

MSC.Nastran 2003

Linear Static Analysis
User's Guide

Corporate

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

Europe

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

Asia Pacific

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

Worldwide Web

www.mscsoftware.com

Disclaimer

This documentation, as well as the software described in it, is furnished under license and may be used only in accordance with the terms of such license.

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 © 2004 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.

The software described herein may contain certain third-party software that is protected by copyright and licensed from MSC.Software suppliers.

MSC, MSC /, MSC., MSC.Dytran, MSC.Fatigue, MSC.Marc, MSC.Patran, MSC.Patran Analysis Manager, MSC.Patran CATXPRES, MSC.Patran FEA, MSC.Patran Laminate Modeler, MSC.Patran Materials, MSC.Patran Thermal, MSC.Patran Queue Manager and PATRAN are trademarks or registered trademarks of MSC.Software Corporation in the United States and /or other countries.

NASTRAN is a registered trademark of NASA. PAM-CRASH is a trademark or registered trademark of ESI Group. SAMCEF is a trademark or registered trademark of Samtech SA. LS-DYNA is a trademark or registered trademark of Livermore Software Technology Corporation. ANSYS is a registered trademark of SAS IP, Inc., a wholly owned subsidiary of ANSYS Inc. ABAQUS is a registered trademark of ABAQUS Inc. ACIS is a registered trademark of Spatial Technology, Inc. CATIA is a registered trademark of Dassault Systemes, SA. EUCLID is a registered trademark of Matra Datavision Corporation. FLEXlm is a registered trademark of GLOBEtrotter Software, Inc. HPGL is a trademark of Hewlett Packard. PostScript is a registered trademark of Adobe Systems, Inc. PTC, CADDs and Pro /ENGINEER are trademarks or registered trademarks of Parametric Technology Corporation or its subsidiaries in the United States and /or other countries. Unigraphics, Parasolid and I-DEAS are registered trademarks of Electronic Data Systems Corporation or its subsidiaries in the United States and /or other countries. All other brand names, product names or trademarks belong to their respective owners.

C O N T E N T S

MSC.Nastran Linear Static Analysis User's Guide

Preface

- About This Book, xii
- List of MSC.Nastran Books, xiii
- Technical Support, xiv
- Internet Resources, xvii
- Permission to Copy and Distribute MSC Documentation, xix

1

Introduction

- Introduction, 2
- Scope, 3
- Using the Finite Element Method, 4
- A Simple Problem, 11

2

Organization of MSC.Nastran Files

- Introduction to Organization of MSC.Nastran Files, 20
- Overview of the MSC.Nastran Input Files, 21
 - The NASTRAN Statement, 23
 - File Management Section, 23
 - Executive Control Section, 23
 - Case Control Section, 23
 - Bulk Data Section, 24
- Overview of the MSC.Nastran Output Files, 27
- The Executive Control Section, 32
- The Case Control Section, 34
- The Bulk Data Section, 41

3

Grid Points, Scalar Points, and Coordinate Systems

- Introduction to Degrees of Freedom, 48
- Grid Points, 49
- Scalar Points, 51
- Coordinate Systems, 52
 - User-Defined Coordinate Systems, 54

4

MSC.Nastran Elements

- CORD1R and CORD2R Bulk Data Entries, 54
 - CORD1C and CORD2C Bulk Data Entries, 58
 - CORD1S and CORD2S Bulk Data Entries, 59
 - The Element and Material Coordinate System, 62
- Introduction to MSC.Nastran Elements, 66
 - Scalar Elements, 66
 - One-Dimensional Elements, 66
 - Two-Dimensional Elements, 67
 - Three-Dimensional Elements, 68
- Scalar Elements, 69
- One-Dimensional Elements, 73
 - The CROD Element, 73
 - The CBAR Element, 77
 - PBARL -- An Alternate and Convenient Method for Defining Bar Cross Sections, 87
 - The CBEAM Element, 103
 - PBEAML-- An Alternate and Convenient Method for Defining CBEAM Cross Sections, 113
- Two-Dimensional Elements, 127
 - The CQUAD4 and CTRIA3 Elements, 127
 - The CQUAD8 and CTRIA6 Elements, 139
 - Shell Element Force, Stress, and Strain Output in the Material Coordinate System, 140
 - Input, 141
 - Output, 141
 - Guidelines and Limitations, 142
 - The CQUADR and CTRIAR Elements, 142
 - The Shear Panel Element (CSHEAR), 143
 - Two-Dimensional Crack Tip Element (CRAC2D), 145
- Three-Dimensional Elements, 148
 - The CHEXA, CPENTA, and CTETRA Elements, 148
 - The CTRIAX6 Element, 152
 - Three-Dimensional Crack Tip Element (CRAC3D), 153
- CBUSH Elements, 156
- The GENEL Element, 160
- CWELD Connector Element, 165
 - Introduction, 165
 - Connectivity Definition, 165
 - Property Definition, 168
 - Finite Element Representation of the Connector, 169
 - Results Output, 170
 - Example (CWELD101a-b.DAT), 171
 - Guidelines and Remarks, 174

5

Material Properties and Composites

- Introduction to Material Properties, 178
- Isotropic Material (MAT1), 179
- Two-Dimensional Anisotropic Material (MAT2), 181
- Axisymmetric Solid Orthotropic Material (MAT3), 182
- Two-Dimensional Orthotropic Material (MAT8), 183
- Three-Dimensional Anisotropic Material (MAT9), 187
 - Part 1 -- Develop the Material Matrix, 189
 - Part 2 -- Verify the Material Matrix, 190
 - Part 3 - Use the Anisotropic Material in the Actual Structure, 193
- The PSHELL Entry, 197
- The Composite Element (PCOMP), 199
- Equivalent Composite Honeycomb Section, 208

6

Static Loads

- Introduction to Static Loads, 218
- Loads on Grid Points and Scalar Points, 219
- Distributed Loads on Line Elements, 223
- Distributed Loads on Surfaces, 230
- Gravity and Centrifugal Force (GRAV, RFORCE), 234
 - Defining Mass in Your Model, 236
- Initial Strains (DEFORM), 241
- Load Combinations, 245
- Thermal Loads, 249
 - Using the SUBCOM with Thermal Loads, 249

7

Constraints

- Introduction to Constraints, 259
- Single-Point Constraints, 260
- Automatic Application of Single-Point Constraints (AUTOSPC), 263
- Enforced Displacements at Grid Points (SPCD, SPC), 267
- Multipoint Constraints, 271
- Contact Problem Using Linear Static Analysis, 275
 - Introduction, 275
 - Input, 275
 - Output, 276
 - Limitations, 276
 - Examples, 277

8

R-Type Elements

9

Modeling Guidelines

10

Model Verification

- Introduction to R-Type Elements, 286
 - Description of the R-Type Elements, 287
 - The RBAR Element, 290
 - The RBE2 Element, 297
 - The RBE3 Element, 301
-
- Introduction to Modeling Guidelines, 312
 - Choosing the Right Element, 313
 - General Guidelines, 313
 - Zero-Dimensional Elements, 313
 - One-Dimensional Elements, 314
 - Two-Dimensional Elements, 314
 - Shell Normals, 316
 - Remarks and Recommendations, 318
 - Three-Dimensional Elements, 318
 - R-Type Elements, 319
 - Mesh Density, 320
 - Mesh Transitions, 327
 - Transition from a Coarse Mesh to a Fine Mesh, 327
 - Mesh Transition Between Dissimilar Element Types, 336
 - Shell-to-Solid Transition Element (RSSCON), 338
 - Grid Point Stresses, 349
 - Description of the Grid Point Stress Method, 351
 - Topological Method - Plate Elements and Solid Elements, 352
 - Treatment of Exception Points, 356
 - Grid Point Stress User Interface, 357
 - Consistent Loading, 360
 - Symmetry, 363
 - User Interface, 365
-
- Introduction to Model Verification, 370
 - Preprocessor Checks, 371
 - Shrink Plot, 371
 - Hidden Line/Shaded Plots, 372
 - Free Edge/Face, 373
 - Zipper Effect, 374
 - CBAR/CBEAM Orientation and Offset Check, 375
 - Duplicate Grid Points/Elements, 375
 - Properties/Material Plots, 375
 - Consistent Plate Normals, 375
 - Strain Energy Output, 379

11

Inertia Relief

12

Matrix Operations

13

Linear Buckling

- Diagnostic Tools, 386
 - Element Summary Output, 387
 - Element Geometry Checks, 387
 - Weight Center of Gravity and Moment of Inertia Check, 395
 - Mechanisms and Singularities, 397
 - Applied Loads Check, 400
 - Reaction Load, 402
 - 1g Load, 403
 - Output of Maximum, Minimum Grid Point-Based Responses, 403
 - Unconstrained Equilibrium Check, 409
 - Automatic Identification of Unintentional Grounding, 410
 - Thermal Equilibrium Check, 412
 - Grid Point Force Balance, 412
- Stress Error Estimators, 418
 - Grid Point Stress Discontinuities, 419
 - Element Stress Discontinuities, 423
 - Discussion of Error Measures, 425
 - User Interface, 426
- Postprocessor Checks, 428
 - Summary, 433
- Introduction to Inertia Relief, 436
- Description of Inertia Relief Using PARAM,INREL,-1, 437
- Implementation of Inertia Relief in MSC.Nastran Using PARAM,INREL,-1, 438
- Automatic Inertia Relief, 451
 - Limitations for the Automatic Inertia Relief Method, 452
- Introduction to Matrix Operations, 456
- Set Definition, 457
 - The Global Displacement Set, 457
 - Subsets of the Global Displacement Set, 457
- Static Condensation (Guyan Reduction), 462
 - User Interface, 463
- Direct Matrix Input, 471
 - DMIG, 472
 - DMIG Bulk Data User Interface, 472
 - DMIG Case Control User Interface, 474
 - Stiffness Matrix Using DMIG, 475
 - Mass Matrix Using DMIG, 481
 - Load Matrix Using DMIG, 481
- Introduction to Linear Buckling, 486

14

Restarts

- Finite Element Approach, 487
- Eigenvalue Extraction Method, 493
 - Inverse Power Method (INV), 493
 - Enhanced Inverse Power Method (SINV), 493
 - Lanczos Method, 494
 - Comparison of Methods, 494
 - User Interface, 494
- Assumptions and Limitations of Linear Buckling Analysis, 497
- Examples, 498
 - Example 1 -- Classical Euler Beam Buckling, 498
 - MSC.Nastran Results for Example 1, 499
 - Theoretical Results for Example 1, 502
 - Example 2 -- Lateral Buckling, 503
 - MSC.Nastran Results for Example 2, 503
 - Theoretical Results for Example 2, 507
 - Example 3 -- Planar Frame Buckling, 508
 - MSC.Nastran Results for Example 3, 509
 - Theoretical Results for Example 3, 510
 - Example 4 -- Buckling of a Stiffened Panel with Transverse Shear Flexibility, 510
 - MSC.Nastran Results for Example 4, 511
 - Theoretical Results for Example 4, 513
 - Example 5 -- Buckling of a Cylinder Under Uniform Axial Load, 514
 - MSC.Nastran Results for Example 5, 515
 - Theoretical Results for Example 5, 517
 - Example 6 -- Multiple Buckling Analyses in a Single Run, 518

- Introduction to Restarts, 522
- Types of Restarts, 523
- Structure of the MSC.Nastran Input File, 524
- User Interface for Read-Write Restart, 525
 - Cold Start Run, 525
 - Restart Run, 525
- Determining the Version for a Restart, 535
- Read-Only Restarts, 537
- Mini-Database Restarts, 538
- Miscellaneous Applications, 539
 - Bulk Data Listing, 539
 - Summary, 539

15

Database Management

- Introduction to Database Management, 542
- Definitions, 543
- MSC.Nastran Database, 544

16

Cyclic Symmetry

- File Management Section, 545
 - INIT, 545
 - ASSIGN, 547
 - EXPAND, 548
 - RESTART, 549
 - DBCLEAN, 550
 - DBDIR, 551
 - INCLUDE, 552
- Guidelines for Large Problems, 553
 - BUFFSIZE, 555
- Introduction to Cyclic Symmetry, 560
- Description of Geometric Symmetry in MSC.Nastran, 561
- Mathematical Description of Structural Symmetry, 567
 - Reflective Symmetry, 567
 - Axisymmetry, 570
 - Rotational Symmetry, 572
 - Dihedral Symmetry, 577
- Bulk Data Entries for Cyclic Symmetry, 582
 - LOADCYN, LOADCYH, and LOADCYT Bulk Data Entries, 585
- Case Control Section for Cyclic Symmetry, 589
- Axisymmetric Example, 594

A

Glossary of Terms

B

Grid Point Weight Generator

- Introduction to Grid Point Weight Generation, 614
- Commonly Used Features, 615
- Example with Direction-Dependent Masses, 618

C

Summary of Basic
Case Control
Commands

D

Summary of Basic
Bulk Data Entries

E

Output Listing

■ Examples List, 640

F

Parameters

G

Resource
Estimation

H

Generating the
Cylinder in
MSC.Patran

■ Procedures, 682

I

References

Preface

- About This Book
- List of MSC.Nastran Books
- Technical Support
- Internet Resources
- Permission to Copy and Distribute MSC Documentation

About This Book

The Version 2001 of the *MSC.Nastran Linear Static Analysis User's Guide* is part of the ongoing project to update existing MSC.Nastran documentation. The primary editors for previous versions of this User's Guide include Dr. Richard MacNeal (Version 61), Mr. Michael Gockel (Version 64), Mr. John Caffrey and Mr. John Lee (Version 68), and Mr. John Lee (Version 69+).

This guide describes additional capabilities related to linear static analysis that have been added since version 69. Some of the major capabilities include weld element, automatic inertia relief, linear gap element, rigid body check, model summary, and grid point results max / min listing.

The editor is grateful to Ms. Wendy Webb, Mr. Gavin Cline, and Ms. Lori Lampert for their patience and dedication in updating this user's guide. The editor would also like to thank Mr. John Halcomb for his technical review of this guide.

John M. Lee

December 2001

List of MSC.Nastran Books

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

Installation and Release Guides

- ☐ Installation and Operations Guide
- ☐ Release Guide

Reference Books

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

User's Guides

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

Technical Support

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

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

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

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

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

Phone and Fax

United States

Telephone: (800) 732-7284

Fax: (714) 784-4343

Frimley, Camberley

Surrey, United Kingdom

Telephone: (44) (1276) 67 10 00

Fax: (44) (1276) 69 11 11

Munich, Germany

Telephone: (49) (89) 43 19 87 0

Fax: (49) (89) 43 61 71 6

Tokyo, Japan

Telephone: (81) (3) 3505 02 66

Fax: (81) (3) 3505 09 14

Rome, Italy

Telephone: (390) (6) 5 91 64 50

Fax: (390) (6) 5 91 25 05

Paris, France

Telephone: (33) (1) 69 36 69 36

Fax: (33) (1) 69 36 45 17

Moscow, Russia

Telephone: (7) (095) 236 6177

Fax: (7) (095) 236 9762

Gouda, The Netherlands

Telephone: (31) (18) 2543700

Fax: (31) (18) 2543707

Madrid, Spain

Telephone: (34) (91) 5560919

Fax: (34) (91) 5567280

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

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

Training

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

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

The MSC Institute of Technology is located at:

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

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

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

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

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

Internet Resources

MSC.Software (www.mscsoftware.com)

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

Simulation Center (simulate.engineering-e.com)

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

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

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

MSC.Linux (www.mscsoftware.com/hpc)

Now with almost 40-years of unparalleled experience in scientific and technical computing, MSC.Software is leveraging this knowledge to deliver its customers state-of-the-art, high performance computing solutions based on clustered computing for running engineering and life sciences applications.

CATIASOURCE (plm.mscsoftware.com)

Your SOURCE for Total Product Lifecycle Management Solutions.

Process Architecture Lab (PAL) (pal.mscsoftware.com/services/pal)

PAL is a virtual product development environment that enables PAL participants and customers to define, validate, and demonstrate advanced tools, processes, and e-business solutions.

Permission to Copy and Distribute MSC Documentation

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

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

Please complete and mail to MSC for approval:

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

Name: _____

Title: _____

Company: _____

Address: _____

Telephone: _____ Email: _____

Signature: _____ Date: _____

Please do not write below this line.

APPROVED: MSC.Software Corporation

Name: _____

Title: _____

Signature: _____ Date: _____

Fold here

Place
Stamp
Here

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

Fold here

CHAPTER

1

Introduction

- Introduction
- Scope
- Using the Finite Element Method
- A Simple Problem

1.1 Introduction

This book is one in a series of user's guides describing the use of MSC.Nastran. The first book in the series, *Getting Started with MSC.Nastran User's Guide*, is intended for those readers who are unfamiliar with the finite element method, those who have never run MSC.Nastran, or those who are interested in a general overview of MSC.Nastran's capabilities. It is assumed that you are familiar with the material in *Getting Started with MSC.Nastran User's Guide* and have MSC.Nastran available.

All important features and concepts in this book are illustrated with examples. As you read this guide, it is highly recommended that you run the example problems and make variations of these problems. When learning a new MSC.Nastran feature, engineers all too often generate a large problem using several hundred or several thousand degrees of freedom as a test case. This practice has become the norm in recent years with the advent of graphics preprocessors and automatic meshing. Rarely is such a large model necessary to learn a new feature--in most cases, it just adds unnecessary complexity. For this reason most of the examples in this book are small--generally less than 100 degrees of freedom. To facilitate the use of these example problems, all of the input files used in this guide are included with the standard MSC.Nastran delivery media.

To keep this user's guide to a reasonable size, complete descriptions of all the MSC.Nastran Bulk Data entries and Case Control commands have not been included. A brief description of the input format is given for entries and commands when it is helpful to understand the material. You should consult the *MSC.Nastran Quick Reference Guide* for detailed descriptions of MSC.Nastran input formats.

1.2 Scope

The main topic of this user's guide is linear static analysis. The goal is to provide material relevant to this subject in such a manner that this book can be used both as a learning tool and as a reference text.

The first eight chapters are devoted to the basics-namely, the generation of an MSC.Nastran input file, definitions of specific input entries (elements, loads, constraints, etc.), and typical results you can expect from a linear static analysis. The latter part of this guide focuses on modeling techniques and model verification. “**Restarts**” on page 521 and “**Database Management**” on page 541 discuss restarts and database management. These topics are generally not needed for the small example problems used throughout this guide. However, these chapters are very useful when running larger production jobs.

“**Linear Buckling**” on page 485 is devoted to linear buckling analysis. It might be argued that buckling does not fall within the realm of linear static analysis. However, the assumption is that those engineers who perform static analyses are also the ones most likely to perform buckling analyses. Accordingly, a full chapter is devoted to buckling.

1.3 Using the Finite Element Method

Before discussing the specifics of the finite element method using MSC.Nastran, it is useful to examine how you might use the finite element method to solve an engineering problem. The discussion in this section is quite general and is not meant to show you the specifics of generating and running a finite element computer model. Rather, the purpose of this discussion is give you some insight into how and why you would use the finite element method.

The finite element method is used in a wide variety of disciplines and engineering applications. One of the most common applications of the method is to solve structural analysis problems. Traditionally, structural analysis applications are divided into two areas—static and dynamic analyses. Performing structural analysis using the finite element analysis is no exception. It is not unusual to find individuals who spend the majority of their time performing static finite element analysis and little or no time performing dynamic analysis. Likewise, it is common to find individuals who spend the majority of their time performing dynamic analysis. This guide deals exclusively with static analysis, whereas the *MSC.Nastran Basic Dynamics User's Guide* is available for those who are interested in dynamic analysis. However, you should find the material in this guide worthwhile regardless of your specialty area.

When performing static analysis using the finite element method, the structure is represented as a collection of discrete elements. Think of these elements as building blocks from which you can construct a model of the actual structure. Using the properties of these individual building blocks (the elements), the stiffness of the whole structure is approximated. The loads on the structure are represented as forces acting on the elements or on the boundary of the elements.

To use the finite element method effectively, you must apply your engineering knowledge to the structure being analyzed. Next you must decide how much detail to put in your model, i.e., the size and number of elements. More detail in the model results in a more accurate solution but it costs more in terms of computer resources. The goal is to create an accurate, yet cost-effective model. The better your understanding of the structure, the applied loads, and the MSC.Nastran solution process, the more effective you will be in finding solutions to your problems.

Suppose you have to design a circular tube as shown in [Figure 1-1](#) and you are considering using MSC.Nastran. The tube has an outer diameter of 20 mm and a wall thickness of 0.5 mm. The tube is attached at both ends to a thick bracket which is fixed rigidly to the wall. Both the tube and the bracket are made of aluminum. The loading consists of a transverse force of 1000 N acting on a ring 10 mm in width located in the center of the tube.

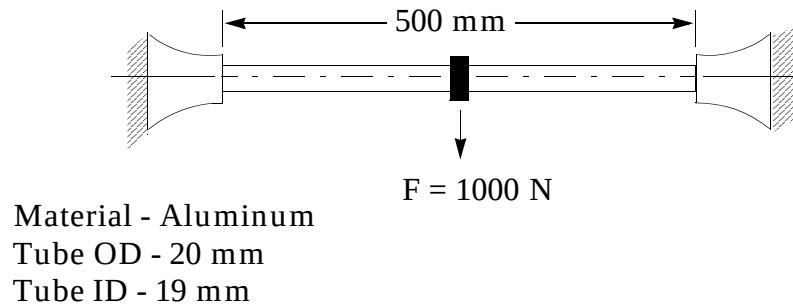


Figure 1-1 Thin Walled Tube with a Transverse Load

Before starting a computer model, ask the following questions:

1. What are the load paths for this structure? Knowing the load path for the structure provides insight regarding what part of the structure needs to be modeled in detail. Furthermore, knowing how the loads are transmitted to the structure helps you to understand and verify the results.
2. What is our design criteria? You may be concerned that the applied load causes local yielding of the material, or yielding may be a secondary concern and your main concern may be deflection. Another concern may be that of the tube buckling. The answer to these questions dictates how much detail is needed in your model.
3. If yielding is of concern, where are the higher stresses expected to occur, and what failure criteria should be used?
4. Is the structure displacement considered small? If the displacement is not small compared to the dimensions of the tube, is it necessary to perform a nonlinear analysis? Planning ahead as to what additional analyses may be necessary can reduce the time necessary to convert the input file to a nonlinear analysis if needed.
5. How can the results of the finite element analysis be verified? In other words, what independent checks can be made to ensure that the answers are reasonable?

Returning to the tube structure, assume that the primary design criteria are to have an adequate margin-of-safety and a critical buckling load that is at least three times the applied load.

The load for this structure consists of a force applied to the stiff ring at the center. The stiff ring prevents the tube from crushing locally where the force is applied. From the geometry of the tube, you expect the displacement and stress distribution to be similar to that of a classical fixed-fixed beam with a center load.

A good first step in any analysis is to estimate the output quantities that you are trying to determine. In the example, the output quantities are the stresses and displacements of the tube. Assuming that the tube behaves as a classical beam, then the stresses and displacements can be found using standard engineering formulas. Once you have an estimate, you can identify modeling errors quickly. For many engineering problems, you

cannot look in a handbook for a formula. However, you can always make some simplifying assumptions and obtain a crude estimate. Having a good estimate, simplifies the model checkout.

For the structure, the maximum stress and displacement are estimated to be 429 N/mm^2 and 2.24 mm , respectively (using standard engineering formulas). If the finite model results are substantially different from these values, further investigation is warranted

Now you are ready to generate the finite element model. At this point, you need to decide what kind of elements should be used and how much detail is needed.

If the tube structure acts like a beam—that is, plane sections remain plane and the neutral plane remains strain free—then the structure can be modeled using the CBAR one-dimensional elements as shown in **Figure 1-2**.

