E+01	YZ -1.265934E+01	B -7.849982E+01	LY 0.78	-0.50	0.38				
		Z -2.382708E+01	ZX 1.345424E+01	C -2.373288E+01	LZ 0.32	-0.21	-0.93				
	535	X -6.114676E+00	XY 2.027994E+01	A 1.650333E+01	LX 0.67	0.74	0.02		3.464958E+00	3.549018E+01	
		Y -1.881383E+00	YZ 3.610294E-01	B -2.444008E+01	LY 0.74	-0.67	0.09				
		Z -2.398818E+00	ZX 1.764150E+00	C -2.458120E+00	LZ 0.08	-0.05	-1.00				
	538	X -3.713385E+01	XY 3.393248E+01	A 8.277396E-01	LX 0.68	0.73	-0.04		2.511808E+01	6.803168E+01	
		Y -2.985895E+01	YZ -7.757474E+00	B -7.030714E+01	LY 0.71	-0.65	0.26				
		Z -8.361427E+00	ZX 1.036139E+01	C -5.874828E+00	LZ 0.17	-0.20	-0.96				
2	OGRID CS 20 GP										
	CENTER	X -6.727074E+01	XY 3.283265E+01	A -1.428954E+00	LX 0.07	0.89	-0.46		3.005879E+01	8.172408E+01	
		Y -2.131677E+01	YZ -1.478333E+00	B -8.451751E+01	LY 0.03	-0.46	0.89				
		Z -1.588852E+00	ZX 3.203474E+00	C -4.229898E+00	LZ 1.00	-0.04	0.06				
	54	X -6.241193E+01	XY 2.159242E+01	A 7.766453E-01	LX -0.13	0.94	-0.30		2.341641E+01	6.989288E+01	
		Y -8.357329E+00	YZ -1.611489E+01	B -7.000050E+01	LY -0.34	-0.33	-0.88				
		Z 5.200263E+01	ZX -1.374627E+00	C -1.025840E+00	LZ 0.93	0.02	-0.36				