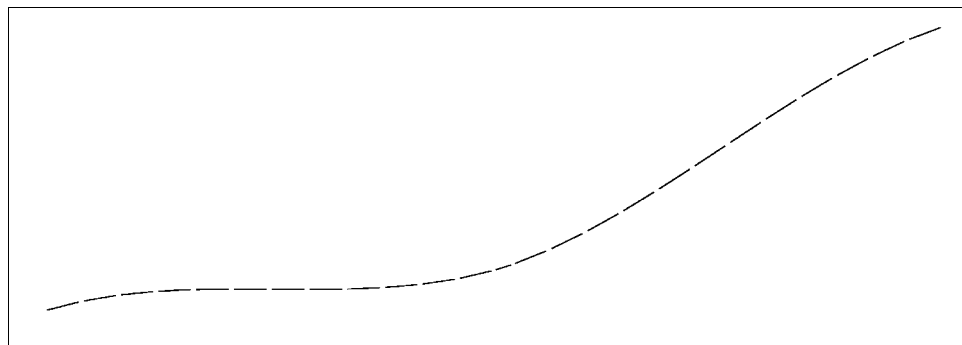


Figure 1-2 Deformed Shape of the Tube Structure Modeled with CBAR Elements

To model the structure with CBAR elements, divide the tube into several short sections along the length of the tube. Each of the short sections is represented as a single CBAR element. The CBAR elements are connected to their neighbor elements at points known as grid points. The physical locations of the grid points determine the length of each of the CBAR elements. Associated with each of the CBAR elements is a property entry with the following properties: the cross-sectional area, two area moments of inertia, and a torsional constant. The properties represent the physical properties of the tube and must be entered for each element. The material properties—consisting of the modulus of elasticity and Poisson's ratio—are entered on a material entry.

Twenty-five elements are chosen primarily for plotting purposes. If less elements are used, the maximum stress will be the same, but the deformed shape will not be as smooth (this is true only for simple 1-D elements). To represent the stiff ring at the center, the CBAR element at the center is given the same length as the ring width, and the properties of the elements in this section are chosen to represent a very stiff section.

The results of the one-dimensional CBAR model are similar to those generated with the hand calculation—the maximum stress is 429 N/mm^2 and the center deflection is 2.34 mm . The deflection of the tube is slightly larger for the finite element model (2.34 versus 2.24) because shear flexibility is included in the finite element model.

Having reviewed the results of the CBAR model, suppose you think that the thin tube may bend such that the beam theory is no longer a valid assumption. If the structure cannot be modeled using beam theory, a more detailed model is required such as the one shown in

Figure 1-3. As can be seen, the tube is treated as a curved surface as opposed to the beam model shown in **Figure 1-2**. The surface is modeled using two-dimensional elements with each element attached to its neighboring elements at the discrete grid points.

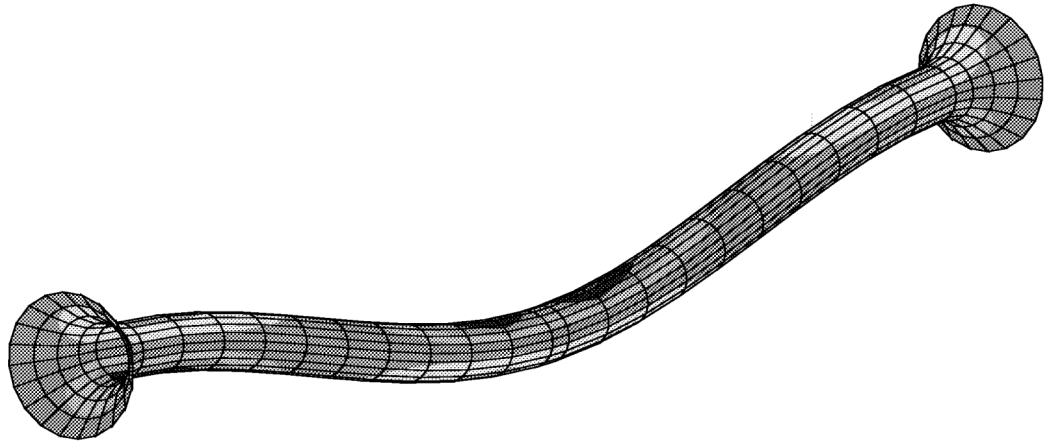


Figure 1-3 Deformed Shape of Coarse CQUAD4 Element Model

This concept of representing the real world structure with finite elements is generally referred to as meshing. Hence, you can say that the tube structure is meshed with CQUAD4 elements. For most models, meshing is performed with the use of a preprocessor such as MSC.Patran. The relative number of elements used to represent the structure or part of a structure is commonly referred to as mesh density. As the mesh density is increased, more elements are needed and the size of the elements is decreased. The process of increasing the mesh density to improve the accuracy of the results is also referred to as refining the model. **Figure 1-4** shows the tube model being refined from the 400 elements shown in **Figure 1-3** to 3104 elements.

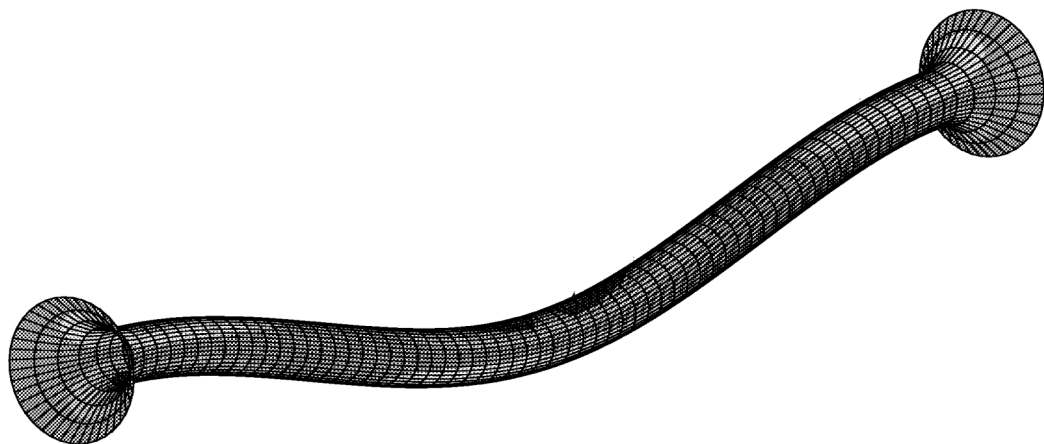


Figure 1-4 Deformed Shape of Fine CQUAD4 Element Model

Table 1-1 summarizes the maximum normal stress for the tube structure for each of the analyses performed.

Table 1-1 Maximum Stress and Displacement for the Tube Structure

	Hand Calculation	Beam Model	Coarse 2-D Model	Refined 2-D Model
	--	24 CBARs	400 CQUAD4s	3140 CQUAD4s
Max Stress (N / mm ²)	429	429	396	432
Center Deflection (mm)	2.24	2.34*	2.51	2.52
CPU Time (sec)	--	12.1	20.8	110.1

*Includes transverse shear flexibility.

Now consider the second part of our design criteria: buckling. Using your knowledge of the structure, you know that if the structure buckles, it will most likely do so by crippling the thin wall of the tube. This type of buckling cannot be predicted using the CBAR model. You need to use a two-dimensional model in which surfaces are capable of representing the buckled shape. The question is: how much detail is needed in a mesh?

Using the coarse CQUAD4 model shown in [Figure 1-3](#), the critical buckling load is calculated to be 37,280 N. The corresponding buckled shape is shown in [Figure 1-5](#).

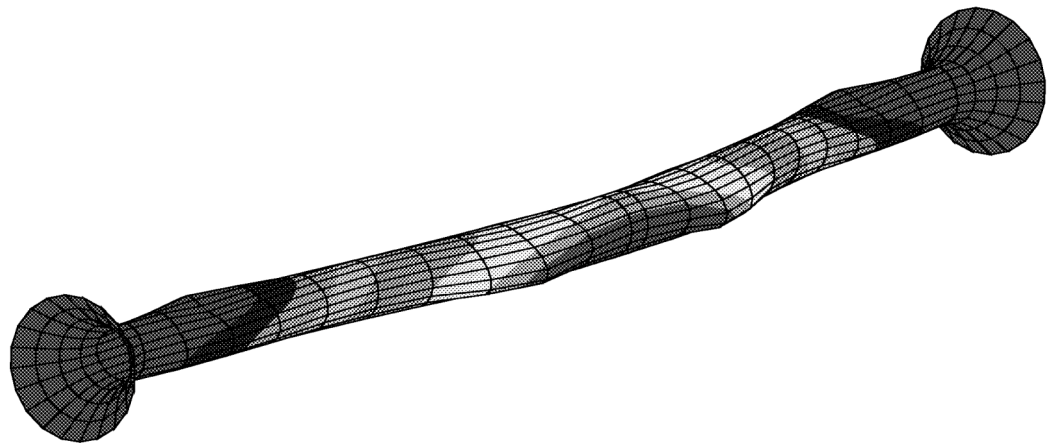


Figure 1-5 Buckled Shape Using the Coarse CQUAD4 Model

Using the refined CQUAD4 model shown in [Figure 1-4](#), the critical buckling load is 23,757 N, which is substantially less than the predicted value using the coarse model. The corresponding buckled shape is shown in [Figure 1-6](#).

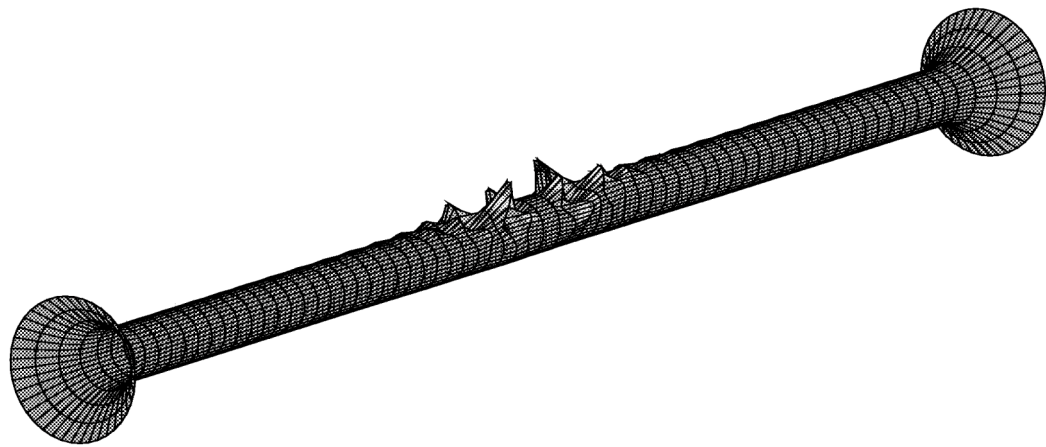


Figure 1-6 Buckled Shape Using the Fine CQUAD4 Model

The reason for the lower predicted critical load is found in [Figure 1-6](#). The buckled shapes shown in [Figure 1-5](#) and [Figure 1-6](#) are very different. The buckled shape for the coarse model looks like an overall crushing of the tube, which is not what you would expect. The buckled shape for the fine model shows localized crippling around in the vicinity of the loaded stiff ring. The coarse model is simply not detailed enough to adequately represent the buckled shape of the thin shell structure.

Returning to our original design goals, which model is best for this project? If you are only interested in the stresses and displacement for the single load, as shown in [Figure 1-1](#), then the CBAR model—even the hand calculation—will suffice. If, however, the applied load is more complex (for example, a torsional load), then the coarse model is useful because the stress pattern is more complex. Also, the CQUAD4 element stress output includes the von Mises stresses that can be useful in your failure analysis.

If buckling is a concern, a fine mesh is required. In fact, even the fine mesh shown in [Figure 1-5](#) is not adequate as can be seen by inspecting the buckled region. This buckled region must have several elements per wavelength and must not have the sharp edges between the elements. Whenever you see the sharp edges, you need to further refine the model to see if the critical load or the buckled shape changes significantly.

As a final comment on this example, if after refining the model to get a reasonable buckled shape, you find the critical load close to the applied load, you probably would want to run a nonlinear analysis including both material and geometric nonlinearity.

When you model a region with a particular element, note that you are approximating the actual stiffness in that region with the stiffness of the element that you have chosen. These finite elements, which are commonly referred to as simply “elements,” can be scalar elements or one-dimensional elements (such as the CBAR element), two-dimensional elements (such as the CQUAD4 element), or three-dimensional elements. All of the MSC.Nastran linear elements are discussed in “[MSC.Nastran Elements](#)” on page 65.

It is your responsibility to choose the appropriate element types and number of elements to obtain the best results. If you use a one-dimensional element to model a two-dimensional plate structure, you can expect poor results. If, on the other hand, you choose a two-dimensional element type for your plate structure, the approximation will be

considerably better. This choice requires engineering judgment. It is easy to use a graphics preprocessor to generate a good-looking model of your structure that contains many elements, but the model can be a poor model for the analysis. This problem is further compounded by the fact that today's postprocessors can take the most meaningless results and present them in a very convincing manner.

Having examined how the finite element method can be used in an engineering project, now look at the steps a finite element program, such as MSC.Nastran, follows in solving a very simple model. This is the subject of the next section.

1.4 A Simple Problem

In the last section, a sample engineering problem was presented that showed how a MSC.Nastran model can fit an engineering project; however, there was no discussion of the internal processing. In this section, the finite method is discussed from the opposite point of view, namely, an examination of the steps performed by MSC.Nastran to solve a linear statics problem. To keep the equations to a reasonable size, a simple two-element model is used. A larger model will not help in understanding the method; it will only add complexity in the relationships between the physical structure and the equations.

Figure 1-7 shows the basic steps that MSC.Nastran follows when solving a linear statics analysis. As shown in the figure, you develop a model of your structure by dividing your structure into small but discrete elements.

Each element is connected to its neighboring elements at the grid points (commonly referred to as “nodes” in many textbooks). Typically, you lay out the pattern of grid points first and connect the grid points by the elements, remembering that both the location of the grid points and the number of grid points determine the size and number of elements in your model. Associated with each of the grid points in the model are six components of motion—namely, the translations and rotations in the three perpendicular directions (e.g., the X , Y , Z , R_X , R_Y , and R_Z directions). These components of motion are referred to as “degrees of freedom.” The collection of all of the degrees of freedom in your model make up the global displacement set.

Connected between the grid points are the elements. The stiffness of each element is represented internally in matrix form, commonly referred to as the “element stiffness matrix.” The size of the element stiffness matrix is dependent on the element type. The terms in the element stiffness matrix are based on user input such as the material type, the element properties, and the element geometry. The element geometry is determined by the location of the grid points to which the element is connected.

All of the element stiffness matrices are assembled into a single matrix called the “global stiffness matrix.” This global stiffness matrix represents the total structural stiffness before the boundary conditions are considered; therefore, the global stiffness matrix is, in general, a singular matrix. The physical meaning of a singular stiffness matrix is that the whole structure or part of the structure can displace as a free body without producing any internal forces in the members.

In static analysis, adequate boundary conditions must be applied to the model in order to prevent any rigid body motion of your structure. If the specified boundary conditions do not adequately constrain the model in all directions, the structure’s stiffness matrix remains singular and the run terminates with an error message. Once the boundary conditions are applied to the model appropriately, the global stiffness matrix is reduced to a nonsingular stiffness matrix representing the constrained structure.

All of the loads that you apply to the model are combined to form the load vector. These applied loads can be in the form of point forces and moments applied directly to the grid points, line loads applied along the length of one-dimensional elements, surface loads applied to two- and three-dimensional elements, or body loads such as gravity. These

different load types may be combined to form a single load vector, which is the same as saying that the loads are applied simultaneously. You also have the option of applying multiple load vectors within a single run.

After the constrained stiffness matrix and the load vector are generated, the static equilibrium matrix equation given by **Eq. 1-1** is solved as follows:

$$[K]\{u\} = \{p\} \quad \text{Eq. 1-1}$$

where:

$[K]$ = system stiffness

$\{u\}$ = grid point displacements

$\{p\}$ = applied load vector

The unknowns in **Eq. 1-1** are the displacements $\{u\}$ at the grid points in the model. Determining the displacements involves the equivalent of inverting the stiffness matrix $[K]$ and multiplying it by the force vector $[p]$. Actually, the process of inverting a matrix is too time consuming; therefore, a process based on the Gauss elimination method is used. The resulting displacements are the same--the solution process is just faster.

Once the displacements at the grid points are known, any desired outputs, such as element forces, strains, and stresses, are computed using those displacements on an element-by-element basis. You can control what type of output is generated and whether the output is printed, punched to a file, or plotted.

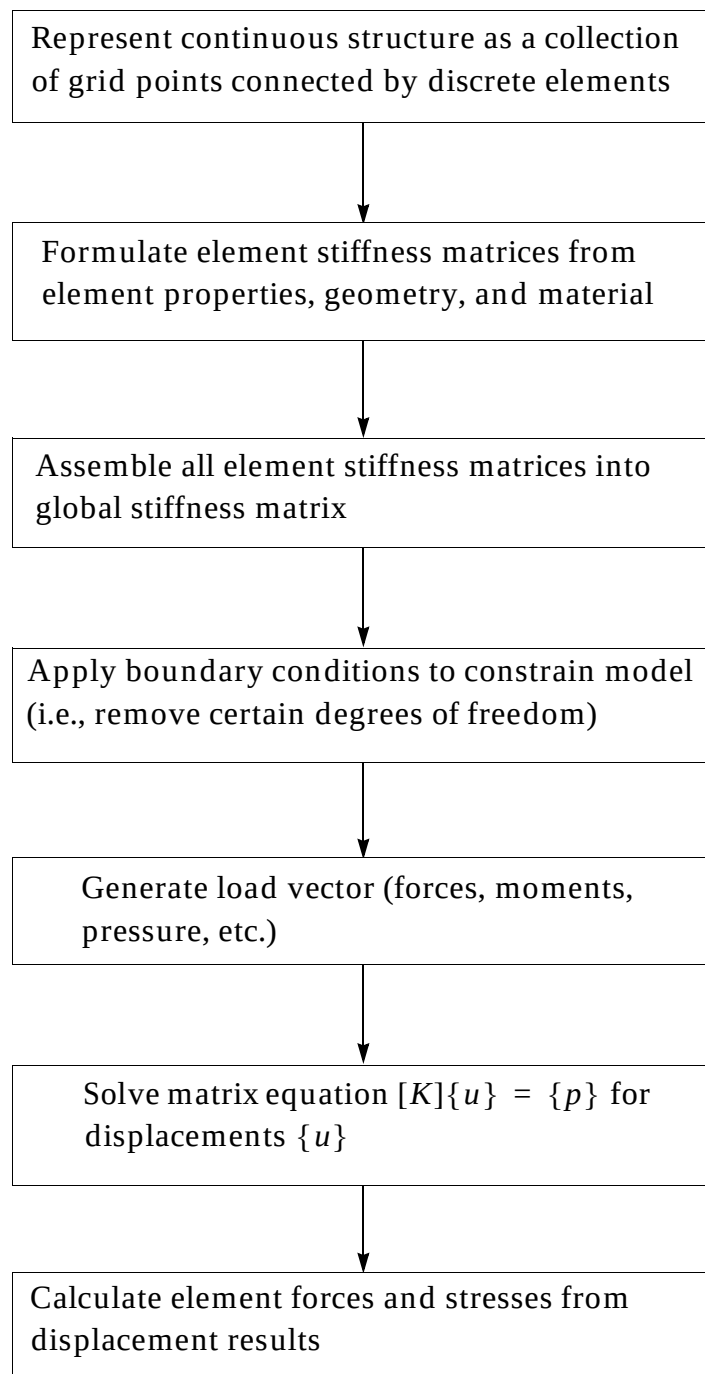


Figure 1-7 Linear Static Analysis in MSC.Nastran

To help understand the procedure described, consider the simple two-element structure shown in [Figure 1-8](#). The structure consists of two circular columns of equal length but different diameters. The lower member is fixed and a 10000 pound load is applied at the top so that both members are placed in compression. The goal is to determine the displacement at the ends of each member and the stresses within the members.

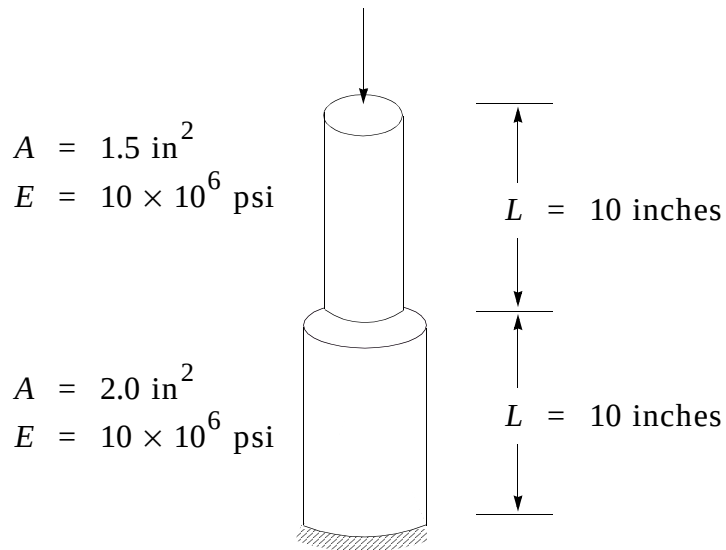


Figure 1-8 Two-Member Structure Under Axial Load

Following the procedure shown in [Figure 1-7](#):

1. Represent the structure as three grid points connected by two discrete elements.

This structure can be effectively modeled using two simple one-dimensional elements. For this example, one-dimensional elements with two degrees of freedom are sufficient. The simple element used in this example is similar to the MSC.Nastran CROD element described in the next chapter but without the torsional stiffness. Using the CBAR element for this example is acceptable; however, the element matrices are larger without providing any additional insight.

The first task involves assigning identification numbers to the elements (element IDs) and to the grid points (grid point IDs) as shown in [Figure 1-9](#).

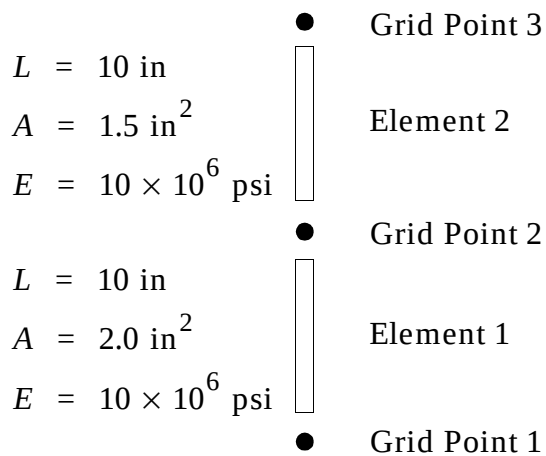


Figure 1-9 Exploded View of the Finite Element Model

2. Formulate the element stiffness matrices from the element properties, geometry, and material.

The element stiffness matrix for our one-dimensional element is given by [Eq. 1-2](#).

$$K_{el} = \begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \quad \text{Eq. 1-2}$$

Using the cross-sectional area A , Young's modulus E , and the length L for each element in [Eq. 1-2](#), the element stiffness matrices are given by “Using the cross-sectional area, Young's modulus, and the length for each element in Eq. 1-2, the element stiffness matrices are given by” on page 15

$$K_{el}^{(1)} = \begin{matrix} & 1 & & \\ & \begin{bmatrix} 2.0 & -2.0 \\ -2.0 & 2.0 \end{bmatrix} & & \\ & \begin{matrix} 1 & 2 \end{matrix} & & \end{matrix} \times 10^6 \text{ lb/in} \quad K_{el}^{(2)} = \begin{matrix} & 2 & & \\ & \begin{bmatrix} 1.5 & -1.5 \\ -1.5 & 1.5 \end{bmatrix} & & \\ & \begin{matrix} 2 & 3 \end{matrix} & & \end{matrix} \times 10^6 \text{ lb/in} \quad \text{Eq. 1-3}$$

3. Assemble the two stiffness matrices into a global stiffness matrix.

The two elemental stiffness matrices are assembled together by simply combining the matrices at the appropriate degrees of freedom associated with the rows and columns in the matrices as shown in [Eq. 1-4](#).

$$K_{global} = \begin{matrix} & 1 & & \\ & \begin{bmatrix} 2.0 & -2.0 & 0.0 \\ -2.0 & 2.0 & 0.0 \\ 0.0 & 0.0 & 0.0 \end{bmatrix} & & \\ & \begin{matrix} 1 & 2 & 3 \end{matrix} & & \end{matrix} \times 10^6 + \begin{matrix} & 1 & & \\ & \begin{bmatrix} 0.0 & 0.0 & 0.0 \\ 0.0 & 1.5 & -1.5 \\ 0.0 & -1.5 & 1.5 \end{bmatrix} & & \\ & \begin{matrix} 1 & 2 & 3 \end{matrix} & & \end{matrix} \times 10^6 \quad \text{Eq. 1-4}$$

$$= \begin{bmatrix} 2.0 & -2.0 & 0.0 \\ -2.0 & 3.5 & -1.5 \\ 0.0 & -1.5 & 1.5 \end{bmatrix} \times 10^6$$

4. Apply the fixed boundary condition to grid point 1, which is attached to ground.

Apply the fixed boundary condition by constraining grid point 1. This constraint is accomplished by partitioning row 1 and column 1 out of the assembled global stiffness matrix as shown in [Eq. 1-5](#). Partitioning the matrix is achieved by removing rows and columns from one matrix to create a smaller matrix.

$$K_{grid\ 1\ constrained} = \begin{matrix} & 1 & & \\ & \begin{bmatrix} 2.0 & -2.0 & 0.0 \\ -2.0 & 3.5 & -1.5 \\ 0.0 & -1.5 & 1.5 \end{bmatrix} & & \\ & \begin{matrix} 1 & 2 & 3 \end{matrix} & & \end{matrix} \times 10^6 \Rightarrow \begin{matrix} & 2 & & \\ & \begin{bmatrix} 3.5 & -1.5 \\ -1.5 & 1.5 \end{bmatrix} & & \\ & \begin{matrix} 3 \end{matrix} & & \end{matrix} \times 10^6 \quad \text{Eq. 1-5}$$

5. Apply the 10000 pound load to the model.

The total load on the structure consists of the 10000 pound acting at grid point 3 as shown in [Figure 1-8](#). Note that the reaction force at grid point 1 is not an applied load and is not included in the loading vector. The constrained degree of freedom (grid point 1) is partitioned out of the load vector in the same manner that it is partitioned out of the global stiffness matrix as shown in [Eq. 1-6](#).

$$\{f\} = \begin{matrix} 1 \\ 2 \\ 3 \end{matrix} \begin{Bmatrix} -0.0 \\ 0.0 \\ -10000. \end{Bmatrix} \Rightarrow \begin{matrix} 2 \\ 3 \end{matrix} \begin{Bmatrix} 0.0 \\ -10000. \end{Bmatrix} \quad \text{Eq. 1-6}$$

6. Solve the matrix equation.

The resulting matrix equation and solution is given by [Eq. 1-7](#).

$$\begin{bmatrix} 3.5 \times 10^6 & -1.5 \times 10^6 \\ -1.5 \times 10^6 & 1.5 \times 10^6 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0.0 \\ -10000. \end{Bmatrix}$$

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} -.005 \\ -.01167 \end{Bmatrix} \quad \text{Eq. 1-7}$$

Solving the matrix equation yields the displacement at grid points 2 and 3. Combining the displacement of grid point 1 (which is 0.0 because it is constrained) to the other grid points yields the complete displacement vector for the entire model as shown in [Eq. 1-8](#).

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0.0 \\ -.005 \\ -.01167 \end{Bmatrix} \quad \text{Eq. 1-8}$$

7. Calculate the element forces and stresses from the displacement of the grid points.

The axial stresses in the rods are computed using [Eq. 1-9](#).

$$\varepsilon_{element\ 1} = \frac{\Delta L}{L} = \frac{u_2 - u_1}{L} = \frac{-.005 - 0.0}{10} = -0.0005$$

$$\varepsilon_{element\ 2} = \frac{\Delta L}{L} = \frac{u_3 - u_2}{L} = \frac{-.01167 - 0.005}{10} = -0.0006667$$

$$\sigma_{element\ 1} = E\varepsilon_{element\ 1} = -5000 \text{ psi}$$

$$\sigma_{element\ 2} = E\varepsilon_{element\ 2} = -6667 \text{ psi}$$

Eq. 1-9

Therefore, the stresses are -5000 psi and -6667 psi for members 1 and 2, respectively.

Although this is a very simple model, it does demonstrate the approach followed by all static solutions. The matrix operations (i.e., matrix assembly, partitioning, etc.) performed in this example are included to show the internal processing in MSC.Nastran. These operations are performed automatically.

If you want to see the MSC.Nastran input file required to solve this problem, it is located on the delivery media with the name `intro.dat`.

Organization of MSC.Nastran Files

- Introduction to Organization of MSC.Nastran Files
- Overview of the MSC.Nastran Input Files
- Overview of the MSC.Nastran Output Files
- The Executive Control Section
- The Case Control Section
- The Bulk Data Section

2.1 Introduction to Organization of MSC.Nastran Files

This chapter describes the basic approach to analyzing a structure using MSC.Nastran. The example used throughout this chapter is a simple two-dimensional truss. Although the truss structure is relatively simple compared to the typical structures analyzed using MSC.Nastran, it is a structure that most engineers recognize.

The best way to learn a new feature is by applying it to an example. You are encouraged to copy the files presented in this guide into your working directories and experiment with them. Do not spend time entering the input files from the listings shown. For most of the examples in this guide, only a partial listing is provided. The partial listing serves two purposes: first, it keeps the examples down to a reasonable size; second, it is easier to highlight the important points for that particular example. The complete input file can be found on the delivery media.

2.2 Overview of the MSC.Nastran Input Files

The purpose of this section is provide an overview of the input; it is not meant to describe all of the options available for static analysis. The next section provides an overview of the output files. The goal of these two sections is to show how a typical MSC.Nastran input file is used to represent a typical model.

To perform an analysis using MSC.Nastran, you must generate an input file describing the structure's geometry, material properties, boundary conditions, and loads. 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 a text file (ASCII) 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 (MSC.Nastran is not an interactive program). Once the input file has been submitted, you have no additional interaction with MSC.Nastran until the job is complete. The only exception is to terminate the job prior to completion if it becomes necessary.

To illustrate the form of input and output files, you can use the simple truss structure shown in [Figure 2-1](#).

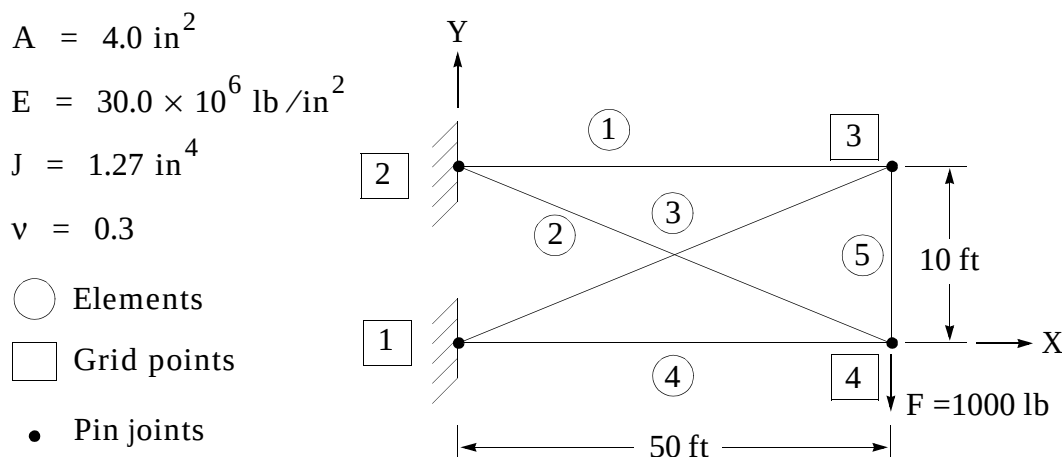


Figure 2-1 Truss Example

The truss structure consists of five members, each with a cross-sectional area (A) of 4 in^2 and a polar moment of inertia (J) of 1.27 in^4 . The material is steel with a Young's modulus E of $30 \times 10^6 \text{ psi}$ and a Poisson's ratio of 0.3 . All the connections between the members are pin joints (i.e., they can transmit forces but not moments). The goal of the analysis is to determine the displacement of the grid points and the stresses and forces within the members.

The MSC.Nastran input file listing for the truss structure is given in [Listing 2-1](#). Although the input file shown represents a fairly simple structure, it is typical of all MSC.Nastran input files. The purpose of the example is not to show how to perform an analysis or discuss the specifics of the model, such as the grid points or elements. Rather, the purpose

is to present the form of the input file itself. Discussion of the actual commands and entries in the input file begins in “[Grid Points, Scalar Points, and Coordinate Systems](#)” on page 47.

Listing 2-1 Truss Example Input File

<pre> \$ FILENAME - TRUSS1.DAT \$ ID LINEAR, TRUSS1 SOL 101 TIME 2 CEND TITLE = LINEAR STATICS USER'S SAMPLE INPUT FILE SUBTITLE = TRUSS STRUCTURE LABEL = POINT LOAD AT GRID POINT 4 LOAD = 10 SPC = 11 DISPLACEMENT = ALL ELFORCE = ALL ELSTRESS = ALL BEGIN BULK \$ \$ THE GRID POINTS LOCATIONS \$ DESCRIBE THE GEOMETRY \$ GRID 1 0. 0. 0. 3456 GRID 2 0. 120. 0. 3456 GRID 3 600. 120. 0. 3456 GRID 4 600. 0. 0. 3456 \$ \$ MEMBERS ARE MODELED USING \$ ROD ELEMENTS \$ CROD 1 21 2 3 CROD 2 21 2 4 CROD 3 21 1 3 CROD 4 21 1 4 CROD 5 21 3 4 \$ \$ PROPERTIES OF ROD ELEMENTS \$ PROD 21 22 4. 1.27 \$ \$ MATERIAL PROPERTIES \$ MAT1 22 30.E6 .3 \$ \$ POINT LOAD \$ FORCE 10 4 1000. 0. -1. 0. \$ SPC1 11 123456 1 2 \$ ENDDATA </pre>	} Executive Control Section } Case Control Section } Bulk Data Section

An MSC.Nastran input file consists of five distinct sections: the NASTRAN statement, the File Management Section (FMS), the Executive Control Section, the Case Control Section, and the Bulk Data Section. Input in the File Management Section and the Executive Control Section is referred to as a statement; input in the Case Control Section is referred to as a command; and input in the Bulk Data Section is referred to as an entry.

Descriptions of these sections follow.

The NASTRAN Statement

The NASTRAN statement is an optional statement used to specify system parameters that control internal solution processing or provide specific diagnostics. The NASTRAN statement, if used, must be placed before the Executive Control Statement. The NASTRAN statement is generally not required, but it is useful when you need to specify system-related parameters, such as setting the BUFFSIZE for the I/O. For a description of the NASTRAN statement options, refer to the “[nastran Command and NASTRAN Statement](#)” on page 1 of the *MSC.Nastran Quick Reference Guide*. Some of the NASTRAN statement options are discussed in “[Database Management](#)” on page 541.

File Management Section

The File Management Section (FMS), if needed, is located before the Executive Control Section. The purpose of the FMS is to assign files, manipulate the database, and perform restarts. It is commonly used when solving larger problems. “[Restarts](#)” on page 521 and “[Database Management](#)” on page 541 discuss the use of the File Management Section. Since the truss model shown in [Listing 2-1](#) is small and the desired analysis is not a restart, the use of the FMS is not needed.

Executive Control Section

The Executive Control Section contains statements that

- Select the solution sequence to be run (e.g., statics, normal modes).
- Set the maximum MSC.Nastran (not system) time permitted for the run.
- Request various diagnostic outputs.
- Contain user-written DMAP. DMAP is an advanced topic and is beyond the scope of this user’s guide.

For the truss example, the Executive Control Section specifies static solution SOL 101 and a TIME statement that specifies a maximum of 2 CPU minutes for the run. The end of the Executive Section is denoted by the CEND statement. The ID statement is an optional statement to help document your input file. If used, it can be located anywhere in the Executive Control Section.

Case Control Section

Immediately following the CEND statement is the Case Control Section. Case Control commands are used to

- Define subcases (e.g., loading and boundary conditions).
- Make selections from the Bulk Data Section (e.g., loads, constraints, etc.).
- Specify output requests.
- Define titles, subtitles and labels for documenting the analysis.

For the truss example, a title, subtitle, and label are defined. These labels are printed on each page of the .f06 output file. The `LOAD = 10` command instructs MSC.Nastran to apply the loading defined by the `FORCE` entry with an ID of 10 in the Bulk Data Section. The `SPC = 11` command instructs MSC.Nastran to apply the constraints defined by the `SPC1` entry with an ID of 11 in the Bulk Data Section. Printed displacements for all the grid points and the forces and stresses within each member are requested with the `DISPLACEMENT = ALL`, `FORCE = ALL`, and `STRESS = ALL` commands. By default, these output requests are printed to the .f06 output file, which is discussed in “[Overview of the MSC.Nastran Input Files](#)” on page 21. The end of the Case Control Section is denoted by the `BEGIN BULK` delimiter.

Bulk Data Section

The Bulk Data Section is used to define the analysis via the grid points, elements, constraints, and loads. The start of the Bulk Data Section is denoted by the `BEGIN BULK` delimiter. The last entry in Bulk Data Section must be an `ENDDATA` delimiter. The `ENDDATA` delimiter also signifies the end of the MSC.Nastran input file. If any of these required delimiters are not present, a fatal error message results when the job is submitted.

Most of the entries in the Bulk Data Section do not need to be called out by a Case Control command in order to be included in the model. In the truss example, the grid point and element entries are included in the model no matter what the Case Control Section specifies.

However, to provide flexibility to the user, load and constraint entries are selected in the Case Control Section. This way, you can control which load and constraint entries specified in the Bulk Data Section are actually used in a given analysis. In other words, the load and constraint entries in the Bulk Data Section are included in the analysis only if they are specified in the Case Control Section. Having loads and constraint entries in the Bulk Data Section that are not specified in the Case Control Section is allowed, but these entries are not used in the analysis and hence do not affect the results.

The truss model consists of four grid points that represent the joints of the structure. The locations of the grid points for this example are given in what is known as the basic coordinate system, which is the default coordinate system in MSC.Nastran. In this example, the origin is located at grid point 1. You should note that the dimensions given for the truss structure in [Listing 2-1](#) are given in feet, but the dimensions used for the grid point locations in the input file are entered in inches. For consistency, the units for the cross-sectional area, torsional stiffness, and Young’s modulus are also entered in inches. It is the user’s responsibility to ensure that the units for all of the input are consistent.

[Figure 2-2](#) summarizes the linking between the Case Control commands and the Bulk Data entries.

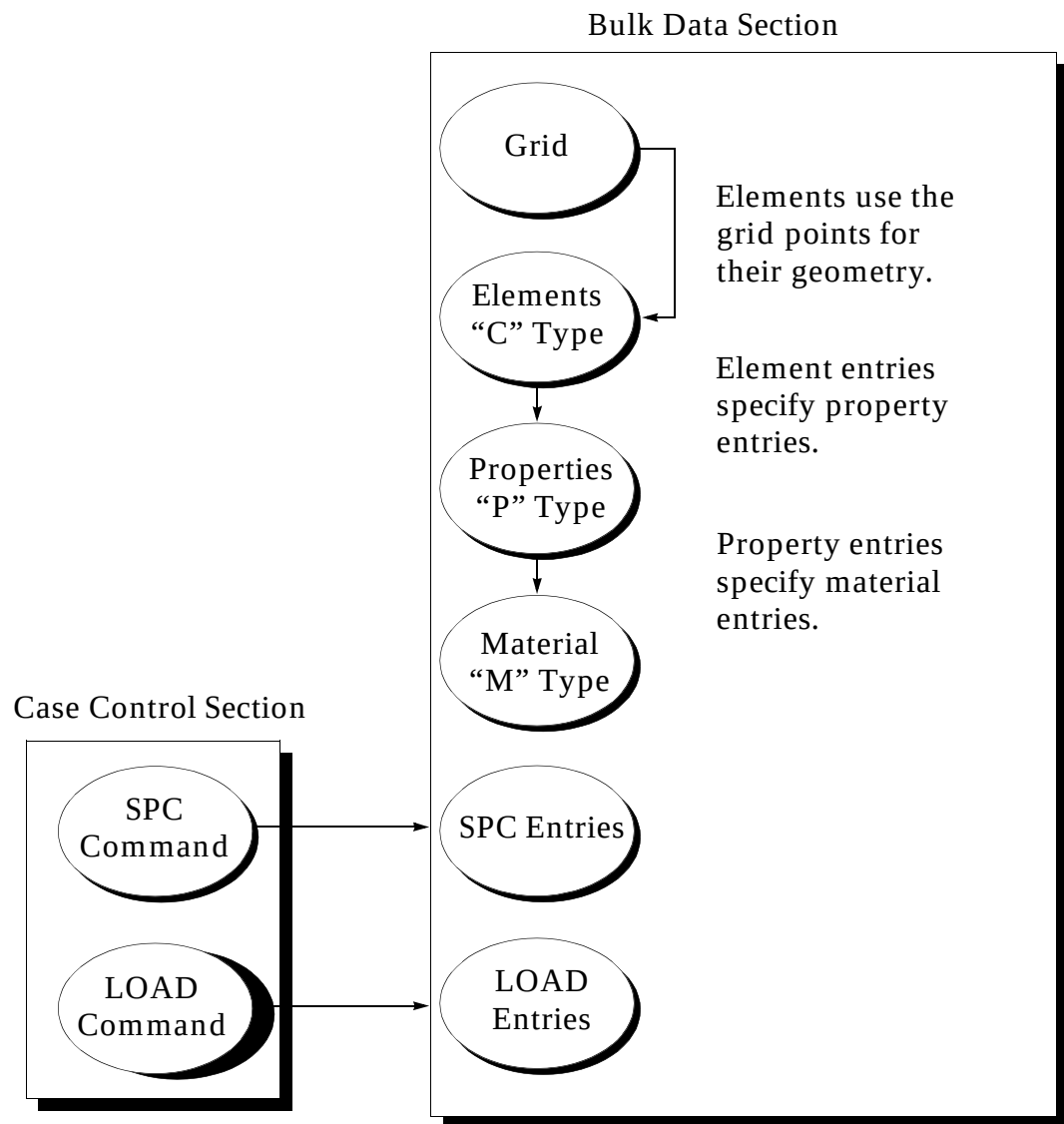


Figure 2-2 MSC.Nastran Input File Structure

Returning to our truss example, the members of the truss are modeled using rod elements. The MSC.Nastran notation for a rod is the CROD Bulk Data entry. The "C" preceding ROD denotes that the entry is for an "element connection", i.e., it connects grid points. This pattern is true for all elements, for example, the Bulk Data entry notation for a element connection of a QUAD4 plate element is CQUAD4. This notation also makes it convenient when trying to find a description of a particular element in the *MSC.Nastran Quick Reference Guide* since they are all ordered alphabetically under the C's.

All elements in the input file must have a unique ID with respect to all the other elements. The IDs of the CROD elements in the truss example are 1, 2, 3, 4, and 5.

Each of the CROD entries refers to the PROD entry 21 (Field 3), which is the property definition of the CROD elements. The PROD entry specifies a material ID of 22 (field 3), the cross-sectional area of 4.0 in^2 , and the torsional stiffness coefficient of 1.27 in^4 . All property entries begin with "P".

The material ID of 22 refers to a MAT1 entry. The MAT1 entry defines an isotropic material. The MAT1 entry is defined formally in “[Material Properties and Composites](#)” on page 177; but for now, it is sufficient to say that an isotropic material requires only two constants to fully describe the material. In this case the material property is described as having a Young’s modulus of 30×10^6 psi and a Poisson’s ratio of 0.3. All material entries start with “M”.

The FORCE and SPC entries, as mentioned previously, are included in the analysis because they were called out in the Case Control Section by their ID number. The FORCE entry in this example specifies a point load of 1000 pounds acting at grid point 4 in the -Y direction. The SPC1 entry specifies that all six degrees of freedom are constrained at grid points 1 and 2.

If you have not performed an MSC.Nastran run before, you are encouraged to copy the truss model “truss1.dat” into your working directory and submit it as an MSC.Nastran job.

2.3 Overview of the MSC.Nastran Output Files

The purpose of this section is to introduce the output file generated by a typical run. If you are new to MSC.Nastran, this section may seem somewhat overwhelming. Like the previous section, however, this section is only meant to provide an overview-the details are covered in the remaining sections and chapters.

After an input file is submitted for execution, several output files may be generated. provides summary descriptions of all the potential files generated by MSC.Nastran (the input file is named 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 2-1 Files Resulting from Submitting the Job Named run1.dat

run1.dat	The input file describing the model, the type of solution, the output requests, etc. Generated with a text editor or preprocessor.
run1.f06	The main output file containing the printed output such as displacements, element forces, stresses, etc.
run1.f04	A history of the assigned files, disk space usage, and modules used during the analysis. Useful for debugging.
run1.log	A summary of the command line options used and the execution links.
run1.DBALL	A database containing the input file, assembled matrices, and solutions. Used also for restarting the run for additional analysis or output.
run1.MASTER	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.
run1.plt	Contains the plot information requested by the NASTRAN plotter commands specified in input file.
run1.pch	Contains the punch output as requested in the input file.
run1.op2, run1.xdb	Graphics database used by MSC.Patran for postprocessing of the results.
miscellaneous scratch files	Several scratch files are generated during the analysis which MSC.Nastran will automatically delete upon completion of the run.

The main output is located in the .f06 file and the .f04 and .log files are useful for debugging your model should problems occur.

In reference to , the files run1.dat, run1.f06, run1.f04, run1.log, and run1.pch are ASCII files and can be viewed using any text editor. The remaining files are binary and therefore cannot be viewed. The binary files are not intended to be used directly; they are to be used for additional analysis, such as restarts or postprocessing. If no restarts are planned for the job run1, you can delete run1.DBALL and can run1.MASTER to save disk space.

The file run1.plt is a binary file that contains the plotting information generated by the MSC.Nastran plotter. If the MSC.Nastran plotter is not used, run1.plt is deleted following the completion of the run. Likewise, if punch output is specified, the file run1.pch is retained after the run is complete; otherwise, it is deleted once the run is complete. The binary file run1.op2 is the graphics database used by MSC.Patran graphics pre- and postprocessors. It is requested by a PARAM,POST entry in the Bulk Data Section.

If no restarts are planned, you may specify “scr = yes” on the command line when submitting the input file for execution. The .DBALL and .MASTER files are placed on the scratch directory and are automatically deleted upon completion of the run. The command line options are discussed in **“Database Management”** on page 541, and **References 16. and 21.**

For the five-member truss, the input file name is truss1.dat. The resulting output files are truss1.f06, truss1.f04, truss1.log, truss1.DBALL, and truss1.MASTER.

If the file truss1.dat is submitted a second time as a cold start (i.e., no restart commands are used), you must delete the files .DBALL and .MASTER before the run is submitted the second time. Also, if an .op2 file already exists for the run, it too should be deleted prior to resubmitting the run.

The output file that you will use most frequently is the .f06 file; therefore, it is useful to discuss a typical one before discussing the details of the input file. A complete listing of the truss1.f06 output file is shown in **Figure 2-3**. This is one of the few occasions that an output file is shown in its entirety; only selected portions of the output will be shown in the remaining examples in this user’s guide. The .f06 suffix is used because the program is writing this file to FORTRAN unit 6 (this piece of trivia makes no difference regarding how you use MSC.Nastran, but many users are curious about where the names come from).

By default, the output file contains an echo (listing) of the input file with the Bulk Data sorted alphabetically as shown in **Figure 2-3**.

```

ID LINEAR,TRUSS1
SOL 101
TIME 2
CEND
LINEAR STATICS USER'S GUIDE                                JUNE 18, 1996 MSC/NASTRAN 6/13/96 PAGE 3
TRUSS STRUCTURE

POINT LOAD AT GRID POINT 4

C A S E   C O N T R O L   D E C K   E C H O

CARD
COUNT
1      TITLE = LINEAR STATICS USER'S GUIDE
2      SUBTITLE = TRUSS STRUCTURE
3      LABEL = POINT LOAD AT GRID POINT 4
4      LOAD = 10
5      SPC = 11
6      DISPLACEMENT = ALL
7      ELFORCE = ALL
8      ELSTRESS = ALL
9      BEGIN BULK
      INPUT BULK DATA CARD COUNT =      33

LINEAR STATICS USER'S GUIDE                                JUNE 18, 1996 MSC/NASTRAN 6/13/96 PAGE 4
TRUSS STRUCTURE

POINT LOAD AT GRID POINT 4

S O R T E D   B U L K   D A T A   E C H O

CARD
COUNT      .   1   ..   2   ..   3   ..   4   ..   5   ..   6   ..   7   ..   8   ..   9   ..  10   .
1-      CROD    1      21      2      3
2-      CROD    2      21      2      4
3-      CROD    3      21      1      3
4-      CROD    4      21      1      4
5-      CROD    5      21      3      4
6-      FORCE   10      4      1000.  0.      -1.      0.
7-      GRID    1      0.      0.      0.      3456
8-      GRID    2      0.      120.  0.      3456
9-      GRID    3      600.  120.  0.      3456
10-     GRID    4      600.  0.      0.      3456
11-     MAT1   22      30.E6      .3
12-     PROD    21      22      4.      1.27
13-     SPC1   11      123456  1      2
      ENDDATA
      TOTAL COUNT=      14

LINEAR STATICS USER'S GUIDE                                JUNE 18, 1996 MSC/NASTRAN 6/13/96 PAGE 5
TRUSS STRUCTURE

POINT LOAD AT GRID POINT 4
USER INFORMATION MESSAGE
ORIGIN OF ASSEMBLY BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.

RESULTANTS ABOUT ORIGIN OF ASSEMBLY BASIC COORDINATE SYSTEM IN ASSEMBLY BASIC SYSTEM COORDINATES.

      OLOAD   RESULTANT
      T1      T2      T3      R1      R2      R3
1  0.0000000E+00 -1.0000000E+03  0.0000000E+00  0.0000000E+00  0.0000000E+00 -6.0000000E+05

LINEAR STATICS USER'S GUIDE                                JUNE 18, 1996 MSC/NASTRAN 6/13/96 PAGE 6
TRUSS STRUCTURE

POINT LOAD AT GRID POINT 4

*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS

1      -5.2155330E-15      6.4518387E+01

```

Figure 2-3 truss1.f06 Output File.