153 OGRID CS 20 GP

AUGUST 21, 2014 MSC Nastran 11/27/13 PAGE 436

SUBCASE 1

STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)									
ELEMENT-ID	GRID-ID	CORNERS -----CENTER AND CORNER POINT STRESSES-----			DIR. COSINES			PRESSURE	VON MISES
		NORMAL	SHEAR	PRINCIPAL	-A-	-B-	-C-		
153	CENTER	X 1.594761E+01 XY 1.076138E+02 A 9.324733E+01 LX 0.81-0.57 0.14 7.884502E+00 2.094569E+02	Y -6.708150E+01 YZ -1.793465E+00 B -1.418370E+02 LY 0.54 0.82 0.19	Z 2.748038E+01 ZX 1.945868E+01 C 2.493618E+01 LZ 0.22 0.07-0.97					
518		X -2.512840E+02 XY 4.449283E+02 A 1.481110E+02 LX 0.57-0.45-0.69 3.780110E+02 1.148725E+03	Y -8.888388E+02 YZ 1.399811E+02 B -1.122998E+03 LY 0.35 0.89-0.29	Z 6.089902E+00 ZX 1.001563E+02 C -1.591456E+02 LZ 0.74-0.07 0.66					
516		X 1.119002E+01 XY -8.629256E+01 A 1.355560E+02 LX-0.57 0.54 0.62 -1.218142E+01 1.866858E+02	Y 7.020064E+01 YZ -2.916058E+01 B -6.369135E+01 LY 0.81 0.50 0.31	Z -4.484641E+01 ZX 3.278870E+00 C -3.532038E+01 LZ-0.14 0.68-0.72					
466		X -1.392379E+02 XY 1.674396E+02 A 2.175180E+01 LX 0.72-0.69-0.02 1.166409E+02 3.058717E+02	Y -1.538442E+02 YZ 8.644416E+00 B -3.155347E+02 LY 0.68 0.72-0.12	Z -5.684048E+01 ZX -1.843089E+01 C -5.613983E+01 LZ-0.09-0.07-0.99					
468		X 1.465074E+01 XY 3.824397E+01 A 1.875545E+01 LX 0.99-0.07-0.10 1.982057E+02 4.726551E+02	Y -5.008588E+02 YZ -1.979684E+01 B -5.045840E+02 LY 0.08 1.00 0.04	Z -1.0804089E+02 ZX -1.131243E+01 C -1.087885E+02 LZ-0.10 0.05-0.99					
1221		X -3.475622E+02 XY 3.768913E+02 A -4.109996E+01 LX-0.42-0.45-0.79 4.386408E+02 9.760743E+02	Y -8.479340E+02 YZ -1.909999E+02 B -1.083555E+03 LY-0.39 0.87-0.29	Z -1.204262E+02 ZX 2.252453E+01 C -1.912670E+02 LZ 0.82 0.18-0.54					
1217		X 1.034380E+02 XY -1.178236E+02 A 2.070459E+02 LX 0.75 0.55-0.37 -7.272825E+01 2.533249E+02	Y 7.298269E+01 YZ 6.218538E+01 B -8.308864E+01 LY-0.66 0.63-0.40	Z 4.176406E+01 ZX 5.187270E+01 C 9.422752E+01 LZ-0.02-0.54-0.84					
1150		X -1.341410E+02 XY 1.279138E+02 A -1.917947E+01 LX 0.74-0.67-0.01 1.221451E+02 2.362160E+02	Y -1.615351E+02 YZ -5.608376E+00 B -2.768166E+02 LY 0.67 0.74 0.04	Z -7.075919E+01 ZX 6.160776E+00 C -7.043926E+01 LZ 0.02 0.04-1.00					
1154		X -4.359519E+00 XY 2.183030E+01 A -3.216376E+00 LX 1.00-0.05 0.04 1.772442E+02 4.203458E+02	Y -4.523264E+02 YZ 2.040710E+01 B -4.544753E+02 LY 0.05 1.00-0.05	Z -7.504665E+01 ZX 1.419538E+00 C -7.404085E+01 LZ 0.03-0.05-1.00					