POINT LOAD AT GRID POINT 4

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
2	G	.0	.0	.0	.0	.0	.0
3	G	1.247578E-02	-1.285377E-01	.0	.0	.0	.0
4	G	-1.252422E-02	-1.290368E-01	.0	.0	.0	.0

POINT LOAD AT GRID POINT 4

F O R C E S I N R O D E L E M E N T S (C R O D)

ELEMENT ID.	AXIAL FORCE	TORQUE	ELEMENT ID.	AXIAL FORCE	TORQUE
1	2.495156E+03	.0	2	2.554449E+03	.0
3	-2.544570E+03	.0	4	-2.504844E+03	.0
5	4.990313E+02	.0			

POINT LOAD AT GRID POINT 4

POINT LOAD AT GRID POINT 4

S T R E S S E S I N R O D E L E M E N T S (C R O D)

ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN
1	6.237891E+02		.0		2	6.386123E+02		.0	
3	-6.361426E+02		.0		4	-6.262109E+02		.0	
5	1.247578E+02		.0						

POINT LOAD AT GRID POINT 4

Figure 2-3 truss1.f06 Output File (Cont.)

After the input file echo, the “OLOAD RESULTANT” is printed which shows the resultant of the loads about the origin of the basic coordinate system in this case. Next is the USER INFORMATION MESSAGE (UIM) 5293 which shows the epsilon and external work for the run. The epsilon is a measure of the numeric conditioning of the model, while the external work is the work due to the applied loads. Both of these values are useful for model checkout and are discussed in more detail in [“Model Verification”](#) on page 369.

In the truss example, output is requested for displacements, element forces, and element stresses. The displacement request is the only grid point output. Element forces and stresses are element output. The last summary is the maximum of the applied loads. After the summaries, the displacement for all of the grid points, the element forces, and the element stresses is printed as requested in the Case Control Section.

For static analysis of the truss model, just one applied load is used. You can, however, specify multiple loading conditions to be analyzed in a single input file if desired. If you do specify multiple loading conditions, the OLOAD RESULTANT and EXTERNAL WORK

are shown for each selected loading conditions. The external work is calculated as $1/2\{P_I\}^T[U_I]$. In the case of enforced displacement, the equivalent load is calculated as the constraint force required to impose the enforced displacement. The external work print-out in this case is meaningless.

The information in the output file is extremely important for model checkout and debugging. Make a habit of reviewing these diagnostics for all of your runs. Experiment with the truss1.dat file; run the input file and view truss1.f06 in your text editor.

Note: The file is 132 columns wide

After this overview of the MSC.Nastran input and output files, the next three sections contain the Executive Control Section, Case Control Section, and Bulk Data Section. The File Management Section is discussed in “[Database Management](#)” on page 541.

2.4 The Executive Control Section

The purpose of the Executive Control Section is to select the solution sequence for the desired analysis, specify the maximum CPU time allowed for the job, and request various diagnostics. Also, it is the location for alters and user-written DMAP (i.e., Direct Matrix Abstraction Program, a matrix manipulation language used to modify the normal solution processing). Most of the commonly used statements are order independent. The statements for which the order is critical are those used for user DMAP and alters and the CEND statement. User DMAP and alters are advanced topics and are normally not used for most analyses.

The Executive Control statements that are commonly used for linear static analysis are described in [Table 2-2](#).

Table 2-2 Commonly Used Executive Control Statements

CEND	Designates the end of the Executive Control Section.
DIAG	Requests diagnostic output or modifies operational parameters.
ECHOON ECHOOFF	Controls the echo of the Executive Control statements.
ID	Optional identifier.
SOL	Requests the solution to be performed.
TIME	Sets the maximum allowable execution.

The format of the Executive Control statements is free-field format between columns 1-72. The statements may begin in any column (except for comments). The operands--that is, the input data following the name--are separated from the name using a comma and / or one or more blanks. If more than one operand is needed, they are also separated by a comma and / or one or more blanks.

Comments may be placed anywhere in the input file and are denoted by placing a “\$” at the start of the comment. If an input line is to be a comment only, it must start in column 1. Comments are ignored by MSC.Nastran and have no affect on the results. You should use comments generously; they greatly improve the readability of the input file.

The ID statement is used to identify the job; this statement is optional. Only one ID statement may be used, and it can be located anywhere in the Executive Control Section. The format of the ID statement is the keyword “ID” followed by a delimiter and a comment. The comment may be any ASCII characters. A typical ID statement is as follows:

```
ID THIS IS RUN 2
```

The SOL statement specifies the desired solution sequence to be used. To invoke a particular analysis type, the SOL name is followed by the ID number of the solution sequence. The solution sequences are classified into two different categories: rigid formats and structured solution sequences. For example, linear static analysis--the main subject of

this guide--can be performed using the rigid format SOL 1 or the Structured Solution Sequence, SOL 101. Although each of the statics solutions works, it is highly desirable to use the Structured Solution Sequence, SOL 101.

The Structured Solution Sequences are the latest generation of solution sequences with most of the commonly used features built-in. The Structured Solution Sequences are numbered in the range of 100 through 200. All of the examples in this guide use the Structured Solution Sequences.

An alternative method of specifying a solution is to use the SOL keyword followed by the name of the solution sequence. For example, SOL SESTATIC is equivalent to SOL 101. Using the solution numbers is generally recommended rather than their names because the solution numbers are more easily recognized by most veteran MSC.Nastran users.

The ECHOON and ECHOOFF statements are used to control the printing of the Executive Control Section. Since the Executive Control Section is generally quite small, it is recommended that it be echoed (the default choice). If the Executive Control is not printed, readability is hindered, especially if you have to come back to the output file several months after the original run was made. Since ECHOON is the default, this statement is not widely used. However, one useful application for the ECHOOFF statement is security. By using ECHOOFF, portions of the Executive Control Section that you may consider proprietary (such as user-written DMAP) may be eliminated from the output file. Multiple ECHOON and ECHOOFF statements may be used.

The TIME statement is optional for most MSC.Nastran runs. The purpose of the TIME statement is to set the maximum CPU time that MSC.Nastran runs before the job is terminated. This value should not be confused with a system time limit imposed by the system administrator. It is a good practice to set the MSC.Nastran time limit to a value less than the system time limit for the reasons described in the next paragraph.

If the system time limit is exceeded during execution, the system aborts the job, which can potentially corrupt the database. Once the database is corrupted, it is difficult, if not impossible, to restart the job to take advantage of the work done up to the point of the abort. On the other hand, if the required execution time exceeds the time specified on the MSC.Nastran TIME statement, the job terminates cleanly and permits restarts if desired. Also, MSC.Nastran estimates the CPU time required to perform certain operations during execution. If the time remaining is not sufficient to complete a particular operation, the job terminates without wasting any additional computer resources. The default value for TIME is infinity.

The DIAG statement is used to request additional diagnostics or output and / or to modify the solution.

The Executive Control Section for the truss example ([Listing 2-1](#)) identifies the job, requests the static Structured Solution Sequence 101, and sets the maximum time limit for the job to two minutes. For more details, see [“Executive Control Statement Descriptions”](#) on page 102 of the *MSC.Nastran Quick Reference Guide*.

2.5 The Case Control Section

The Case Control Section is used to document the run, make selections from the Bulk Data Section, request output, and define subcase structure. Subcase structure provides the means to perform an analysis with different combinations of loads and boundary conditions within a single run. To denote a subcase, the SUBCASE command is used together with an ID operand that identifies the subcase number. Requests may be made above the first subcase or within any of the subcases. Those requests made above the first subcase are the defaults for all of the subcases. These defaults apply to all of the subcases unless they are specifically overridden within the subcase. If a SUBCASE command is not present in the Case Control Section, as is the case for the truss example described in “[Overview of the MSC.Nastran Output Files](#)” on page 27, a single subcase with ID 1 is assumed.

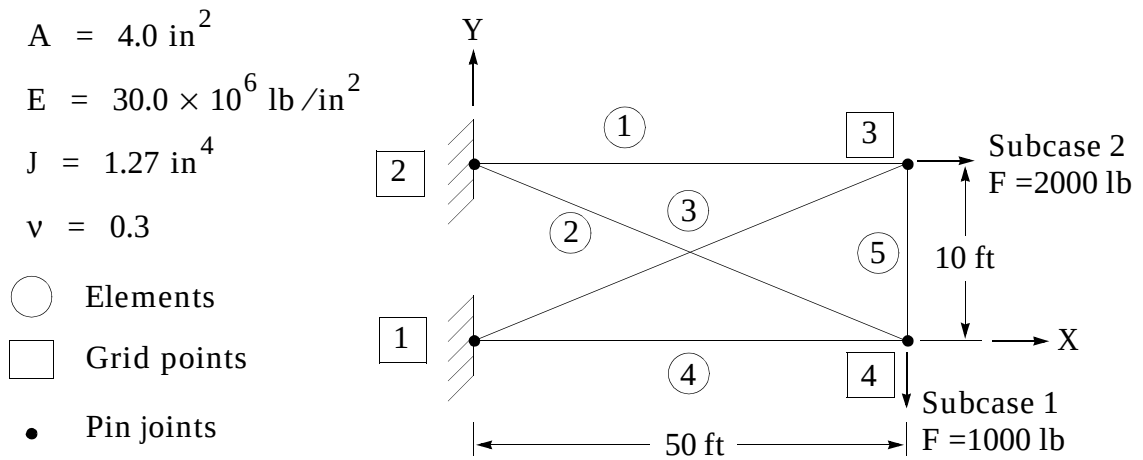


Figure 2-4 Truss Example with Two Loading Conditions.

As an example, a second load case is added to the previous truss structure as shown in [Figure 2-4](#). The two separate analyses are performed during a single run. The first subcase consists of the 1000-pound force acting at grid point 4 in the negative Y-direction. The second subcase consists of the 2000-pound force acting at grid point 3 in the X-direction. The displacements are requested for all the grid points for both load cases, but only the element forces are desired for the second load case.

An input file meeting these requirements is shown in [Listing 2-2](#).

Listing 2-2 Truss Input File with Two Loadings

```

$
$ Filename - TRUSS2.DAT
$
ID LINEAR,TRUSS2
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE - TWO SUBCASES
SUBTITLE = TRUSS STRUCTURE
SPC = 11
DISPLACEMENT = ALL
$
SUBCASE 1
  LABEL = POINT LOAD AT GRID POINT 4
  LOAD = 10
$
SUBCASE 2
  LABEL = POINT LOAD AT GRID POINT 3
  LOAD = 11
  ELFORCE = ALL
$
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS
$ DESCRIBE THE GEOMETRY
$
GRID    1           0.      0.      0.      3456
GRID    2           0.     120.     0.      3456
GRID    3          600.     120.     0.      3456
GRID    4          600.      0.      0.      3456
$
$ MEMBERS ARE MODELED USING
$ ROD ELEMENTS
$
CROD    1          21       2       3
CROD    2          21       2       4
CROD    3          21       1       3
CROD    4          21       1       4
CROD    5          21       3       4
$
$ PROPERTIES  OF ROD ELEMENTS
$
PROD    21         22       4.      1.27
$
$ MATERIAL PROPERTIES
$
MAT1    22         30.E6           .3
$
SPC1    11         123456  1       2
$
$ POINT LOAD SUBCASE 1
FORCE   10         4           1000.  0.    -1.    0.
$
$
$ POINT LOAD SUBCASE 2
FORCE   11         3           2000.  1.    0.    0.
$
ENDDATA

```

The Case Control Section consists of two subcases identified with ID numbers 1 and 2. The choice of the ID numbers is arbitrary, but they must be strictly in ascending order. For example, if the first subcase is 10, the ID number of the second subcase can be any integer number greater than 10.

Any title information, output requests, or Bulk Data selections made prior to the first subcase are the defaults for all the subcases unless they are overridden by a request within a specific subcase. For the Case Control Section shown in [Listing 2-2](#), a displacement request is placed above the first subcase but not within either of the two subcases. Therefore, the displacements are output for all of the grid points for both subcases. Since an element force request appears only in Subcase 2, no element force output is generated for Subcase 1. The element forces are output for all elements for the loading condition defined in Subcase 2.

It is often desirable to limit the output to a select number of grid points or elements; this is accomplished with the use of the SET command. The SET command identifies a group of grid points and/or elements as a specific set ID. For example, suppose you want to output all of the grid point displacements for Subcase 1 and the grid point displacements for grid points 3 and 4 for Subcase 2. In addition, you want the element force output for element 3 for Subcase 1 and elements 3 and 4 for Subcase 2. A Case Control Section that meets these requirements is shown in [Listing 2-3](#).

Listing 2-3 Controlling Output with Set Commands

```
$
$ FILENAME - TRUSS3.DAT
$
ID LINEAR,TRUSS3
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE - SET EXAMPLE
SUBTITLE = TRUSS STRUCTURE
LOAD = 10
SPC = 11
DISPLACEMENT = ALL
SET 1 = 3,4
SET 2 = 3
$
SUBCASE 1
  LABEL = POINT LOAD AT GRID POINT 4  LOAD = 10
  ELFORCE = 2
$
SUBCASE 2
  LABEL = POINT LOAD AT GRID POINT 3
  LOAD = 11
  DISPLACEMENT =1
  ELFORCE = 1
BEGIN BULK
```

For large models, you may need to further partition the results in the output files in a certain manner or repeat the same output in different subcases. The REPCASE command is designed for this purpose. The REPCASE command provides a method of organizing the output based on the element and/or grid point ID numbers, which is convenient for large models or when more than one engineer is reviewing the results.

As an example, again consider the truss example shown in [Figure 2-2](#). Suppose you want to separate the results into three separate sections in the .f06 output file. The input file is shown in [Listing 2-4](#). In this example, the Repcase 2 command provides for additional output from the solution generated in Subcase 1 without performing any additional analysis. For the Case Control shown in [Listing 2-4](#), Subcase 1 output consists of the displacement for all of the grid points and the element forces for elements 1 and 2. The output for Repcase 2 consists of the element force for elements 3 through 5. Note that the default output request for the grid point displacements is overridden by the `DISPLACEMENT = NONE` command. Subcase 3 is a new loading condition with its own output requests. A second static solution is performed for this subcase. Also note that only two static solutions are performed for this input file.

Listing 2-4 Truss Input File with the REPCASE Command

```
$
$ FILENAME TRUSS4.DAT
$
ID LINEAR,TRUSS4
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE - REPCASE EXAMPLE
SUBTITLE = TRUSS STRUCTURE
LOAD = 10
SPC = 11
DISPLACEMENT = ALL
SET 1 = 3,4
SET 2 = 1,2
SET 3 = 3 THRU 5
$
SUBCASE 1
  LABEL = POINT LOAD AT GRID POINT 4
  LOAD = 10
  ELFORCE = 2
$
REPCASE 2
  LABEL = POINT LOAD AT GRID POINT 4
  ELFORCE = 3
  DISPLACEMENT = NONE
$
SUBCASE 3
  LABEL = POINT LOAD AT GRID POINT 3
  LOAD = 11
  DISPLACEMENT =1
  ELFORCE = 1
BEGIN BULK
```

The SUBCOM command is used to generate linear combinations of the previous subcases. Returning to the truss structure shown in [Figure 2-2](#), the two load cases were previously treated as separate loads, each with their own subcase. Now, suppose a third load case is desired that is a linear combination of the first two subcases. The input file shown in [Listing 2-5](#) demonstrates this feature.

Listing 2-5 SUBCOM Input File

```
$ FILENAME - TRUSS5.DAT
$
ID LINEAR, TRUSS5
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE SUBCOM EXAMPLE
SUBTITLE = TRUSS STRUCTURE
LOAD = 10
SPC = 11
DISPLACEMENT = ALL
$
SUBCASE 1
  LABEL = POINT LOAD AT GRID POINT 4
  LOAD = 10
$
SUBCASE 2
  LABEL = POINT LOAD AT GRID POINT 3
  LOAD = 11
  ELFORCE = ALL
$
SUBCOM 3
  SUBSEQ 1.0, 0.5
  ELFORCE = ALL
$
SUBCOM 4
  SUBSEQ 1.0, -0.5
  ELFORCE = ALL
$
BEGIN BULK
```

For the Case Control Section shown in [Listing 2-5](#), the SUBCOM 3 output is as follows:

$$\begin{Bmatrix} SUBCOM\ 3 \\ OUTPUT \end{Bmatrix} = 1.0 \begin{Bmatrix} SUBCASE\ 1 \\ OUTPUT \end{Bmatrix} + 0.5 \begin{Bmatrix} SUBCASE\ 2 \\ OUTPUT \end{Bmatrix}$$

The output for SUBCOM 4 is as follows:

$$\begin{Bmatrix} SUBCOM\ 4 \\ OUTPUT \end{Bmatrix} = 1.0 \begin{Bmatrix} SUBCASE\ 1 \\ OUTPUT \end{Bmatrix} - 0.5 \begin{Bmatrix} SUBCASE\ 2 \\ OUTPUT \end{Bmatrix}$$

The coefficients 1.0, and 0.5 for Subcom 3 are input on a SUBSEQ command. These coefficients used in the SUBSEQ command refer to the previous Subcases 1 and 2. The 1.0 value in Subcom 3 is the coefficient corresponding to Subcase 1 and the 0.5 is the coefficient associated with Subcase 2. Subcom 3 is not considered a subcase. Likewise, the coefficients 1.0 and -0.5 for Subcom 4 refer to Subcases 1 and 2. If you specify a third SUBSEQ coefficient for either subcom, the job will fail.

SUBCOM commands may be used to combine any number of previous subcases; however, there must be a coefficient defined on the SUBSEQ command for each subcase. If you wish to exclude a subcase from the summation, set the coefficient to 0.0 for that particular subcase.

As a final note, you can define the combined loads in their own subcases and not use subcoms. If your goal is only to generate the output of the the combined loads (so you only need two subcases), this is the preferred way. However, using subcoms is desirable if you want to review the structural response to the individual loads in addition to the combined load. The SUBCOM command sums up the results of the previous subcases multiplied by the appropriate scale factors. One of the best applications of the SUBCOM command is combining the results generated with different constraints.

The ECHO command in the Case Control Section is used to control the printing of the Bulk Data Section. There are five options for this command:

SORT	Prints the Bulk Data Section in sorted form; free-field format is converted to small-field format and is sorted alphabetically. This is the default.
UNSORT	Prints the Bulk Data Section exactly as it is input.
BOTH	Prints the Bulk Data Section in the sorted and unsorted format.
PUNCH	Punches the Bulk Data to an ASCII file.
NONE	Suppresses printing of the Bulk Data.

For example, to echo the Bulk Data in the unsorted form, you use the following ECHO command:

```
ECHO = UNSORT
```

The output requests, such as DISPLACEMENT, FORCE, STRESS, etc., are needed whenever data recovery quantities are to be computed, even if they are not printed. Such is the case when you use a postprocessor to view the results. Typical output requests are as follows:

```
SET 1 = 5, 6, 7
SET 3 = 1, 5, 9
STRESS = ALL
DISP(PLOT) = 1 $ WHERE 1 IS THE ID OF A SET OF GRID POINTS
ELFORCE(PUNCH) = 3 $ WHERE 3 IS THE ID OF SET OF ELEMENTS
```

The PLOT option suppresses the printed output, but the data recovery is still performed. This option is useful for large models where the quantity of printed output would be excessive, but the data recovery is still required for postprocessing. The PUNCH option writes the results to an ASCII file instead of the .f06 file. This option makes it convenient for exporting the results to other programs.

A complete description of all of the Case Control commands is included in the *[MSC.Nastran Quick Reference Guide](#)*.

2.6 The Bulk Data Section

The Bulk Data Section is the primary method of describing the structural model and is usually the majority of the input file. The order of Bulk Data entries is not critical (continuations without continuation identifiers are an exception to this rule and are discussed later). All of the Bulk Data entries are sorted alphabetically prior to the start of execution and are printed in the sorted order by default. For very large problems, this sorting process can be CPU intensive. If you anticipate that the same input file will be submitted several times, it is advantageous to sort the Bulk Data once and use the sorted input file. A sorted Bulk Data file can be generated using the ECHO = PUNCH command in the Case Control Section (note: comments in the Bulk Data Section are lost). Alternatively, the sorted Bulk Data is stored on the database and is available using the Restart feature. Restarts are discussed in “[Restarts](#)” on page 521.

Bulk Data entries may be entered in three different formats: small-field, large-field, and free-field. For the small-field format, it is convenient to visualize each input line consisting of 80 columns. The 80 columns are divided into 10 fields with each field being 8 columns wide as shown in [Figure 2-5](#).

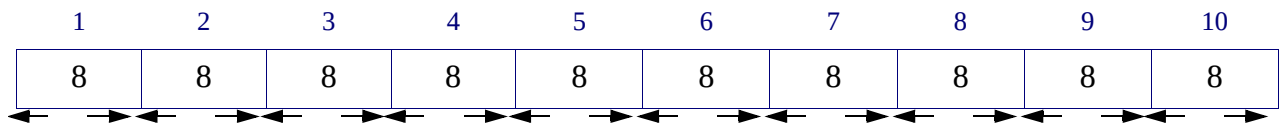


Figure 2-5 Small-Field Format

The name of the Bulk Data entry is entered in field 1 beginning in column 1. Fields 2 through 9 are used for data input. The only limitation is that the data within these fields must lie completely within the field, have no embedded blanks, and must be of the proper type. There are three distinct types of input used for Bulk Data entries: real, integer, and character; they are not interchangeable. If a particular field requires an integer, such as an ID number on the GRID entry, entering a real number causes a fatal error.

Field 10 of the Bulk Data entry is used for two purposes. If the Bulk Data entry does not have a continuation line, Field 10 may be used as an optional comment field. If the Bulk Data entry has a continuation line, field 10 is used for the continuation identifier. The continuation identifier must be unique with respect to all the other identifiers in your Bulk Data Section.

Real numbers may be entered in various ways; however, the number must contain a decimal point. For example, the real number 7.0 may be entered as 7., 7.0, 7.E0, .7E+1, 700.E-2, etc. Integer numbers do not have decimal points. Character input consists of one to eight alphanumeric characters, the first of which must be alphabetic.

A blank may be used in many of the data fields, in which case the appropriate default is used. The defaults for data fields, if they are applicable to a particular field, are described in the [MSC.Nastran Quick Reference Guide](#). There are certain situations that allow either

an integer or a real number in a data field. In these situations, the choice of an integer or a real number in these data fields affects the problem solution. Fortunately, there are only a few such situations, and they are discussed in the appropriate sections of this user's guide.

Consider the definition for grid point 2 of the truss model shown in **Figure 2-6**. The name of the entry is GRID, which begins in column 1 of field 1. The grid point ID (2 in this case) must be an integer (no decimal point) greater than 0. Fields 3 and 7 represent coordinate system IDs and also must be integers. Since these fields are blank, the default of 0 is used. Fields 4, 5, and 6 represent the physical location of the grid point, and they must be entered as real numbers. Optional fields are field 8, which is used to define permanently constrained degrees of freedom, and field 9, the superelement ID field. If they are used, only integers are acceptable. Since the GRID entry does not have a continuation line, field 10 may be used as a comment, if desired.