Results for Subcase 2



153 OGRID CS 20 GP										PAGE 1667	
GEAR TOOTH EXAMPLE										SUBCASE 2	
GEAR TOOTH UNDER 6000 N CONCENTRATED LOAD											
STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)											
ELEMENT-ID	CORNER	-----CENTER AND CORNER POINT STRESSES-----				DIR.	COSINES	MEAN			
	GRID-ID	NORMAL	SHEAR	PRINCIPAL	-A-	-B-	-C-	PRESSURE	VON MISES		
	CENTER	X 3.748960E+01	XY 9.274559E+02	A 9.582781E+02	LX 0.67	-0.57	0.48	8.210181E+01	1.850457E+03		
		Y -2.163637E+02	YZ 5.144284E+02	B -1.176404E+03	LY 0.67	0.74	-0.05				
		Z -6.743133E+01	ZX -1.774132E+01	C -2.817965E+01	LZ 0.32	-0.35	-0.88				
518	X -6.415311E+03	XY 1.404840E+04	A 8.396160E+03	LX 0.48	-0.39	-0.79	1.135106E+04	4.166988E+04			
		Y -3.060588E+04	YZ 8.579084E+03	B -3.814591E+04	LY 0.35	0.91	-0.23				
		Z 2.968004E+03	ZX 2.786056E+03	C -4.303435E+03	LZ 0.80	-0.16	0.57				
516	X -3.302911E+02	XY -3.906447E+03	A 6.124154E+03	LX -0.50	0.49	-0.71	-3.324583E+01	9.495015E+03			
		Y 2.457141E+03	YZ -3.097814E+03	B -4.504655E+03	LY 0.80	0.57	-0.17				
		Z -2.027112E+03	ZX 2.621886E+02	C -1.519762E+03	LZ -0.32	0.66	0.68				
466	X -3.816256E+03	XY 2.414713E+03	A -1.048250E+03	LX -0.61	-0.65	-0.45	3.478813E+03	5.237950E+03			
		Y -4.691678E+03	YZ 1.890051E+02	B -6.865418E+03	LY -0.37	0.74	-0.57				
		Z -1.928507E+03	ZX -1.133415E+03	C -2.522771E+03	LZ 0.70	-0.18	0.69				
468	X -2.568760E+03	XY -5.898957E+02	A -6.934109E+02	LX 0.74	0.07	-0.67	5.457353E+03	8.831174E+03			
		Y -1.088833E+04	YZ 2.780636E+02	B -1.093242E+04	LY -0.06	1.00	0.03				
		Z -2.914962E+03	ZX -2.002919E+03	C -4.746223E+03	LZ -0.67	-0.02	-0.74				
1221	X 2.762466E+03	XY -3.584811E+03	A 1.198577E+04	LX 0.27	0.78	0.56	-5.937540E+03	1.022724E+04			
		Y 6.407855E+03	YZ -3.622293E+03	B 1.876837E+02	LY -0.64	0.58	-0.50				
		Z 8.642300E+03	ZX 2.908471E+02	C 5.639167E+03	LZ 0.72	0.22	-0.66				
1217	X 1.019409E+03	XY 4.563454E+02	A 1.777177E+03	LX 0.84	-0.50	-0.20	-1.335409E+02	2.501639E+03			
		Y -5.576572E+02	YZ 3.338279E+00	B -9.330429E+02	LY 0.17	0.60	-0.78				
		Z -6.112294E+01	ZX 1.107969E+03	C -4.435118E+02	LZ 0.51	0.63	0.59				
1150	X 1.260393E+02	XY 1.424698E+02	A 1.327638E+03	LX 0.17	0.73	0.66	-5.422709E+02	1.650370E+03			
		Y 1.302401E+03	YZ 5.642347E-01	B -5.177253E+02	LY 0.98	-0.06	-0.19				
		Z 1.983722E+02	ZX -6.730178E+02	C 8.168997E+02	LZ -0.10	0.68	-0.72				
1154	X 9.730048E+01	XY 5.365998E+02	A 8.835891E+02	LX 0.47	-0.26	-0.85	7.327244E+02	3.834592E+03			
		Y -1.696323E+03	YZ 1.785479E+03	B -3.256111E+03	LY 0.44	0.76	-0.47				
		Z -5.991506E+02	ZX -7.796404E+02	C 1.743489E+02	LZ 0.77	-0.59	-0.24				

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 6 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 1.14E3 MPa at grid point 518 of CHEXA element 153 (page 436 of the output in Listing 13-6). Figure 1-10 shows the overlay of undeformed and deformed shape with stress contour for subcase 1. Although the f06 shows 1.14E3 MPa, Patran shows 1.12E3 MPa due to the averaging method used by Patran during results processing. If the averaging method is set to 'None' the results will match. For misaligned gear teeth (subcase 2), the peak von Mises stress is 4.17E4 MPa at grid point 1357 of the output).



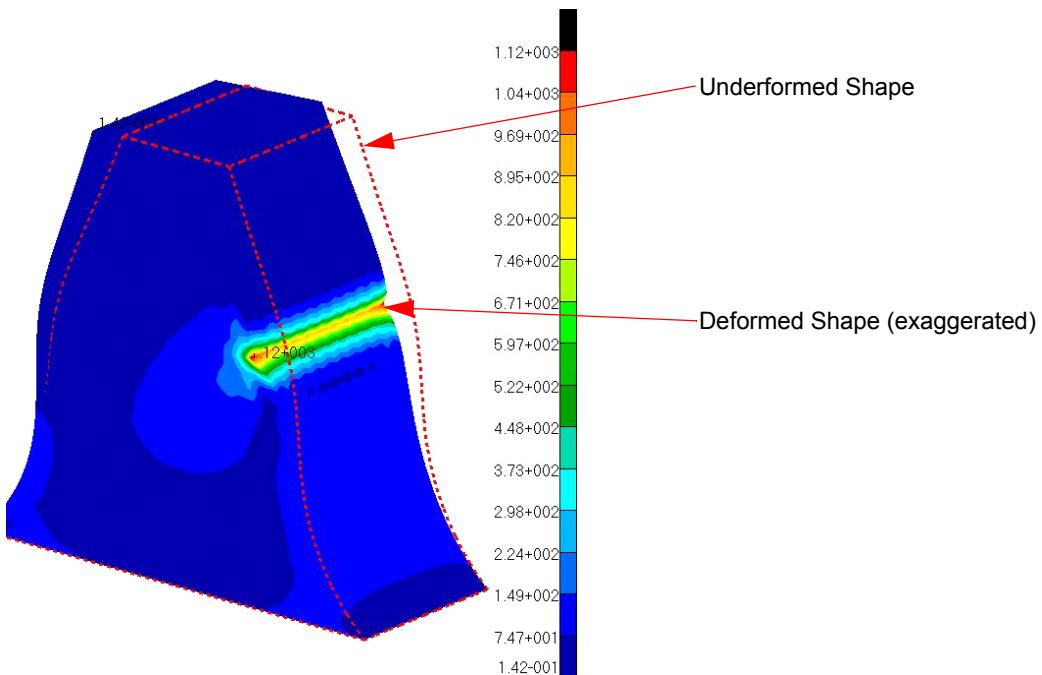


Figure 1-10 Stress and Deformation Contour of Subcase 1

Observe that for both subcases the von Mises stresses differ at grid points shared by two adjacent elements. 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 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.



Glossary

A

ACMS	Automated Component Modal Synthesis
Automated Component Modal Synthesis	
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 <i>Local Coordinate System</i> .
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

CAE	See <i>Computer Aided Engineering</i>
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.
CASI	See <i>CASI Iterative Solver</i>
CASI Iterative Solver	
CEL	See <i>Current Error List</i>
CMS	See Component Mode Synthesis
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.
Component Mode Synthesis	
Computer Aided Engineering	

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 of 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 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 The .dat file is an input file that contains the complete MSC Nastran finite element model. This file is submitted to MSC Nastran which then executes the analysis. It is also called the input file. It 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(s) of Freedom 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.

Direct Matrix Abstraction Program

It 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.

Distributed Memory Processing**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.

DMP

See Distributed Memory Processing

DOF**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 <i>Global Stiffness Matrix</i> .
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 <i>Ill-conditioning</i> .
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 <i>User Fatal Message</i> .
File Management Section	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.
FMS	See <i>File Management Section</i>
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*.

General purpose graphics processing units**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*.

GPU**GGPU****Grid Point**

See *General Purpose Graphics Processing Units*

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.

H**HDD**

See Hard Disk Drive

HPC

See High Performance Computing.

Hard Disk Drive**High Performance Computing**

It is the use of parallel processing for running advanced application programs efficiently, reliably, and quickly. The term applies especially to systems that function above a teraflop or 10^{12} floating-point operations per second. A highly efficient HPC system requires a high-bandwidth, low-latency network to connect multiple nodes and clusters.

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 not necessarily 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.
Mega Floating-Point Operations	It is a common measure of the speed of computers used to perform floating-point calculations. The greater the number of FLOPS a computer can calculate, the more powerful it is.
Million Instructions Per Second.	It is another common measure of computer speed and power, which indicates integer performance.
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.
MFLOP	<i>Mega Floating-Point Operations.</i>
MFLOPS	See <i>Mega Floating-Point Operations per second.</i>
MIPS	See <i>Million Instructions Per Second.</i>
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 <i>DMAP</i> .
MSCLDL	MSC Sparse Direct Solver
MSC Sparse Direct Solver	

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 <i>MSC Nastran Quick Reference Guide</i> 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.
Pardiso Sparse Direct Solver	
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 731 of the <i>MSC Nastran Quick Reference Guide</i> .
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.
PRDL DL	See <i>Pardiso Sparse Direct Solver</i>
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 <i>.f06 File</i> .
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.
RMS	See Root Mean Square
Root Mean Square	
Rotation	Displacement about a coordinate axis. A grid point has three rotational degrees of freedom, one about each axis. See also <i>Degree(s) of Freedom</i> and <i>Translation</i> .