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	2		0.0	120.	0.		2345		

Figure 2-6 A Bulk Data Example: the GRID Entry

If a Bulk Data entry requires more than eight data fields, continuation lines must be used. When a continuation line is used, field 10 contains the continuation identifier that must be unique with respect to all other continuation identifiers. Column 1 of field 10 is always ignored and is not considered part of the identifier. The small field continuation line is denoted by a + symbol in column 1 of field 1. The continuation line is matched to its parent entry by the identifier in columns 2 through 8 of Field 1. An example is shown in **Figure 2-7**.

1	2	3	4	5	6	7	8	9	10
TYPE									+ID1
+ID1									+ID2
+ID2									

Figure 2-7 Small-Field Entry with Continuation Lines

The identifiers used in **Figure 2-7** are arbitrary. The only restriction is that the alphanumeric character strings used as the continuation identifiers do not contain the symbols *, =, or \$. The + symbol used in column 1 of field 10 is for clarity.

It is often more convenient to let MSC.Nastran generate the continuation identifiers automatically. To do this, leave both field 10 of the parent entry and field 1 of the continuation line blank, and place the continuation line directly following the parent entry. The blank field indicates that it is a small-field continuation of the previous line. The advantage of using continuation identifiers is that the input file may be unsorted, that is, the continuation lines do not have to follow the parent line. However, this benefit is generally not worth the inconvenience of using the continuation identifiers.

The most common form of input when generating the Bulk Data entries manually is the free-field format. In the free-field format, commas are used to separate the fields (blank spaces are also acceptable, but not recommended). An entry using the free-field format is identified by a comma or an equal sign in any of the first eight columns of the entry. Internally, the free-field format is converted to small-field format, so the same rules apply to free-field format entries as for the small-field format entries. Integers and character entries must be eight characters or less, and real numbers longer than eight characters are rounded to eight characters. Free-field format entries must start in column 1. If automatic continuation is to be used, the continuation line starts with a comma in field 1, indicating that the first field is blank.

You may find it necessary to input integers larger than eight characters or real numbers with more significant digits than can be placed in an eight-character field. In such cases, a large-field format option is available. With large-field format, as shown in [Figure 2-8](#), Fields 2 through 9 are 16 characters in length and require two lines instead of one.

1a	2	3	4	5	10a
TYPE*					*ID1

1b	6	7	8	9	10a
*ID1					

Figure 2-8 Large-Field Input Format

The large-field format entry is identified by the * symbol after the entry name. The large-field format continuation is denoted by the * symbol in column 1.

It is possible to mix small-field and large-field format continuations; however, it is generally not recommended because it becomes difficult to discern the locations of the fields. There is no automatic continuation feature for the large-field format. If the large-field format is used, the continuation identifiers must be used.

[Figure 2-9](#) shows a CORD2R entry using the three different formats. Knowledge of the CORD2R entry is not necessary for this example except to note that this entry requires a continuation line. Coordinate systems are discussed in “[Grid Points, Scalar Points, and Coordinate Systems](#)” on page 47.

1	2	3	4	5	6	7	8	9	10
CORD2R	1	0	0.0	0.0	0.0	1.0	0.0	0.0	+COR1
+COR1	1.0	1.0	0.0						

Small-Field Format with Continuation Identifier

CORD2R	1	0	0.0	0.0	0.0	1.0	0.0	0.0	
	1.0	1.0	0.0						

Small-Field Format without Continuation Identifier

CORD2R, 1, 0, 0.0, 0.0, 0.0, 1.0, 0.0, 0.0
, 1.0, 1.0, 0.0

Free-Field Format without Continuation Identifier

CORD2R*	1	0	0.0	0.0	*COR1
*COR1	0.0	1.0	0.0	0.0	*COR2
*COR2	1.0	1.0	0.0		*COR3
*COR3					

Large-Field Format

Figure 2-9 Input Format Example

In the examples shown in [Figure 2-9](#), column 1 of field 10 contains a + symbol for the small-field format and a * symbol for the large-field format. Since this column is ignored, the choice of the + and the * is arbitrary; however, these choices do improve readability because they serve as a reminder for the format of the continuation line. This convention is commonly used in the MSC.Nastran user community.

All Bulk Data entries are shown in small-field or large-field format throughout this user's guide whenever possible. This does not mean that free-field format should not be used; it is just less readable for this guide.

CHAPTER

3

Grid Points, Scalar Points, and Coordinate Systems

- Introduction to Degrees of Freedom
- Grid Points
- Scalar Points
- Coordinate Systems

3.1 Introduction to Degrees of Freedom

When you create a finite element model of a structure, you are creating a mathematical model representing your structure in matrix form. The unknowns in the matrix equation are the displacements in the model. These displacements consist of the six components for each of the grid points and one component for each of the scalar points. In general, the displacements at the grid points and the scalar points are referred to as the degrees of freedom of the model.

A scalar point is similar to a grid point in the way that it is assembled into the matrix equation. However, a scalar point has no spatial orientation; it is simply an additional degree of freedom that you can define in your model. Scalar points are useful for modeling with scalar elements, multipoint constraints (MPCs), CBEAM warping, and other applications.

This chapter describes both grid points and scalar points. Section [3.4](#) is devoted to the several types of coordinate systems available in MSC.Nastran.

3.2 Grid Points

The GRID Bulk Data entry is used to identify a grid point, specify the location of the grid point in space with respect to a reference coordinate system, assign permanent constraints, and define the directions of motions at the grid point. The format of the grid point entry is as follows:

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

Field	Contents
ID	Grid point identification number.
CP	Identification number of coordinate system in which the location of the grid point is defined.
X1, X2, X3	Location of the grid point in coordinate system CP.
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are defined at the grid point.
PS	Permanent single-point constraints associated with the grid point.
SEID	Superelement identification number.

The grid point ID must be unique with respect to all other grid points and scalar points in your model. The IDs do not have to be contiguous, and they do have to begin at 1. You will often find it convenient to use groups of IDs for a particular section of your model. For instance, if a model of an automobile is generated, the door may have grid point IDs, for example, in the range from 1 to 1000. The grid points used to model the hood may be in the range from 1001 to 2000, etc. Use whatever is convenient for clarity in understanding the results. The prudent choice of a grid point numbering scheme and the use of the SET command in the Case Control Section can help simplify the interpretation of the results, especially when more than one engineer is involved in the project.

In MSC.Nastran, each grid point has six degrees of freedom: three translational and three rotational. The degrees of freedom are denoted as u_1 , u_2 , u_3 , θ_1 , θ_2 , and θ_3 , or as T_1 , T_2 , T_3 , R_1 , R_2 , and R_3 . T_1 , T_2 , and T_3 are the three orthogonal components of translation parallel to the 1, 2, and 3 directions of the grid point's displacement coordinate system, respectively. R_1 , R_2 , and R_3 are the components of rotation in the same directions.

Field 8 of the GRID entry is known as the PS field (Permanent Single-Point Constraint) and may be used to constrain any or all the degrees of freedom associated with the grid point. These constraints are applied in the output coordinate system, which is referred to as the CD field (field 7), not the coordinate system that you use to specify the grid point location in, which is referred to as the CP coordinate system (field 3). If you are a new MSC.Nastran user, it may seem strange to define your constraints in a different coordinate system from the one your model was originally created in. However, doing so can give you considerable versatility in modeling a variety of joints and boundary conditions.

The output coordinate system that you specify in the CD field is the coordinate system used to apply the constraints, solve for the displacements, and output the results. The CP field (field 3) is used only to define the location of the grid point in space.

The constraints applied to the grid point using field 8 of the GRID entry are considered permanent because they cannot be changed during the run. Any degrees of freedom specified in the PS field are constrained for all subcases. In many situations, you want to analyze your structure using more than one set of constraints. For these situations, do not specify the constraints on the GRID entry. Instead, specify the constraints using an SPC entry (Single-Point Constraint) in the Bulk Data Section. The only difference from using the GRID entry is that constraints applied using an SPC entry must be selected through the Case Control Section and therefore may be different for different subcases. All constraints, whether they are applied with the use of an SPC entry or using the PS field of the GRID entry, are applied in the output coordinate system.

As an example of the GRID entry, consider the truss structure discussed in “**Organization of MSC.Nastran Files**” in Chapter 2. The geometry of the structure is defined by the location of the four grid points. These locations are specified in fields 4, 5, and 6 of the GRID entry with respect to the basic coordinate system. The basic coordinate system is the default coordinate system for both the CP and CD fields of the GRID entry. In this example, both the CP and CD fields are left blank so that the geometry is defined in the basic coordinate system and the displacement results are output in the basic coordinate system.

You are not confined to using the basic coordinate system for the CP or CD fields; they may be any user-defined coordinate system, which is commonly referred to as a local coordinate system. The CP and CD fields may reference the same coordinate system if you like, but it is not a requirement. It is quite common to use a local coordinate system to input the geometry (using the CP field) and obtain your output in the basic system (leave the CD field blank or use a 0). The local coordinate systems may be rectangular, cylindrical, or spherical. All of the coordinate systems are discussed in “**Coordinate Systems**” on page 52.

3.3 Scalar Points

A scalar point can be used whenever you need a single degree of freedom in your model. A scalar point has only one degree of freedom, not six as in the case of a grid point. A scalar point does not have a location in space, so it does not require a coordinate system. Common uses for scalar points include meshing models with scalar elements, computing relative and average motions, and defining the warping coefficients of a thin-walled beam.

In general, a grid point can be used as a scalar point. However, a grid point requires more input and requires you to constrain five of the six degrees of freedom. From an input point of view, it is more efficient to use scalar points when you need only a single degree of freedom. Furthermore, using scalar points is more efficient computationally than using a grid point with five of its six components constrained.

Scalar points are defined with the SPOINT entry as shown. You can specify up to eight scalar points on a single SPOINT entry. Scalar points can also be defined implicitly when you use scalar elements.

The format of the SPOINT entry is as follows:

SPOINT

SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Field	Contents

IDi	Scalar point identification number.

Examples of the use of scalar points are provided in “[MSC.Nastran Elements](#)” on page 65.

3.4 Coordinate Systems

Whenever you create a model using MSC.Nastran, you must generate a set of grid points which, together with the elements, define the size and shape of your model. In “[Grid Points](#)” on page 49, the location of the grid point is defined by three components (Fields 4, 5, and 6 of the GRID entry). However, when you define a location of the grid point in space using these components, you are inherently defining the point relative to a coordinate system. In the examples presented in “[Organization of MSC.Nastran Files](#)” on page 19, the grid points are defined in the basic coordinate by default. The basic coordinate system is a rectangular coordinate system implicitly defined in all MSC.Nastran finite element models.

You will often find it convenient to model your structure or part of your structure in a coordinate system other than the basic coordinate system. For example, if you are modeling a cylindrical pressure vessel pressure with spherical ends, then you choose a cylindrical coordinate system most convenient for the body of the vessel and a spherical coordinate system most convenient for the ends. The cylindrical and spherical coordinate systems are referred to as local coordinate systems. Another common example where you may wish to use local coordinates is when you are modeling structures with multiple components. By defining the components with their coordinate system, you can use dimensions directly from the drawings without having to convert the locations of the grid points to a common origin. Furthermore, when the components have their own coordinate system, their location and orientation may be changed by modifying the coordinate system entry, thereby minimizing the need for remeshing

The examples above are just two of the many reasons for using local coordinate systems to define your model. Before you start to create your model, keep in mind that prudent use of local coordinate systems can simplify the initial model generation and future model changes.

To specify the coordinate system for the location of the grid point, you enter the ID of that coordinate system in the CP field (Field 3) of the GRID entry. These local coordinate systems may be rectangular, cylindrical, or spherical. If the CP field is left blank, the grid point is defined in the basic coordinate system.

The ID of the basic coordinate system is 0; hence, entering a 0 in the CP field produces the same results as leaving the field blank. The local coordinate systems are identified by a positive integer ID. All local coordinate systems must directly or indirectly be defined in terms of the basic coordinate system. An example is presented later in this section.

It is very important to understand that the coordinate system identified in the CP field is used to define the location of the grid point. The grid point results (displacement, grid point forces, etc.) are generated and output in the coordinate system defined by the CD field of the GRID entry (Field 7). Since the CD field identifies the coordinate system in which the output quantities are output, it is often called the output coordinate system or the displacement coordinate system.

(For convenience, it is common to use a local coordinate system to generate the model but have the results displayed in the basic coordinate system.) In this case, you enter the ID of the local coordinate system in the CP field and leave the CD field blank. If, on the other hand, you need to output the displacement information in particular coordinate system, you enter the ID of that local coordinate system in the CD field. For many practical modeling situations, you may have several different output coordinate systems in a model. It is conceivable that every grid point in your model can have a different output coordinate system; however, the output may be difficult to interpret.

Although the CD field is referred to as the displacement coordinate system, it is much more than just the coordinate system used to output the results. In fact, it has a much more fundamental role in MSC.Nastran. The collection of all the output coordinate systems—that is, all the coordinate systems defined on the CD field of all the grid points—constitutes the global coordinate system. It is important to note that the global coordinate system is not necessarily a single coordinate system but rather the collection of all of your CD coordinate systems. The global stiffness matrix is assembled in the global system.

The concept of a global stiffness matrix was introduced in the example in “[Introduction](#)” on page 1. All of the degrees of freedom in your model are assembled into the global stiffness matrix. Remember, the global coordinate system is not a unique coordinate system but rather the collection of all the CD coordinate systems specified on all of the GRID entries. Therefore, the output coordinate system that you specify in the CD field for a particular grid point is the coordinate system used to apply the constraints, solve for the displacements, and output the results associated with that grid point. Since it is very important that you understand the significance of the MSC.Nastran global stiffness matrix, it is useful to discuss how the definition of coordinate systems in MSC.Nastran differs from the classical textbook approach.

Many finite element textbooks refer to a “global coordinate system,” but this term means something entirely different in MSC.Nastran. The global coordinate system referred to by most textbooks is actually the basic coordinate system in MSC.Nastran. These differences in terminology often confuse new users who have taken a finite element class or have used other FEA programs. It is important to understand the MSC.Nastran definition of the global coordinate system.

If you are a new user and have a background in FEA, there is a significant difference between the way MSC.Nastran defines its global coordinate system versus the way it is defined in most textbooks. The classical textbook approach is to generate the element stiffness matrices for all the elements and then transform these stiffness matrices into a unique common coordinate system called the global coordinate system. These individual elemental stiffness matrices are assembled into a single stiffness matrix commonly referred to as the global stiffness matrix. Each row and column of this global stiffness matrix represents a degree of freedom of the structural model. The coordinate system associated with each one of these degrees of freedom is the same.

Now consider what MSC.Nastran does. MSC.Nastran generates the element stiffness matrix in a similar manner as that used in the classical approach. However, instead of transforming the element stiffness matrices into a unique coordinate system, these matrices

are transformed into the coordinate system specified in the CD field for the grid points where the element is attached. The individual elemental stiffness matrices are then assembled to form the global stiffness matrix. However, the coordinate systems associated with the degrees of freedom in the MSC.Nastran global stiffness matrix may be different. This approach opens the door to a variety of elegant modeling techniques. If you prefer to have the global coordinate system be the same as the basic coordinate system, which is a unique system, you can leave the CD field blank for all the grid points in the model.

As can be seen, local coordinate systems play an important role in generating your models, solving for the solution, and displaying the results. The next section shows how to define a local coordinate system.

User-Defined Coordinate Systems

There are two methods to define a coordinate system. The first method (Method 1) is to define the location of the new coordinate system by referencing three grid points. The Bulk Data entries for this method are CORD1R (rectangular), CORD1C (cylindrical), and CORD1S (spherical).

The second method of defining a coordinate system is by specifying the location of three points. The Bulk Data entries for the second method are CORD2R (rectangular), CORD2C (cylindrical), and CORD2S (spherical).

In Method 2 the new coordinate system is defined in terms of an existing coordinate system, which may, or may not be the basic coordinate system. A word of caution: if this reference coordinate system is modified, the new local coordinate system is also modified. Similarly, if a new coordinate system is defined using Method 1 and the reference grid points are moved, the orientation of the new coordinate system is also changed. This problem is illustrated in a later example.

CORD1R and CORD2R Bulk Data Entries

Defining a rectangular coordinate with CORD1R entry requires three reference grid points: G1, G2, and G3. Referring to [Figure 3-1](#), grid point G1 defines the origin of the coordinate system. A vector taken from G1 to G2 defines the Z-axis. Grid point G3, together with this Z-axis, defines the XZ plane. The X-axis is defined to be in this XZ plane and perpendicular to the Z-axis. Finally, the Y-axis is generated from the X- and Z-axes using the right-hand rule. The format of the CORD1R is as follows:

CORD1R

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	GiB	G2B	G3B	

Field	Contents
CIDA, CIDB	Coordinate system identification number.
GiA, GiB	Grid point identification numbers.

The CORD2R uses the locations of three points A, B, and C in the same manner as the three grid points used for the CORD1R entry. These points are defined in space using coordinates relative to a reference coordinate system (Field 3). The reference coordinate system must be independently defined. The format of the CORD2R is as follows:

CORD2R

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Field	Contents
CID	Coordinate system identification number.
RID	Identification number of a coordinate system that is defined independently from the CID coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.

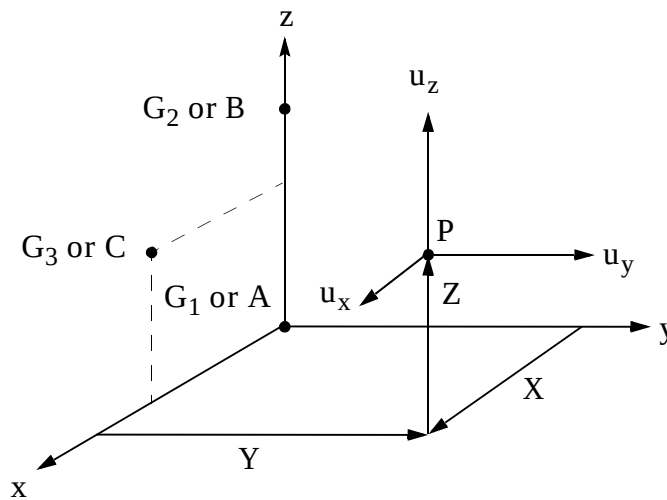


Figure 3-1 CORDIR and CORD2R Definitions

To better understand the use of a CORD1R entry, consider the truss structure introduced in “[Organization of MSC.Nastran Files](#)” in Chapter 2. The input file is modified so that the location of grid points 1 through 4 are defined in a rectangular coordinate system with an ID of 11 as shown in [Figure 3-2](#).

$$A = 4.0 \text{ in}^2$$

$$E = 30.0 \times 10^6 \text{ lb/in}^2$$

$$J = 1.27 \text{ in}^4$$

$$\nu = 0.3$$

○ Elements

□ Grid points

● Pin joints

○ Reference Grid Points

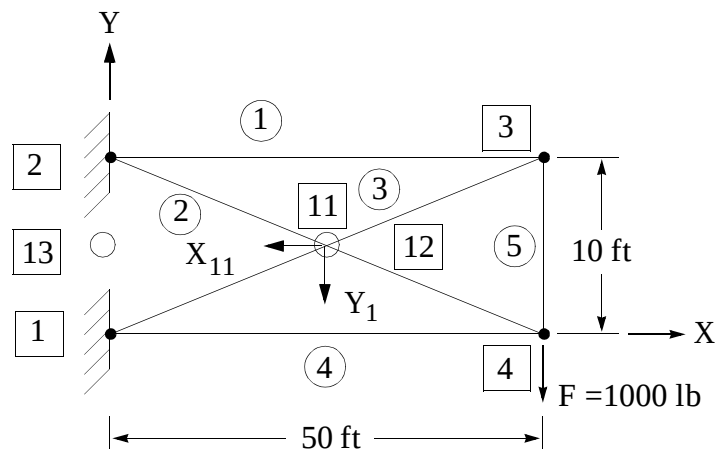


Figure 3-2 Grid Point Locations Using a Local Coordinate System

The new rectangular coordinate system with an ID of 11 has an origin located at the center of the truss and is rotated 180 degrees about the basic Z-axis.

The corresponding input file for the structure is given in [Listing 3-1](#). Note that it is necessary to add three additional grid points to the input file. Grid point 11 is located at the center of the truss structure, but it is not attached to the structure and is fully constrained. Grid point 12 is located 10 inches above grid point 11 in the basic Z-direction. The Z-axis for the new coordinate system is therefore parallel to the basic Z-axis. The choice of 10 inches is arbitrary since any positive value produces the same results. Grid point 13 defines the local XZ plane from which the X- and Y-axes of coordinate system 11 are determined. The choice of 0 inches for the X location of grid point 13 is also somewhat arbitrary since it is used only to define the XZ plane--any real number less than 300 inches produces the same results. If the X-component of grid point 13 is greater than 300 inches, then the X-axis will be in the basic X-direction instead of the direction shown.

Listing 3-1 Truss Using CORD1R

```

$
$ FILENAME - TRUSS6.DAT
$
BEGIN BULK
$
$ LOCAL COORDINATE SYSTEMS ARE DEFINED IN TERMS
$ GRID POINTS FOR METHOD 1
$
CORD1R  5          11          12          13
$
$ THE GRID POINTS LOCATIONS
$ DESCRIBE THE GEOMETRY IN CP FIELD COORDINATE SYSTEMS
$
GRID    1          5          -300.    -60.    0.          3456
GRID    2          5          -300.    60.    0.          3456
GRID    3          5          300.    60.    0.          3456
GRID    4          5          300.    -60.    0.          3456
$
$ THE FOLLOWING GRID POINTS ARE ADDED ONLY FOR DEFINING
$ LOCAL COORDINATE SYSTEM 5 AND ARE FULLY CONSTRAINED
$
GRID    11          300.    60.    0.          123456
GRID    12          300.    60.    10.         123456
GRID    13          0.    60.    5.          123456
$
      .
      .
      .
ENDDATA

```

It is important to note that grid point 11 is not attached to the structure in any way, although it occupies the same physical space as the structure shown in [Figure 3-2](#). The addition of the three grid points 11, 12, and 13 does not affect the stiffness of the structure. It is also important to note that the input file shown in [Listing 3-1](#) defines the same structure as shown in [Figure 2-2](#). Since the CD field of grid points 1, 2, 3, and 4 are the same in both files, the output of the two files will be the same.

The above example demonstrates the Method 1 approach (using grid points instead of vectors) to define a local coordinate system. For this example, three additional grid points are added to the model and are fully constrained. The next example also shows the Method 1 approach, but the three existing grid points are used instead of defining three new ones.

Consider the truss structure shown in [Figure 3-3](#). Suppose you want the positive T1 component of the displacements for all the grid points to be in the negative X_b -direction as shown in [Figure 3-3](#) (a). Therefore, you define a new coordinate system and arbitrarily give it an ID of 11. It is convenient to generate the new coordinate system using Method 1 with grid points 2, 4, and 1. The coordinate system 11 is entered in the CD field of all the grid points. These grid points already exist in the model; however, if you use them, you

may experience potential problems as shown in this example. The model was submitted using the existing grid points to define local coordinate system 11. After inspecting the results, everything appears to be fine.

Then you decide that the model needs to be modified. The change consists of moving the location of grid point 4 as shown in [Figure 3-3 \(b\)](#). You view the model using your graphics preprocessor. Again, everything looks fine, so you submit the model for another MSC.Nastran analysis.

However, in the process of moving grid point 4, you also inadvertently rotate the local coordinate system 11. When you look at the model using your graphics postprocessor, the deformed shape seems acceptable. However, the output shown in the output file is not in the direction you want. Why did this problem fail to appear on the preprocessor? The reason is that most postprocessors transform the results from the output coordinate system into their own screen coordinate system. If you are like many users, you check the overall results using the postprocessor, but if you need a displacement at a specific location, you review the output file. In this case, you were misled by failing to note that your model's coordinate orientation changed.

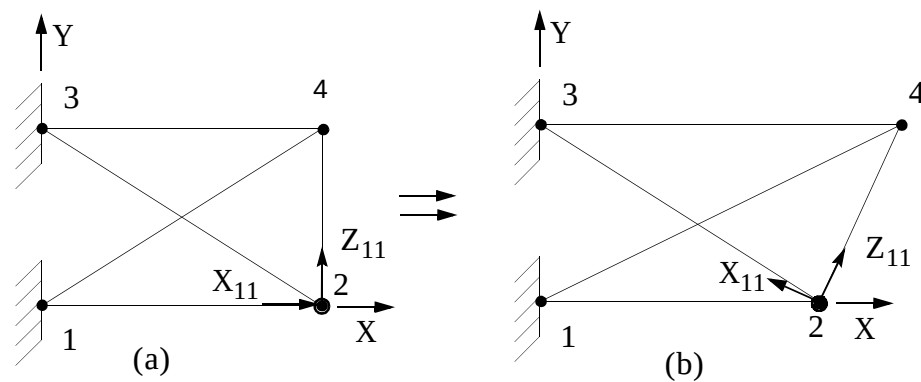


Figure 3-3 Model Change

To avoid this type of problem when using Method 1, the recommendation is to define three fully constrained grid points that are not part of the structure. Then if the model is modified, the local coordinate system remains unchanged.

CORD1C and CORD2C Bulk Data Entries

The CORD1C entry is used to define a cylindrical coordinate system using three grid points to define its location and orientation in space as shown in [Figure 3-4](#). Grid point G1 defines the origin of the cylindrical system. A vector from grid point G1 to grid point G2 defines the Z-axis. Grid point G3 together with the Z-axis defines the $\theta = 0$ plane. The format of the CORD1C is as follows:

CORD1C

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Field	Contents
CIDA, CIDB	Coordinate system identification number.
GiA, GiB	Grid point identification numbers.

The CORD2C uses the location of three points A, B, and C in the same manner as the three grid points are used for the CORD1C entry. These points are defined in space using coordinates relative to the reference coordinate system (Field 3). The reference coordinate system must be independently defined. The format of the CORD2C is as follows:

CORD2C

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

Field	Contents
CID	Coordinate system identification number.
RID	Identification number of a coordinate system that is defined independently from this coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.

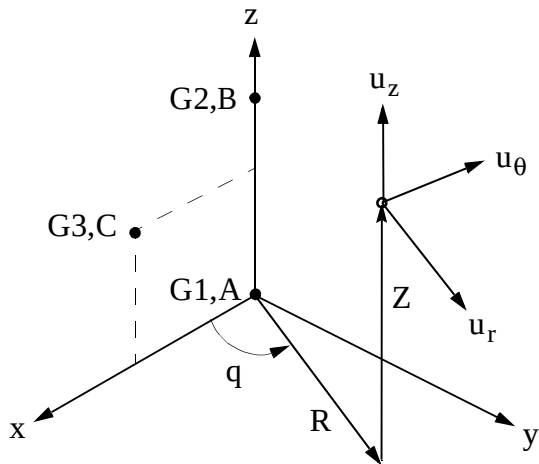


Figure 3-4 CORD1C and CORD2C Definitions

CORD1S and CORD2S Bulk Data Entries

The CORD1S entry is similar to the CORD1C entry since three grid points (G1, G2, and G3) are used to define the new coordinate system. As shown in [Figure 3-5](#), grid point G1 defines the origin of the spherical system. The vector starting at grid point G1 and passing through grid point G2 is the $\theta = 0$ axis. This axis, together with grid point G3, determines the $\phi = 0$ plane. The $\phi = 0$ axis is in this plane and is oriented using the right-hand rule. The format of the CORD1S entry is as follows:

CORD1S

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Field	Contents
CIDA, CIDB	Coordinate system identification numbers.
GiA, GiB	Grid point identification numbers.

The CORD2S entry uses the location of three points A, B, and C in the same manner as the three grid points are used for the CORD1S entry. These points are defined in space using coordinates relative to the reference coordinate system (Field 3). The reference coordinate system must be independently defined. The format of the CORD2S is as follows:

CORD2S

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Field	Contents
CID	Coordinate system identification number.
RID	Identification number of a coordinate system that is defined independently from this coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.

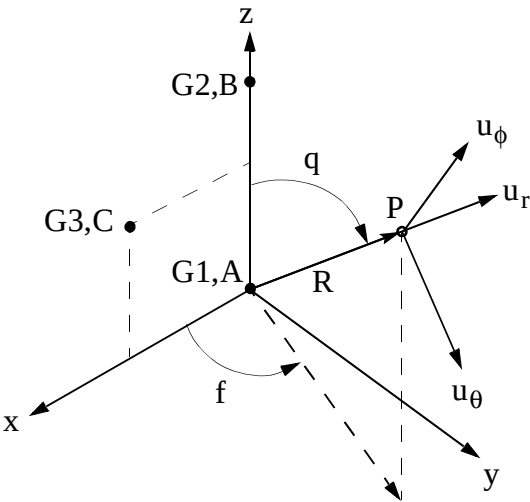


Figure 3-5 CORD1S and CORD2S Definitions

Note: θ for the spherical coordinate system is different than the θ in the cylindrical coordinate system.

To see the use of cylindrical and spherical coordinate systems, consider the analysis of the grain silo shown in **Figure 3-6**. The cylindrical body is modeled using a cylindrical coordinate system; the cap is modeled using a spherical coordinate system.

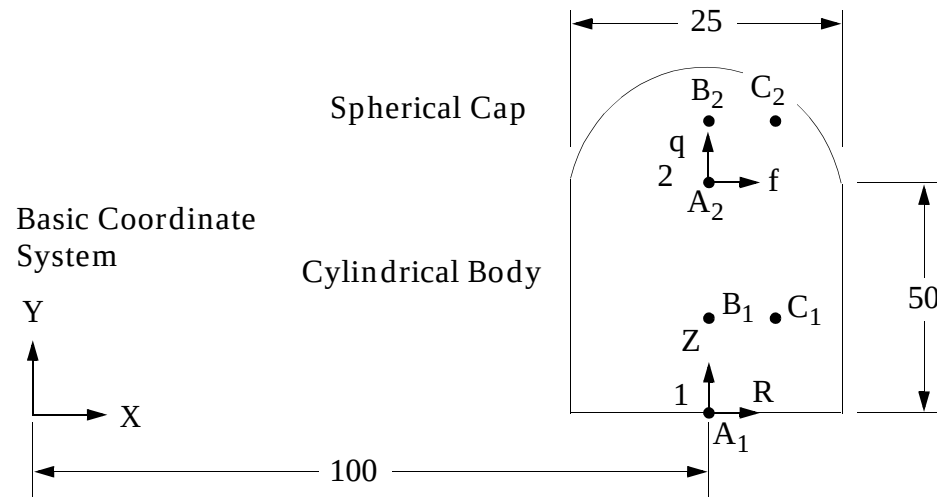


Figure 3-6 Silo Model

Coordinate system 1 is a cylindrical coordinate system defined using a CORD2C entry. This coordinate system describes the body of the silo and is defined with respect to the basic coordinate system. The origin is given by point A_1 , and located by the vector $(100,0,0)$ as shown in **Figure 3-6**. The Z-axis is defined by the line extending from point A_1 to point B_1 , which is located at $(100,1,0)$. The Z-axis along with point C_1 , which is located at $(101,1,0)$, defines the $\theta = 0$ plane. The R-axis is defined in this plane and is perpendicular to the Z-axis. The direction of the θ -axis is determined using the right-hand rule.

The spherical coordinate system with an ID of 2 is defined in terms of coordinate locations with respect to the reference coordinate system 1. The location of origin of the spherical coordinate system 2 is given by point A_2 at $(0,0,50)$. This location is with respect to coordinate system 1. The $\theta = 0$ axis is the line extending from A_2 to B_2 , which is located at $(0,0,51)$. Finally, the $\phi = 0$ plane is located by the $\phi = 0$ axis and point C_2 , which is located at $(1,0,51)$.

Since coordinate system 1 is defined in terms of the basic coordinate system, coordinate system 2 is also indirectly defined in terms of the basic coordinate system. It is also acceptable to define both coordinate systems directly in terms of the basic coordinate system (use whichever method is most convenient for you). However, since coordinate system 2 is linked to coordinate system 1, adjusting the position of the silo (for example, due to a model change) is only a matter of changing the locations of the grid points on the CORD2C entry.

Listing 3-2 Grain Silo Coordinate Systems

```
BEGIN BULK
$
$ THE CYLINDRICAL SYSTEM USED FOR THE SILO BODY
$
CORD2C 1      0      100.  0.    0.    100.  1.    0.    +COR1
+COR1  101.    1.    0.
$
$ THE SPHERICAL SYSTEM USED FOR THE SILO CAP
$
CORD2S 2      1      0.    0.    50.    0.    0.    51.    +COR2
+COR2  1.      0.    51.
      .
      .
      .
ENDDATA
```

The Element and Material Coordinate System

Every element in the MSC.Nastran library has its own unique element coordinate system that is used to output the element forces, moments, and stresses as well as to orient section properties. For some elements, such as the CROD element, the element coordinate systems are implicitly defined based on the element connectivity. For other elements, you must explicitly define the orientation of the element coordinate system. For further discussion of element coordinate systems, refer to [“MSC.Nastran Elements”](#) on page 65.

In addition to the element coordinate system, an element may also have an optional material coordinate system that can be used to define the orientation of orthotropic or anisotropic material. See [“Material Properties and Composites”](#) on page 177 for a discussion of material coordinate systems.

MSC.Nastran Elements

- Introduction to MSC.Nastran Elements
- Scalar Elements
- One-Dimensional Elements
- Two-Dimensional Elements
- Three-Dimensional Elements
- CBUSH Elements
- The GENEL Element
- CWELD Connector Element

4.1 Introduction to MSC.Nastran Elements

The MSC.Nastran element library consists of numerous elements that can be categorized as scalar, one-dimensional (1-D), two-dimensional (2-D), or three-dimensional (3-D). Before discussing the elements, it is useful to define each of these categories, which elements fall into each category, and how the categories are used in a general sense.

Scalar Elements

Scalar elements, also referred to as zero-dimensional elements, consist of the springs, masses, and viscous dampers. For static analysis, the scalar spring is the most commonly used scalar element. The scalar mass elements are used less frequently; however, they are useful when you need to model a concentrated mass in one direction. The scalar dampers are not used in static analysis. All of the scalar elements are defined between two degrees of freedom in your model or between one degree of freedom and ground.

The scalar elements do not derive their stiffness from physical properties; you define the stiffness directly. For example, in the CROD elements used in “[Organization of MSC.Nastran Files](#)” on page 19 and “[Grid Points, Scalar Points, and Coordinate Systems](#)” on page 47, the axial stiffness is computed by MSC.Nastran using the cross-sectional area entered on the PROD entry, the modulus of elasticity defined on the MAT1 entry, and the location of the grid points to which the CROD is connected. If you want to use scalar spring elements instead of the CROD element, you connect the appropriate degrees of freedom and enter the spring rate for each of the springs.

The scalar elements used in static analysis consist of the following:

- CELAS1, CELAS2, CELAS3, CELAS4, CBUSH - Scalar Spring Elements
- CMASS1, CMASS2, CMASS3, CMASS4 - Scalar Mass Elements

One-Dimensional Elements

A one-dimensional element is one in which the properties of the element are defined along a line or curve. Typical applications for the one-dimensional element include truss structures, beams, stiffeners, and many others. In fact, the truss structure used in “[Organization of MSC.Nastran Files](#)” on page 19 and “[Grid Points, Scalar Points, and Coordinate Systems](#)” on page 47 was modeled using CROD elements which are considered one-dimensional elements. A truss example was used to introduce MSC.Nastran because the CROD element is the simplest of all the structural elements. A list of the one-dimensional elements discussed in this chapter includes

- CROD - An element with axial stiffness and torsional stiffness about the axis for the element.
- CBAR - A straight prismatic element with axial, bending, and torsional stiffness.
- CBEAM - An element similar to the CBAR but with additional properties, such as variable cross-section, shear center offset from the neutral axis and others.
- CBEND - A curved element capable of internal pressure.

Two-Dimensional Elements

Two-dimensional elements, commonly referred to as plate and shell elements, are used to represent areas in your model where one of the dimensions is small in comparison to the other two. As shown **Figure 4-1**, the thickness is substantially less than dimensions a or b .

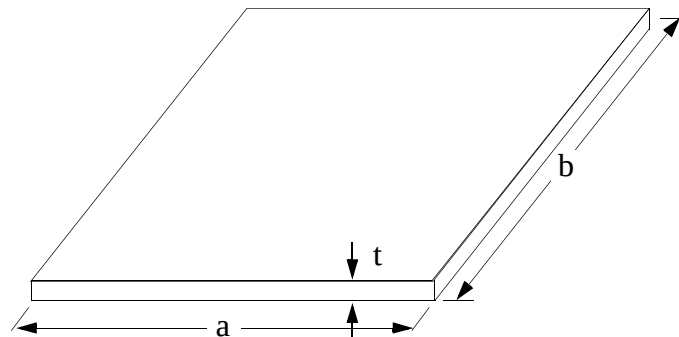


Figure 4-1 Typical Plate Element

In the finite element field, the membrane stiffness of the two-dimensional elements can be calculated using one of two theories: “plane stress” or “plane strain.” In the plane strain theory, the assumption is made that the strain across the thickness t is constant. Note that a two-dimensional element can be in either plane stress or plane strain, but not both. By default, the commonly used linear two-dimensional elements in MSC.Nastran are plane stress elements. The exception is the CSHEAR element which does not have inplane membrane stiffness and the CRAC2D, in which you must define whether the element is a plane stress or plane strain element. The plane strain formulation is specified using the PSHELL and PRAC2D entries.

Each of these two formulations, plane stress and plane strain, is applicable to certain classes of problems. If you are not familiar with the plane stress and plane strain theories, you probably need to use the plane stress formulation (the default). Most thin structures constructed from common engineering material, such as aluminum and steel, can be modeled effectively using plane stress. In other words, unless you have reason not to do so, use the plane stress formulation.

The two-dimensional elements that are discussed in this chapter are

- CSHEAR - A shear panel element, i.e., the element can transmit inplane shear forces only.
- CQUAD4, CTRIA3 - General-purpose plate elements capable of carrying inplane force, bending forces, and transverse shear force. This family of elements are the most commonly used 2-D elements in the MSC.Nastran element library.
- CQUAD8, CTRIA6 - Higher order elements that are useful for modeling curved surfaces with less elements than are required if you used the CQUAD4 and CTRIA3 elements. In general, the CQUAD4 and CTRIA3 elements are preferred over the CQUAD8 and CTRIA6 elements.
- CQUADR and CTRIAR - This family of 2-D elements are complementary to the CQUAD4 and CTRIA3 elements.

- CRAC2D - A element used to model surfaces with a discontinuity due to a crack.

Three-Dimensional Elements

Whenever you need to model a structure that does not behave as a bar or plate structure under the applied loads, you need to use one or more of the three-dimensional elements. The three-dimensional elements are commonly referred to as solid elements. Typical engineering applications of solid elements include engine blocks, brackets, and gears.

Three-dimensional elements that are discussed in this chapter include

- CHEXA, CPENTA, and CTETRA - General-purpose solid elements. This family of elements is recommended for most solid model applications.
- The CTRIA6 - A axisymmetric solid of revolution element. This element is used only in axisymmetric analysis.
- CRAC3D - A element used to model solids with a discontinuity due to a crack.

As a final comment on all of the elements, the purpose of this chapter is to discuss the commonly used elements and their applications to typical engineering problems. The elements are discussed in order of their size in terms of the number of degrees-of-freedom --they are not discussed in order of importance. Do not assume that the CBAR is more useful or preferred over a CQUAD4, for instance, because it is discussed first. The only exception to this are the scalar elements, which are discussed after the three-dimensional elements. This order is chosen because the scalar elements are used less frequently than the other elements.

Also note that the discussion in this chapter is limited to isotropic material properties, i.e., properties that can be described using a MAT1 entry. Non-isotropic and composite material properties are the subject of “[Material Properties and Composites](#)” on page 177.

4.2 Scalar Elements

A scalar element is an element that connects two degrees of freedom in the structure or one degree of freedom and ground. The degrees of freedom may be any of the six components of a grid point or the single component of a scalar point. Unlike the one-, two- and three-dimensional elements that are discussed in the previous sections, the scalar element lacks geometric definition. Hence, scalar elements do not have an element coordinate system.

Scalar elements are commonly used in conjunction with structural elements where the details of the physical structure are not known or required. Typical examples include shock absorbers, joint stiffness between linkages, isolation pads, and many others. Whenever scalar elements are used between grid points, it is highly recommended that the grid points be coincident. If the grid points are noncoincident, any forces applied to the grid point by the scalar element may induce moments on the structure, resulting in inaccurate results.

For static analysis, the linear scalar springs (CELASi, i = 1-4) and concentrated masses (CMASSi, i = 1-4) are useful. There are four types of scalar springs and mass definitions. The formats of the CELASi entries (elastic springs) are as follows:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

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

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Field	Contents
EID	Unique element identification number.
PID	Property identification number of a PELAS entry (CELAS1 and CELAS3).
G1, G2	Geometric grid point or scalar identification number (CELAS1 and CELAS2).
C1, C2	Component number (CELAS1 and CELAS2).
S1, S2	Scalar point identification numbers (CELAS 3 and CELAS4).
K	Stiffness of the scalar spring (CELAS2 and CELAS4).

S Stress coefficient (CELAS2).
 GE Damping coefficient (CELAS2).

Type one scalar elements (CELAS1, CMASS1) can reference both grid and scalar points. The element connectivity entry defines an ID of an appropriate property entry, similar to the CROD entry introduced in “[Organization of MSC.Nastran Files](#)” on page 19. Type two scalar elements (CELAS2, CMASS2) can also connect to both grid and scalar points; however, the property definition is on the element entry itself. Types three and four are equivalent to types one and two, respectively, except they can only reference scalar points. If a model consists of many scalar elements connected only to scalar points, it is more efficient to use these two latter types. The format of the PELAS entry is as follows:

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

Field	Contents
PIDi	Property identification number.
Ki	Elastic property value.
GEi	Damping coefficient.
Si	Stress coefficient.

As an example of the MSC.Nastran model consisting of only scalar points, consider the structure shown in [Figure 4-2](#). In this example, the CROD elements are replaced with equivalent springs.

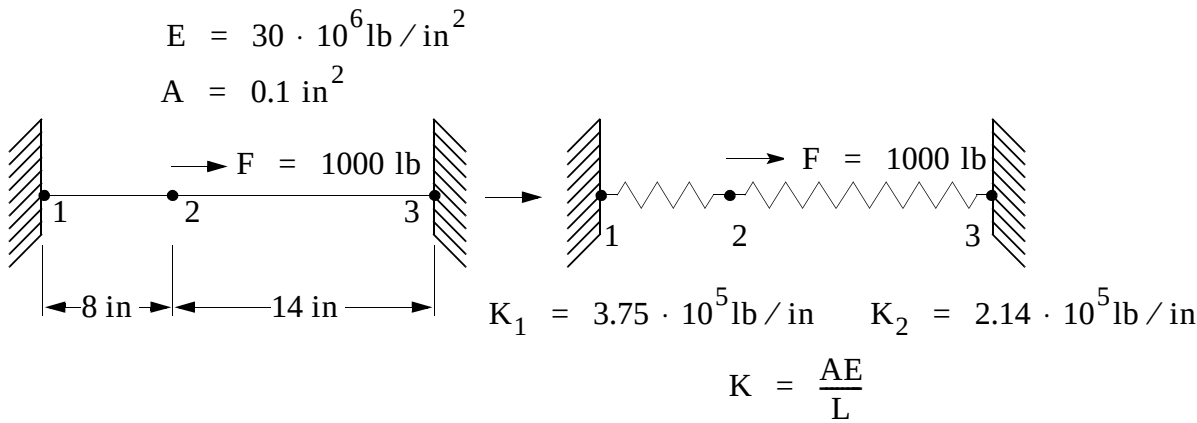


Figure 4-2 Equivalent Spring Model

The input file for the structure shown is given in [Listing 4-1](#).

Listing 4-1 Equivalent Spring Model

```

$
$ FILENAME - SPRING1.DAT
$
ID LINEAR,SPRING1
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE PROBLEM 4.1
SUBTITLE = ROD STRUCTURE MODELED USING SCALAR ELEMENTS
LABEL = POINT LOAD AT SCALAR POINT 2
LOAD = 1
SPC = 2
DISP = ALL
FORCE = ALL
BEGIN BULK
$
$ THE RESEQUENCER IN MSC/NASTRAN REQUIRES AT LEAST ONE GRID POINT
$ IN THE MODEL. IT IS FULLY CONSTRAINED AND WILL NOT AFFECT THE RESULTS
$
GRID      99          0.      0.      0.          123456
$
$ THE SCALAR POINTS DO NOT HAVE GEOMETRY
$
SPOINT   1          2          3
$
$ MEMBERS ARE MODELED SPRING ELEMENTS
$
CELAS4   1          3.75E5  1          2
CELAS4   2          2.14E5  2          3
$
$ POINT LOAD
$
SLOAD    1          2          1000.
$
SPC1     2          0          1          3
$
ENDDATA

```

This example is used only to demonstrate the use of the scalar element. In general, you do not replace structural elements with scalar springs. If this were an actual structure being analyzed, the preferred method would be to use CROD elements. The resulting output listing for this model is given in [Figure 4-3](#).

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	S	0.0	1.697793E-03	0.0				
99	G	0.0	0.0	0.0	0.0	0.0	0.0	
F O R C E S I N S C A L A R S P R I N G S (C E L A S 4)								
ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	
1	-6.366724E+02	2	3.633277E+02					

Figure 4-3 The Equivalent Spring Model Output

The displacement vector output also includes the displacement results for the scalar points. Since a scalar point has only one degree of freedom, MSC.Nastran displays up to six scalar points displacements per line. For the output shown in [Figure 4-3](#), the displacement of scalar points 1, 2, and 3 are labeled T1, T2, and T3, respectively. The POINT ID of 1 is the ID of the first scalar point in that row. The TYPE “S” indicates that all the output in that row is scalar point output.

The sign convention for the scalar force and stress results is determined by the order of the scalar point IDs on the element connectivity entry. The force in the scalar element is computed by [Eq. 4-1](#).

$$F_{\text{SPRING}} = K(U_1 - U_2) \quad \text{Eq. 4-1}$$

For the equivalent spring model, the force in element 1 is found to be

$$3.75 \cdot 10^5 \cdot (0.0 - 0.001698) = -636.7 \text{ lb}$$

This result agrees with the force shown in [Figure 4-3](#). If you reverse the order of SPOINT ID 1 and 2 in the CELAS2 entry for element 1, the force in the spring will be

$$3.75 \cdot 10^5 \cdot (0.001698 - 0.0) = 636.7 \text{ lb}$$

Neither answer is wrong-they simply follow the convention given by [Eq. 4-1](#).

The input file “spring1.dat” is included on the MSC.Nastran delivery CD. The same structure modeled with two CROD elements is also on the delivery media with the name “rod1.dat”.

4.3 One-Dimensional Elements

The one-dimensional elements are used to represent structural members that have stiffness along a line or curve between two grid points. Typical applications include beam type structures, stiffeners, tie-down members, supports, mesh transitions, and many others. The one-dimensional elements that are discussed in this chapter include the CROD, CBAR, CBEAM, and CTUBE elements.

The CROD Element

The CROD element is a straight prismatic element (the properties are constant along the length) that has only axial and torsional stiffness. The CROD element is the simplest element of all the elements that have geometry (the scalar elements are simpler; however, they do not have geometry associated with them). If desired, the CBAR or CBEAM element can be used to represent a rod member; however, these elements are somewhat more difficult to define because you need to specify an element coordinate system explicitly. If you need an element with only tension-compression and torsion, the CROD element is an ideal choice.

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

CROD

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

Field	Contents
EID	Element identification number.
PID	Property identification number of a PROD entry.
G1, G2	Grid point identification numbers of connection points.

The CROD element is defined by specifying the two grid points G1 and G2 that denote the end points of the element. The PID identifies the PROD entry that defines the cross-sectional area A, and the torsional constant J associated with the CROD element. The format of the PROD entry is as follows:

PROD

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

Field	Contents
PID	Property identification number.
MID	Material identification number.

A	Area of the rod.
J	Torsional constant.
C	Coefficient to determine torsional stress.
NSM	Nonstructural mass per unit length.

Referring to the above PROD entry, if the cross-sectional area field (field 4) is left blank on the PROD entry, the corresponding CROD element lacks axial stiffness. Likewise, if the field corresponding to the torsional stiffness is left blank (field 5), the corresponding CROD element does not have torsional stiffness.

An example using the CROD element is given in “**Grid Points, Scalar Points, and Coordinate Systems**” on page 47 without much description. In that example, the truss members are connected with pin joints. Since pin joints cannot transmit a moment, the only force acting on the members is an axial force. The CROD element is an ideal choice for this type of model because the CROD element only has stiffness along the axis of the element and in torsion about the axis of the element.