S

See Single Point Constraint

SSD	See Solid State Device
SMP	See Shared Memory Processing
Set	A collection, or grouping, of particular items in the MSC Nastran model – examples include load sets, constraint sets, and collections of elements.
Shared Memory Processing	
Single Point Constraint	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 degrees of freedom. 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.
Solid State Device	
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 <i>DMAP</i> .
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	ee <i>Global Stiffness Matrix</i> and <i>Elemental Stiffness Matrix</i> .

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 <i>Decomposition</i> .
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:

$$\sigma_{von} = \frac{1}{\sqrt{2}} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{yz})^2 + 6(\tau_{zx})^2 + 6(\tau_{xy})^2]^{1/2}$$

Bibliography

1. *MSC Nastran Bibliography (Second Edition)*. Miller, M. and Navarro, E.; MSC Software 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.)
2. *MSC Nastran Primer for Linear Analysis*. Harry G. Schaefer; MSC Software Corporation, 2001.
3. *An Introduction to the Finite Element Method*. J. Reddy, McGraw Hill, 2005.
4. *The Finite Element Method: Basis and Fundamentals*. Olek Zienkiewicz, Robert Taylor, J. Z. Zhu; Elsevier, 2014.
5. *The Finite Element Method: Linear Statics and Dynamics*. Thomas J. R. Hughes; Dover, 1987.

Index

MSC Nastran Getting Started Guide

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

A

Aeroelasticity 59
Aspect ratio
 of CHEXA 208
 of CQUAD4 206
Axisymmetric analysis 252, 288

B

Basic element library 112
Beam element stress output 124
Beam orientation vector 121
Boundary conditions
 examples of 265
Bulk Data entries
 CBAR
 characteristics of 119
 examples of 256
 force and moment conventions 124
 format 120
 general description of 112
 CBEAM 116
 CELAS2 112
 CHEXA
 example of 274
 format of 140
 general description of 112
 CONROD 112, 116
 CORD1C 105
 CORD1R 105
 CORD1S 105
 CORD2C
 example of 108
 general description of 105
 CORD2R 105
 CORD2S 105
 CPENTA 112, 142
 CQUAD4

corner stress output 201
distortion of 206
element coordinate system 129
examples of 134, 264
force and moment conventions 130
format 128
general description of 112, 128
use in mesh transitions 98

CROD

format 118
general description of 112
CTETRA 112, 145
CTRIA3 98, 112, 131
DEFORM 189
FORCE 169, 173
FORCE1 173
FORCE2 173
GRAV 172, 173
GRID 214
LOAD 173
MOMENT 169, 173, 257
MOMENT1 173
MOMENT2 173
PBAR 112, 125
PLOAD 170, 173, 182
PLOAD1 169, 173, 177
PLOAD2 171, 173, 183
PLOAD4 171, 173, 184
PSHELL 112, 133
PSOLID 112, 147
RBE2
 general description of 112
SPC1 265
SPCD 173