The conventions for the element coordinate system and the internal forces of the CROD element are shown in **Figure 4-4**.

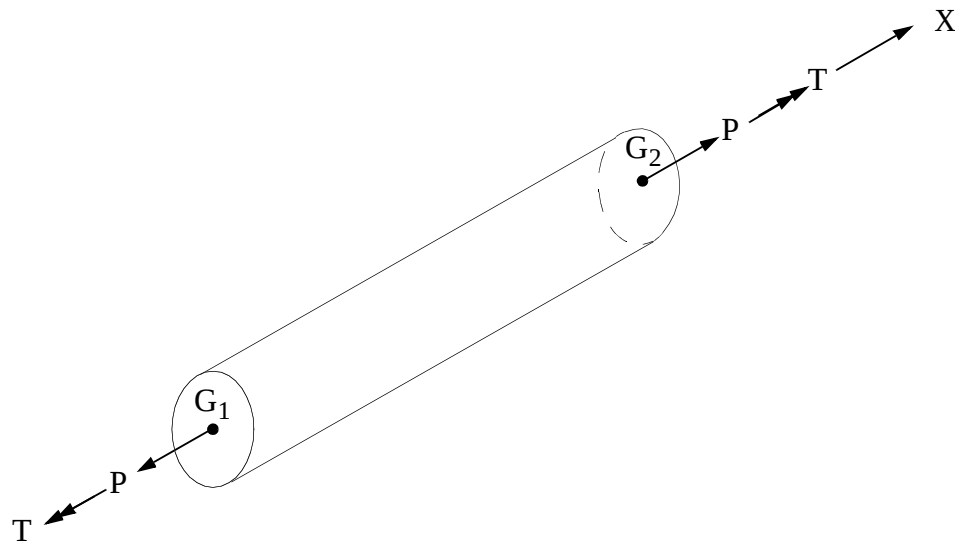


Figure 4-4 CROD Element Internal Forces and Moments

To illustrate the use of the CROD element, consider a three-member truss structure attached to a rigid wall as shown in **Figure 4-5**. The input file is shown in **Listing 4-2**.

Fixed Rigid Wall

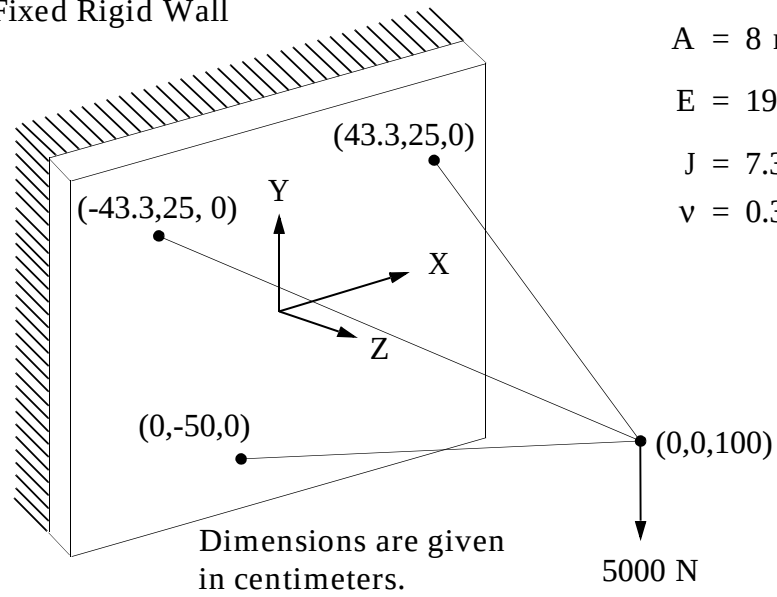


Figure 4-5 Three-Member Structure

Listing 4-2 Three-Member Truss Example

```

$
$ FILENAME - ROD2.DAT
$
ID LINEAR,ROD2
SOL 101
TIME 2
CEND
TITLE = LINEAR STATICS USER'S GUIDE
SUBTITLE = THREE ROD TRUSS MODEL
LABEL = POINT LOAD AT GRID POINT 4
LOAD = 1
DISPLACEMENT = ALL
STRESS = ALL
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID      1          -433.    250.    0.          123456
GRID      2           433.    250.    0.          123456
GRID      3           0.     -500.    0.          123456
GRID      4           0.      0.     1000.
$
$ MEMBERS ARE MODELED USING ROD ELEMENTS
$
CROD      1          1          1          4
CROD      2          1          2          4
CROD      3          1          3          4
$
$ PROPERTIES OF ROD ELEMENTS
$
PROD      1          1          8.          7.324
$
$ MATERIAL PROPERTIES
$
MAT1      1          19.9E4          .3
$
$ POINT LOAD
$
FORCE     1          4          5000.    0.          -1.          0.
$
ENDDATA

```

A selected portion of the output illustrating the displacements and the stresses is shown in **Figure 4-6**.

D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	0.0	0.0	0.0	0.0	0.0	0.0	
2	G	0.0	0.0	0.0	0.0	0.0	0.0	
3	G	0.0	0.0	0.0	0.0	0.0	0.0	
4	G	0.0	-1.170466E+01	-2.575031E-05	0.0	0.0	0.0	

S T R E S S E S I N R O D E L E M E N T S (C R O D)									
ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN
1	4.658455E+02		0.0		2	4.658455E+02		0.0	
3	-9.316950E+02		0.0						

Figure 4-6 Three-Member Truss Selected Output

Several important points should be made regarding the input file. The boundary conditions of the truss are applied as permanent constraints using the value 123456 in field 8 of the GRID entries with IDs 1, 2, and 3. There are no constraints applied to grid point 4, so it is free to move in any direction. To verify that boundary conditions are applied correctly, review the displacement vector in [Figure 4-6](#). As can be seen, the displacements of grid points 1, 2, and 3 are exactly equal to zero in each direction.

You should also note that grid point 4 displacement is acting in the YZ plane. This result makes sense because the structure is symmetric about the YZ plane and the applied load is in the Y-direction. Another item to note is that the rotation of grid point 4 is equal to zero in each of the three directions. This may be a little disconcerting if you are a new user since you may expect the grid point to rotate about the X-axis. However, the CROD element cannot transmit a bending load, and as such, the rotation at the end of the CROD element is not coupled to the translational degrees of freedom. In fact, if the rotation degrees of freedom for grid point 4 is constrained, the answers are the same.

The stress output shows an axial and torsional stress only. There are no bending stresses in a CROD element. In this case, the torsional stress is zero because there is no torsional load on the members. The safety of margin is blank in our example because the stress limits fields specified in the MAT1 entry are left blank. The MAT1 entry is discussed in [“Material Properties and Composites”](#) on page 177.

There are two other elements that possess axial and torsional stiffness only: the CONROD and the CTUBE element. The CONROD element is identical to the CROD element except that the rod properties are entered on the CONROD entry itself. The CTUBE element is the same as the CROD element except that its section properties are expressed as the outer diameter and the thickness of a circular tube. The extensional and torsional stiffness are computed from these tube dimensions. The properties of the CTUBE element are defined using a PTUBE entry.

The CBAR Element

The CBAR element is a straight one-dimensional element that connects two grid points. The capabilities and limitations of the CBAR element are as summarized below:

- Extensional stiffness along the neutral axis and torsional stiffness about the neutral axis may be defined.

- Bending and transverse shear stiffness can be defined in the two perpendicular directions to the CBAR element's axial direction.
- The properties must be constant along the length of the CBAR element. This limitation is not present in the CBEAM element.
- The shear center and the neutral axis must coincide. This limitation is not present in the CBEAM element.
- The ends of the CBAR element may be offset from the grid points.
- The effect of out-of-plane cross-sectional warping is neglected. This limitation is not present in the CBEAM element.
- The stress may be computed at up to four locations on the cross section at each end. Additional output may be obtained for intermediate locations along the length of the CBAR using the CBARAO Bulk Data entry.
- Transverse shear stiffness along the length of the CBAR can be included.

The stiffness of the CBAR element is derived from classical beam theory (plane cross sections remain plane during deformation).

The connectivity of the CBAR element is entered with the CBAR Bulk Data entry, and its format is as follows:

CBAR

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

Alternate Format

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

Field	Contents
EID	Unique element identification number.
PID	Property identification number of a PBAR entry.
GA, GB	Grid point identification numbers of connection points.
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA.
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0.

PA, PB Pin flags for bar ends A and B, respectively.

W1A, W2A, W3A Components of offset vectors WA and WB, respectively.

W1B, W2B, W3B

The properties of the CBAR elements are entered on the PBAR entry that is identified by the PID entered in field 3. The format the PBAR entry is as follows:

PBAR

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

Field	Contents
PID	Property identification number.
MID	Material identification number.
A	Area of bar cross section.
I1, I2, I12	Area moments of inertia.
J	Torsional constant.
NSM	Nonstructural mass per unit length.
Ci, Di, Ei, Fi	Stress recovery coefficients.
K1, K2	Area factor for shear.

As in the case of the CROD element, any of the stiffnesses and flexibilities may be omitted by leaving the appropriate fields on the PBAR entry blank. For example, if fields 5 and 6 are blank, the element will lack bending stiffness.

One the most difficult aspects of the CBAR (or CBEAM element) for the first-time users is understanding the need to define an orientation vector. The best way to see the need for the orientation vector is by an example. Consider the two I-beams shown in [Figure 4-7](#). The I-beams have the same properties because they have the same dimensions; however, since they have different orientations in space, their stiffness contribution to the structure is different. Therefore, simply specifying the I-beam properties and the location of the end points via the grid points is insufficient-you must also describe the orientation. This is done using the orientation vector.

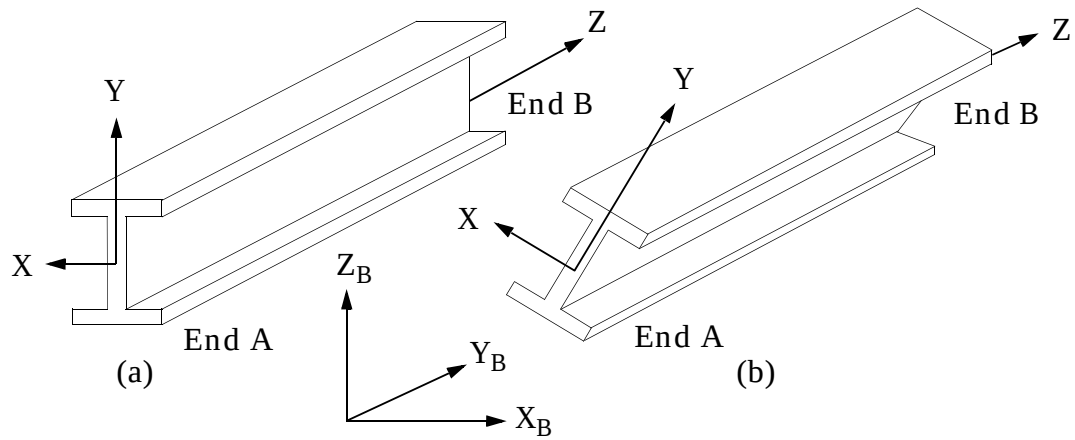


Figure 4-7 Demonstration of Beam Orientation

Defining the orientation of a element in space is accomplished with the use of an orientation vector. Another way of looking at the orientation vector is that it is a vector that specifies the local element coordinate system. Since the geometric properties are entered in the element coordinate system, this orientation vector specifies the orientation of the element.

The orientation vector $\vec{v}$ as it is related to the CBAR element coordinate system is shown in [Figure 4-8](#). Vector $\vec{v}$ defines plane 1, which contains the elemental x- and y-axes.

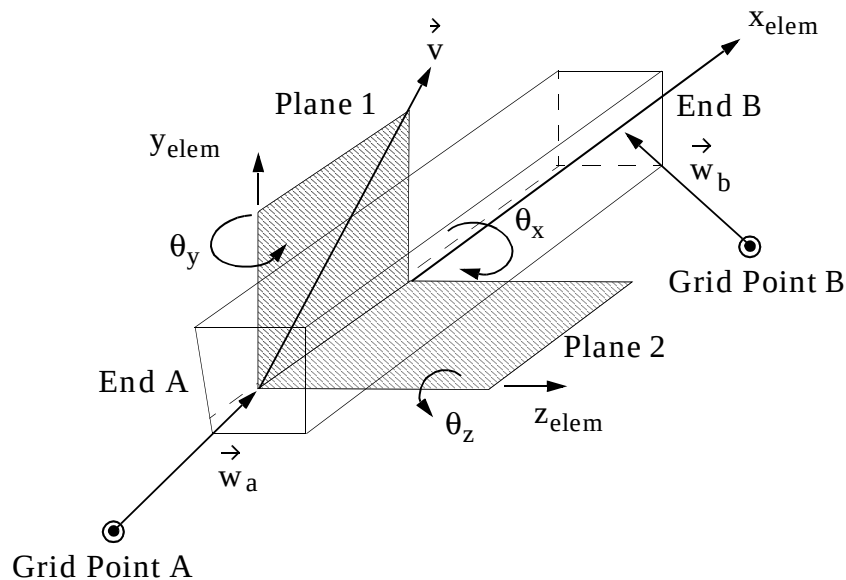


Figure 4-8 CBAR Element Coordinate System

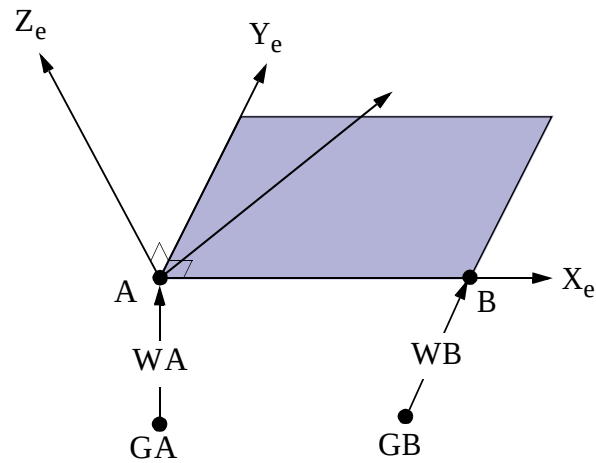
Referring to [Figure 4-8](#), the element x-axis is defined as the line extending from end A (the end at grid point GA) to end B (the end at grid point GB). Grid points GA and GB are defined on the CBAR entry. Also note that the ends of the CBAR element may be offset from the grid points using WA and WB as defined on the CBAR entry. Therefore, the element x-axis does not necessarily extend from grid point GA to grid point GB; it extends from end A to end B.

The element y-axis is defined to be the axis in Plane 1 extending from end A and perpendicular to the element x-axis. It is your responsibility to define Plane 1. Plane 1 is the plane containing the element x-axis and the orientation vector $\vec{v}$. After defining the

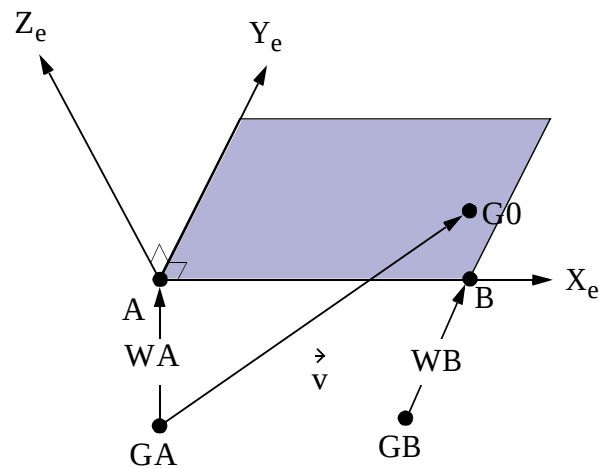
element x- and y-axes, the element z-axis is obtained using the right-hand rule, $z = x \otimes y$. Finally, Plane 2 is the plane containing element x- and z-axes. Note that once you defined grid point GA and GB and the orientation vector $\vec{v}$, the element coordinate system is computed automatically by MSC.Nastran.

The vector $\vec{v}$ shown in **Figure 4-8** may be defined by one of two methods on the CBAR entry. The first method is to define vector $\vec{v}$ by entering the components of the vector, (X1, X2, X3), which is defined in a coordinate system located at the end of the CBAR. This coordinate system is parallel to the displacement coordinate system of the grid point GA, which is defined in field 7 of the GRID entry. You can alternatively define the vector $\vec{v}$ with the use of another grid point called G0, which is entered in field 6 of the CBAR entry.

As mentioned previously, the ends of the CBAR may be offset from the grid points using the vectors WA and WB. If the CBAR is offset from the grid points and the components of vector $\vec{v}$ are entered (i.e., field 6 of the CBAR entry is a real number), then the tail of vector $\vec{v}$ is at end A, not grid point GA. If the vector $\vec{v}$ is defined with the use of another grid point G0 (field 6 of the CBAR entry is an integer number), then vector $\vec{v}$ is defined as the line originating at grid point GA, not end A, and passing through G0. Note that Plane 1 is parallel to the vector GA-G0 and passes through the location of end A.



Vector $\vec{v}$ Defined Using the Component Method



Vector $\vec{v}$ Defined Using the Alternate (Grid G0) Form

Figure 4-9 C-BAR Element Orientation Vector

The offset values WA and WB are entered by specifying the components of an offset vector in the displacement coordinate systems for GA and GB, respectively. The three components of the offset vectors are entered using fields 4 through 9 of the C-BAR continuation entry. When you specify an offset, you are effectively defining a rigid connection from the grid point to the end of the element.

The element forces and stresses are computed and output in the element coordinate system. [Figure 4-10](#) shows the forces acting on the C-BAR element. V_1 and M_1 are the shear force and bending moment acting in Plane 1, and V_2 and M_2 are the shear force and bending moment acting in Plane 2.

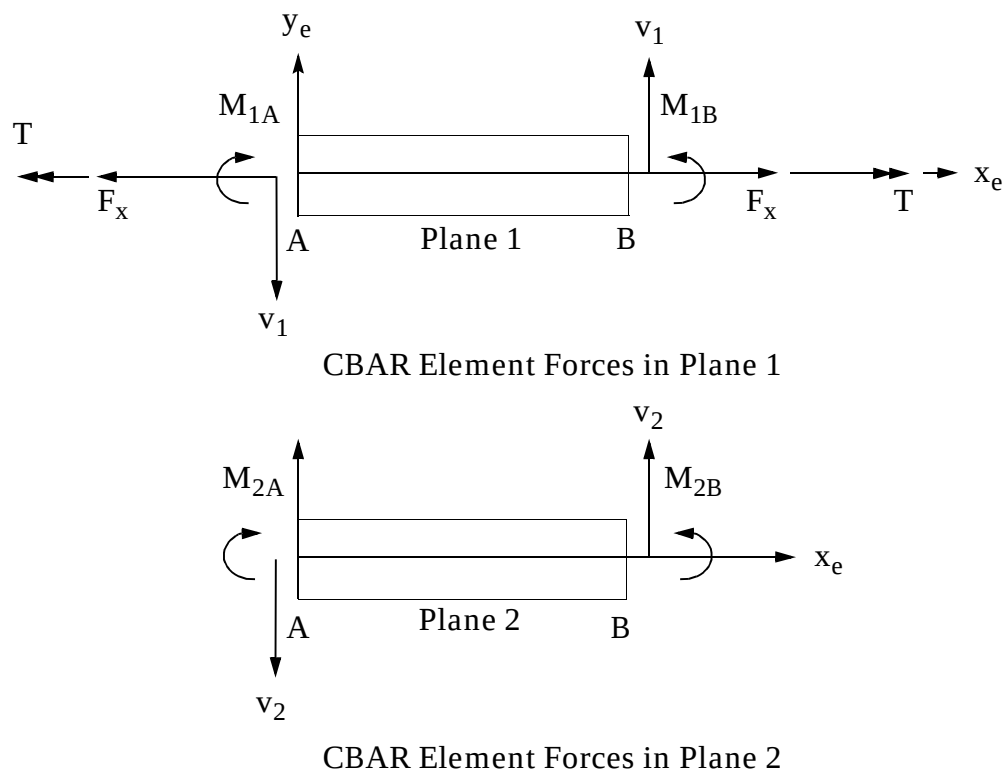
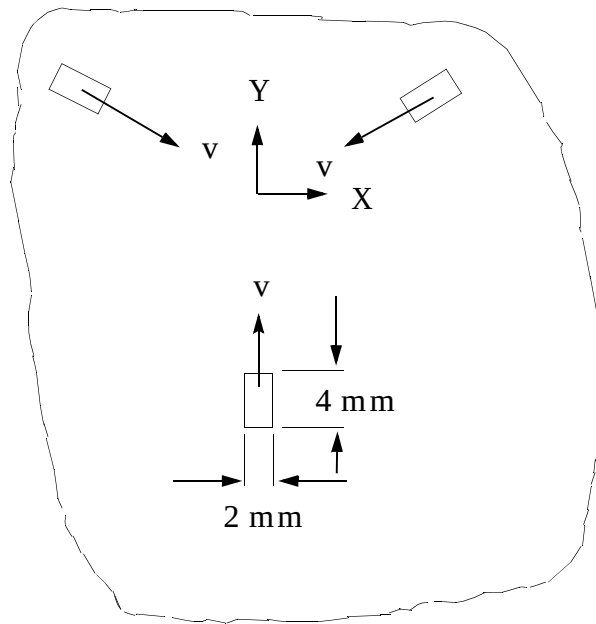


Figure 4-10 CBAR Element Forces

The area moments of inertia I_1 and I_2 are entered in fields 5 and 6, respectively, of the PBAR entry. I_1 is the area moment of inertia to resist a moment in Plane 1. I_1 is not the moment of inertia about Plane 1 as many new users may think at first. Consider the cross section shown in [Figure 4-10](#); in this case, I_1 is what most textbooks call I_{zz} , and I_2 is I_{yy} . The area product of inertia I_{12} , if needed, may be entered using field 4 of the second continuation entry. For most common engineering cross sections, it is usually not necessary to define an I_{12} . By aligning the element y - and the z -axes with the principal axes of the cross section, I_{12} is equal to zero and is therefore not needed.

As an example, consider the three-member truss structure in the previous section. Now suppose the joints are rigidly connected so that the members are to carry a bending load. Since CROD elements cannot transmit a bending load, they cannot be used for this problem. The CBAR element is a good choice because it contains bending stiffness (if area moments of inertia are input within the PBAR entry).

The dimensions and orientations of the member cross sections are shown in [Figure 4-11](#).



All three cross sections are the same.

$$A = 8 \text{ mm}^2$$

$$I1 = 10.67 \text{ mm}^4$$

$$I2 = 2.67 \text{ mm}^4$$

$$J = 7.324 \text{ mm}^4$$

Figure 4-11 CBAR Cross Sections

In this example, the area moments of inertia for each of the members are $I1 = 10.67$ and $I2 = 2.67$ for Plane 1 and Plane 2, respectively. The Bulk Data input for this problem is shown in [Listing 4-3](#). The vector $\vec{v}$ for each of the CBAR elements is defined using the component method. The fact that field 6 of the CBAR entries contains a real number (a decimal point is present) tells MSC.Nastran that the component method is being used. Note that the length of vector $\vec{v}$ is not critical--only the orientation is required to define Plane 1.

Listing 4-3 Vector $\vec{v}$ Entered Using the Components of the Vector.

```

$
$ FILENAME - BAR1.DAT
$
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID      1      -433.    250.    0.      123456
GRID      2      433.    250.    0.      123456
GRID      3        0.   -500.    0.      123456
GRID      4        0.     0.   1000.
$
$ MEMBERS ARE MODELED USING BAR ELEMENTS
$ VECTOR V DEFINED USING THE COMPONENT METHOD
$
CBAR      1      1      1      4      43.3   -25.    0.
CBAR      2      1      2      4     -43.3   -25.    0.
CBAR      3      1      3      4        0.     1.     0.
$
$ PROPERTIES OF BAR ELEMENTS
$
PBAR      1      1      8.      10.67   2.67   7.324
          2.      1.     -2.      1.      2.     -1.     -2.     -1.
$
$ MATERIAL PROPERTIES
$
MAT1      1      19.9E4      .3
$
$ POINT LOAD
$
FORCE     1      4      5000.    0.     -1.     0.
$
ENDDATA

```

You may notice that all of the orientation vectors point toward the geometric center of the bars. It is just as easy to define the orientation vectors for this structure to use the G0 method as shown in [Listing 4-4](#).