C

Case Control commands
 DEFORM 173



A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

- ECHO 196
 ESE 201
 FORCE 201
 LOAD 173
 STRAIN 201
 STRESS 201
 summary list of 81, 83
- Case Control Section
 examples of 199
 general description of 196
 sets 199
- Elements
 CHEXA sixsided solid 112, 140, 274
 Comment entry (\$) 209
 Composite materials 62
 Coordinate system
 basic system 105
 CORD2C Bulk Data entry
 example of 108
 general description of 105
 local system 105, 109
- Elements
 CPENTA fivesided solid 112, 142
 crack growth 68
 crack initiation 64, 67
- Elements
 CTETRA foursided solid 112, 145
- Current error list 209
 cyclic material properties 65
 Cyclic symmetry 252
- D**
 DEFORM Case Control command 173
 Degrees of freedom
 common nomenclature for 103
 description of 102, 103
 Design sensitivity and optimization 56
 DISP Case Control command 200
 Dynamic analysis 42
- E**
 ECHO Case Control command 196
 Element strain energy 200
 Elemental stiffness matrix 37
- Elements
 CBAR simple beam 112, 119, 256
 CONROD rod 112, 116
 CQUAD4 quadrilateral plate 98, 112, 128, 134, 201, 206, 264
 CROD rod 112, 118
 CTRIA3 triangular plate 98, 112, 131
 distortion and accuracy of 206
 general notes concerning all 112
 output quantities 200
 PBAR simple beam property 112, 125
 PSHELL plate and shell element property 112, 133
 PSOLID solid element property 112, 147
 RBE2 rigid element 112
 spring element 113
 stress recovery coefficients (CBAR element) 126
- Enforced displacement 190
 Epsilon 205
 ESE Case Control command 201
 Example problems
 cantilever beam with distributed load and concentrated moment 256
 rectangular plate with concentrated loads 134
 rectangular plate with pressure load 264
 solid element gear tooth 274
- F**
 finite element methods 66
 Fluid structure interaction 62
 FORCE Case Control command 201
- G**
 geometry information 65
 Global stiffness matrix 40
 Grid point
 definition of 102
 degrees of freedom 102
 GRID Bulk Data entry 103
 output quantities 200
 rotation 102
 translation 102



A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

I

- Ill-conditioning 205
- Input File
 - character data 92
 - continuations 96
 - echo of, in the .F06 file 196
 - field formats, overview 86, 93
 - free field format 87, 94
 - large field format 88, 95
 - real numbers 92
 - small field format 87, 94
 - units 92
- Integer input data 92

J

- J (torsional constant) 117

K

- K (area factor for shear) 126

L

- life prediction methods 66
- Line (1D) Elements 115
- Elements
 - line (1D) elements 115
- Linear static analysis
 - flowchart for in MSC Nastran 37
- LOAD Case Control command 173
- loading information 65
- Loads
 - acceleration 186
 - concentrated forces 173, 223
 - concentrated moments 175
 - distributed loads on line elements 177
 - gravity 186
 - linear combinations of 190
 - load sets 173
 - multiple load cases using SUBCASE 192
 - overview of basic types 168
 - pressure loads 182
- local strain 64

M

- material information 65
- Material properties
 - composite materials 62
 - homogenous 152
 - isotropic 152
 - linear elastic (limitation) 152
 - MAT1 example 155
 - Poisson's ratio 152
- Matrix
 - stiffness (elemental) 37
 - stiffness(global) 40
- Mesh transitions 98
- Model design
 - accuracy 99
 - project budget 98
 - summary of issues in 98
- Model geometry
 - general description of 102
- monotonic material properties 65
- MSC Nastran
 - flowchart for linear static analysis 37
 - version of 226

O

- Optimization 56
- Orientation vector for CBAR and CBEAM 120
- Output titles 198

R

- Reactions
 - checking in statics 231
- Real input data 92
- Rigid element (RBE2) 148
- Rotation 102, 103

S

- SET Case Control command 199
- Skew, of CQUAD4 207
- S-N method 64
- Solid (3D) elements 139, 274
- Elements

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

- solid (3D) elements [139](#)
- Sorted bulk data echo [196](#)
- Stiffness matrix
 - elemental [37](#)
 - global [40](#)
- STRAIN Case Control command [201](#)
- strain-life [64](#)
- STRESS Case Control command [201](#)
- stress concentration [65](#)
- Stress recovery coefficients (CBAR element) [126](#)
- stress-life [64](#)
- Subcase [274](#)
- Superelements [250](#)
- Surface (2D) Elements [127](#)
- Elements
 - surface (2D) elements [127](#)
- System fatal message [204](#)
- System information message [204](#)
- System warning message [204](#)

T

- Test models [100](#)
- Thermal analysis [61](#)
- Torsional constant J [117](#)
- total life [64, 67](#)
- Translation [102, 103](#)

U

- Units [92](#)
- Unsorted bulk data echo [196](#)
- User fatal message [204](#)
- User information message [204](#)
- User warning message [204](#)

W

- Warp
 - of CHEXA [208](#)
 - of CQUAD4 [207](#)



A B C D E F G H I J K L M N O P Q R S T U V W X Y Z