Although the G0 grid point may be any grid point in the model, it is good modeling practice to use a grid point that is not attached to the structure. If the grid point is part of the structure and the structure is modified, the vector orientation may be inadvertently changed, resulting in a modeling error. A modeling error of this nature is usually very difficult to identify. In this example, a new grid point with ID 99 was created and fixed at location (0,0,0). Both methods shown in [Listing 4-3](#) and [Listing 4-4](#) produce the same results.

Listing 4-4 Vector Entered Using the G0 Method

```

$
$ FILENAME - BAR1A.DAT
$
GRID      1      -433.    250.    0.      123456
GRID      2      433.    250.    0.      123456
GRID      3        0.   -500.    0.      123456
GRID      4        0.     0.   1000.      123456
GRID     99        0.     0.     0.      123456
$
$ MEMBERS ARE MODELED USING BAR ELEMENTS
$ VECTOR V DEFINED USING THE COMPONENT METHOD
$
CBAR      1        1        1        4        99
CBAR      2        1        2        4        99
CBAR      3        1        3        4        99

```

The displacement and stress results are shown in [Figure 4-12](#). As expected, the displacement of grid point 4 is in the YZ plane due to symmetry (this was also case for the previous CROD example). However, the displacement of grid point 4 is slightly less because of the addition of the bending stiffness. Also, the rotations at grid point 4 are now nonzero values because they are connected to the structure and are free to move. It is interesting to note that the bending stiffness makes little difference for this structure (the Y-displacement is 11.70445 for the CROD model versus 11.70466 for the CBAR model). This difference results because the axial forces are the primary loads for this structure. For models such as these, you can save CPU time by using the CROD elements instead of the CBAR elements.

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	0.0	0.0	0.0	0.0	0.0
3	G	0.0	0.0	0.0	0.0	0.0	0.0
4	G	0.0	-1.170445E+01	-2.574947E-05	1.447189E-02	0.0	0.0

ELEMENT ID.	STRESSES IN BAR ELEMENTS				AXIAL STRESS	(CBAR)		M.S.-T M.S.-C
	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4		SA-MAX SB-MAX	SA-MIN SB-MIN	
1	8.434532E-01	1.054002E+01	-1.054002E+01	-8.434532E-01	4.658372E+02	4.763772E+02	4.552972E+02	
	-2.004801E+00	-1.397634E+00	1.397634E+00	2.004801E+00		4.678420E+02	4.638324E+02	
2	-1.054002E+01	-8.434532E-01	8.434532E-01	1.054002E+01	4.658372E+02	4.763772E+02	4.552972E+02	
	1.397634E+00	2.004801E+00	-2.004801E+00	-1.397634E+00		4.678420E+02	4.638324E+02	
3	9.696071E+00	-9.696071E+00	9.696071E+00	-9.696071E+00	-9.316785E+02	-9.219824E+02	-9.413745E+02	
	6.073934E-01	-6.073934E-01	6.073934E-01	-6.073934E-01		-9.310710E+02	-9.322859E+02	

Figure 4-12 Displacement and Stress Results for the Three-Member Bar Structure

The SA_i and SB_i are the bending stresses at ends A and B, respectively. The $i = 1, 2, 3$, and 4 stress recovery locations correspond to the locations C, D, E, and F on the cross section, respectively. The location of these stress recovery coefficients are defined in CBAR's element coordinate system. Consider the cross section for our example as shown

in **Figure 4-13**. By request, the stresses are computed at four locations. As is commonly done, the stress locations represent the farthest points from the neutral axis of the cross section. These points are the locations of the maximum bending stress.

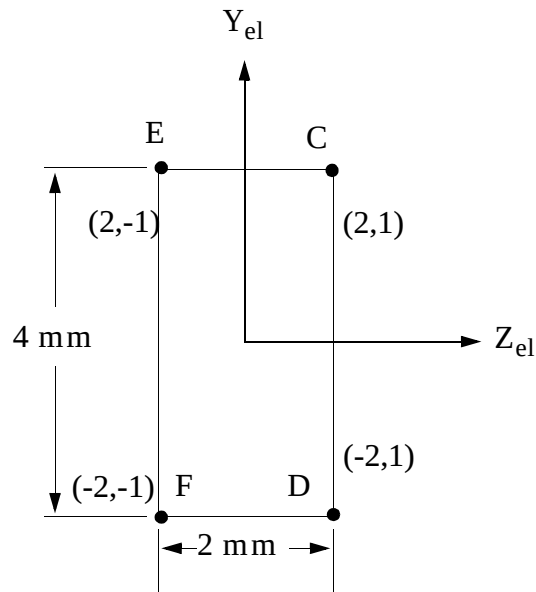


Figure 4-13 Stress Recovery Locations

In addition to the normal stress included in the stress output, there is axial stress in the CBAR elements, which is constant along the length of the bar. The SA-MAX, SB-MAX, SA-MIN, and SB-MIN stresses are the maximum and minimum combined bending and axial stresses for each end. There is no torsional stress recovery for the CBAR element. The last column of the stress output is the margin-of-safety calculation based on the tension and compression stress limits entered on the material entry. Since the stress limit fields are left blank, margin-of-safeties are not computed for this example.

It is important to remember that the margin-of-safety computation does not include the torsional stress. If the torsional stress is important in your stress analysis, use the torsional force output to compute the stress outside of MSC.Nastran. Although this may seem like an unnecessary burden, it is very logical. The torsional stress is highly dependent on the geometry of the CBAR's cross section, which MSC.Nastran does not know. For this example, the cross-sectional properties (A, I1, I2, J) of each member are input, but the fact that the cross-section is rectangular is unknown to MSC.Nastran. To compute the torsional stress, a formula for a rectangular cross section should be used.

PBARL -- An Alternate and Convenient Method for Defining Bar Cross Sections

The PBAR input that has been discussed so far requires you to calculate the cross-sectional properties of the beam (such as area, moments of inertia, shear center, etc.). Although this is not a particularly difficult task for standard cross sections, it is tedious and prone to unnecessary input errors.

A new beam cross-section library is introduced in Version 69 to provide you a simple interface to input beam cross sections into MSC.Nastran. A number of common cross-section types such as bar, box, I-beam, channel, angle, etc. are described within the Bulk

Data Section by the cross section's dimensions instead of the section properties. For example, a rectangle cross section may be defined by its height and depth rather than the area, moments of inertia, etc. The section properties are calculated based on thin wall assumptions.

The PBARL Bulk Data entry allows you to input cross section types along with their characteristic dimensions. You can choose from 19 different cross section shapes. These shapes are as follows: ROD, TUBE, I, CHAN (channel), T, BOX, BAR (rectangle), CROSS, H, T1, I1, CHAN1, Z, CHAN2, T2, BOX1, HEXA (hexagon), HAT (hat section), and HAT1.

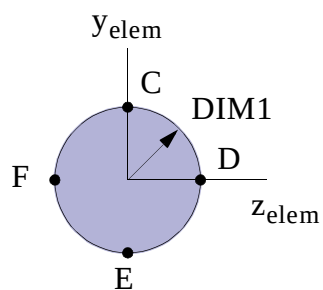
For some of these shapes (I, CHAN, T, and BOX), you can also select different orientations. All the 19 different shapes and orientations are shown in [Figure 4-14](#).

To define section attributes such as height and width on the DIMi fields, use the PBARL entry. To define section properties such as area and moment of inertia, use the PBAR Bulk Data entry. The new PBARL entries are easier to use and still retain most of the capabilities of the existing PBAR method, including non-structural mass.

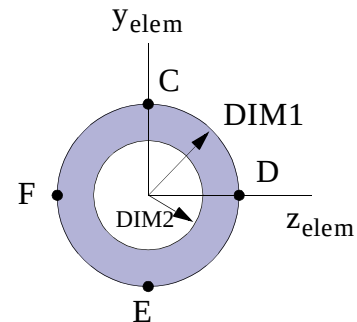
An additional difference between the PBARL and the PBAR entries is that stress recovery points need not be specified to obtain stress output for the PBARL entry. The stress recovery points are automatically calculated at the locations shown in [Figure 4-14](#) to give the maximum stress for the cross section.

In addition to the standard sections provided by MSC, you can add your own library of cross sections to suit your needs. To add your own library, write a few simple subroutines in FORTRAN to interface with MSC.Nastran through Inter-Process Communications. See the [MSC.Nastran Reference Manual](#) and the [MSC.Nastran 2005 Release Guide](#) for further details regarding the creation of your own library of cross sections.

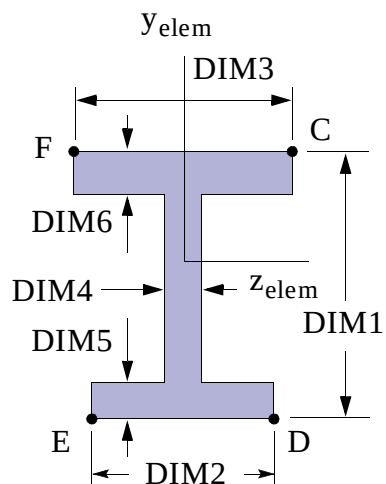
This automatic cross section computation greatly simplifies the formulation of design variables for design optimization applications.



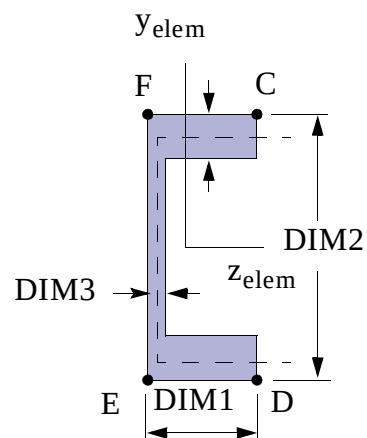
TYPE="ROD"



TYPE="TUBE"



TYPE="I"



TYPE="CHAN"

Figure 4-14 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element

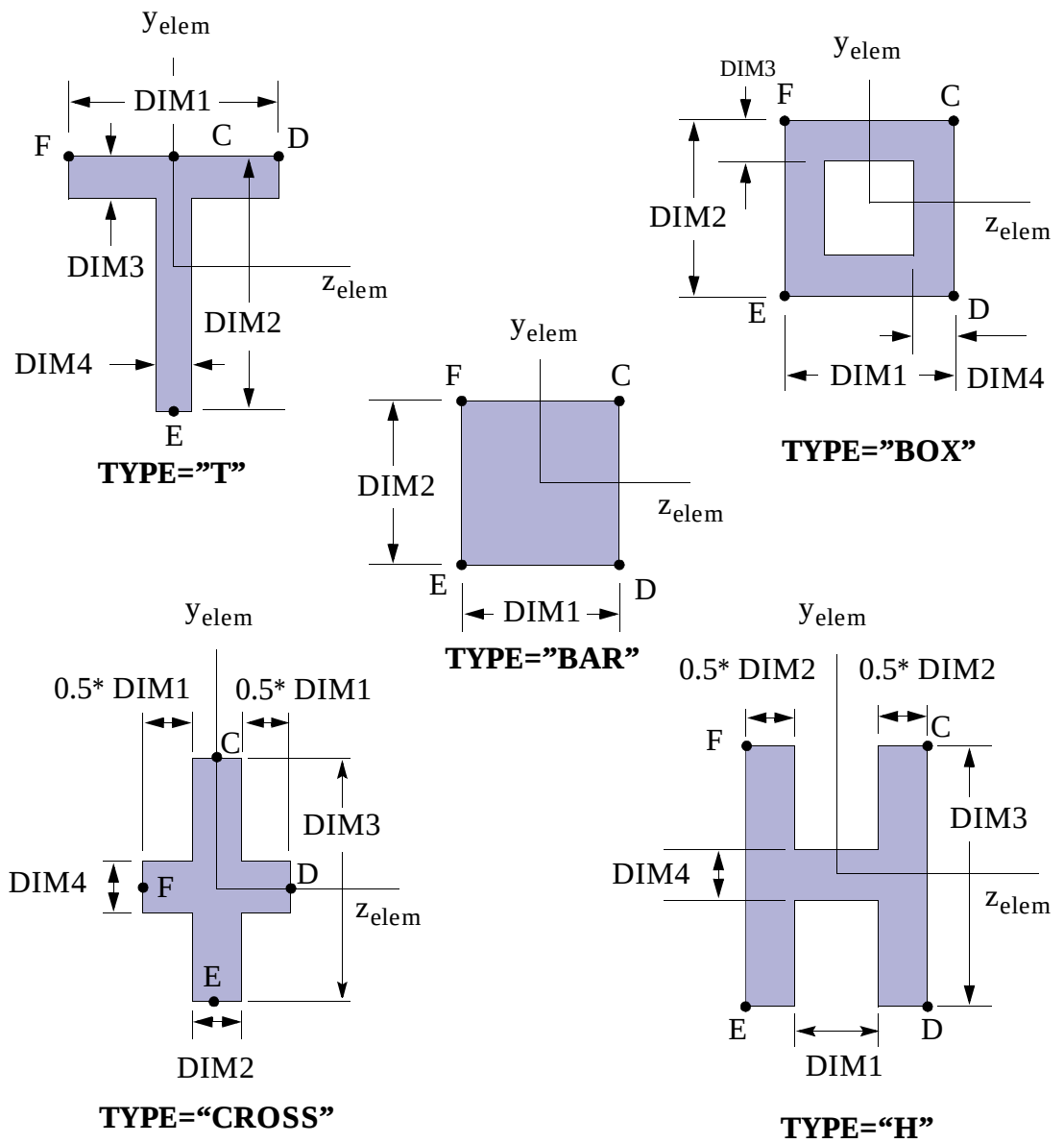


Figure 4-12 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)

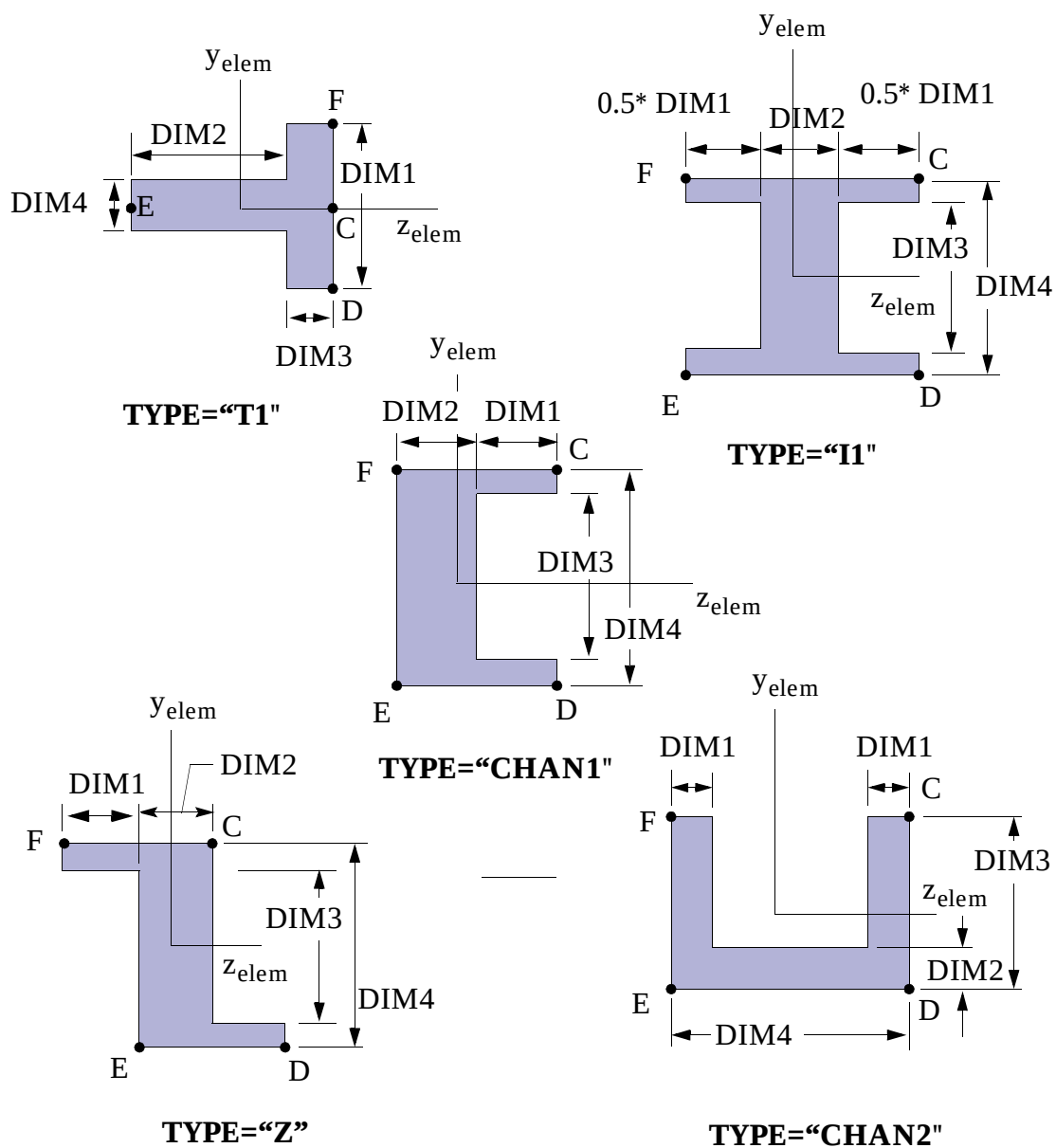
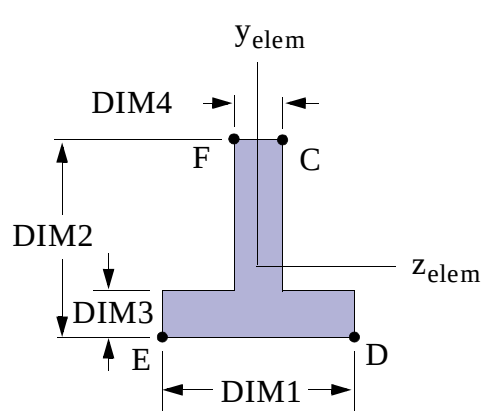
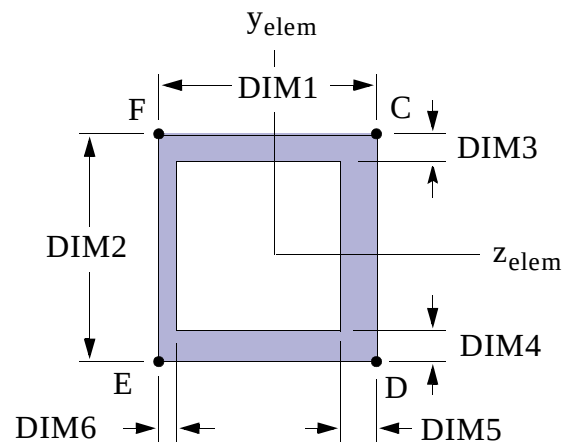


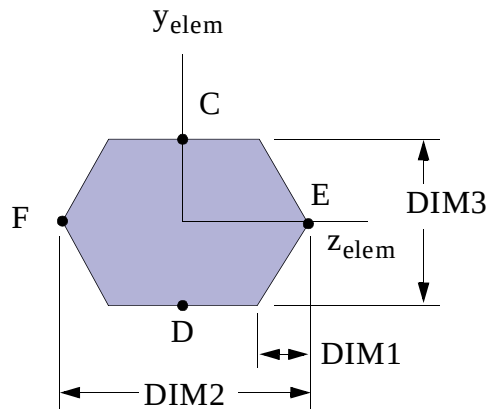
Figure 4-12 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)



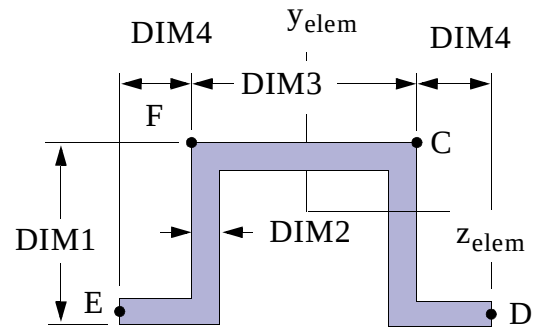
TYPE="T2"



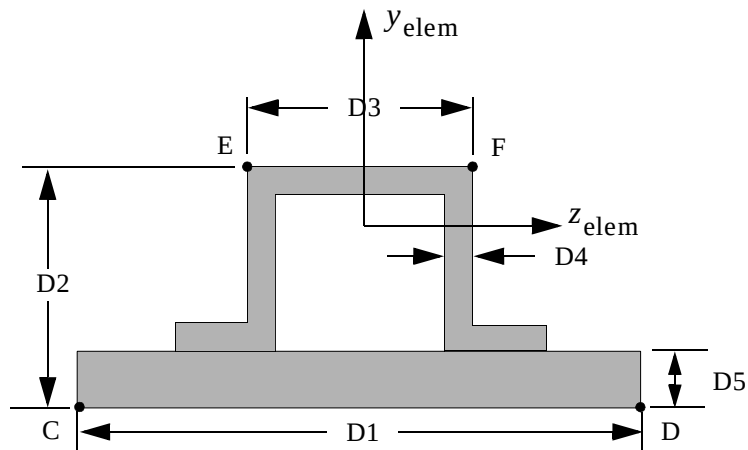
TYPE="BOX1"



TYPE="HEXA"



TYPE="HAT"



TYPE="HAT1"

Figure 4-12 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)

PBARL

The format of the PBARL entry is as follows:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID	GROUP	TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Default = "MSCBML0")
TYPE	Cross-section shape. (Character: "ROD", "TUBE", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", and HAT1 for GROUP+"MSCBML0") See Figure 4-14 .
DIMi	Cross-sectional dimensions. (Real > 0.0)
NSM	Nonstructural mass per unit length. NSM is specified after the last DIMi. (Real; Default = 0.0)

The example shown in [Listing 4-5](#) is the same one used in [Listing 4-3](#), except the PBAR entry is replaced by the PBARL entry. A condensed version of the corresponding output is shown in [Figure 4-15](#). A slight difference in the output can be attributed to the fact that only four significant digits are provided in the PBAR example shown in [Listing 4-3](#). Note also that the order of the stress data recovery points is different in the two examples.

Listing 4-5 CBAR Element Defined by Cross-Sectional Dimension

```

$
$ FILENAME - BAR1N.DAT
$
ID LINEAR, BAR1N
SOL 101
TIME 2
CEND
TITLE = THREE-BAR FRAME MODEL
SUBTITLE = USING BAR DIMENSION FOR PROPERTY DEFINITION
LABEL = POINT LOAD AT GRID POINT 4
LOAD = 1
DISPLACEMENT = ALL
STRESS = ALL
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID    1          -433.   250.    0.          123456
GRID    2           433.   250.    0.          123456
GRID    3            0.   -500.    0.          123456
GRID    4            0.    0.   1000.
$
$
CBAR    1          1          1          4          43.3   -25.    0.
CBAR    2          1          2          4          -43.3  -25.    0.
CBAR    3          1          3          4           0.     1.    0.
$
$ DIMENSIONS FOR RECTANGULAR SECTION
$
PBARL   1          1          BAR
        2.          4.
$
$ MATERIAL PROPERTIES
$
MAT1    1          19.9E4          .3
$
$ POINT LOAD
$
FORCE   1          4          5000.    0.     -1.     0.
$
ENDDATA

```

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		
2	G	.0	.0	.0	.0	.0	.0		
3	G	.0	.0	.0	.0	.0	.0		
4	G	.0	-1.170445E+01	-2.574947E-05	1.447124E-02	.0	.0		

S T R E S S E S I N B A R E L E M E N T S (C B A R)									
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S. -T	M.S. -C
0 1	8.434553E-01	1.054030E+01	-8.434553E-01	-1.054030E+01	4.658372E+02	4.763775E+02	4.552969E+02		
	-2.004751E+00	-1.398322E+00	2.004751E+00	1.398322E+00		4.678419E+02	4.638324E+02		
0 2	-1.054030E+01	-8.434553E-01	1.054030E+01	8.434553E-01	4.658372E+02	4.763775E+02	4.552969E+02		
	1.398322E+00	2.004751E+00	-1.398322E+00	-2.004751E+00		4.678419E+02	4.638324E+02		
0 3	9.696349E+00	-9.696349E+00	-9.696349E+00	9.696349E+00	-9.316785E+02	-9.219821E+02	-9.413748E+02		
	6.066558E-01	-6.066558E-01	-6.066558E-01	6.066558E-01		-9.310718E+02	-9.322851E+02		

Figure 4-15 Displacement and Stress Results for Bar Structure Using PBARL

Model Creation in MSC.Patran. The bar cross-sections can also be created in MSC.Patran, using the steps outlined below.

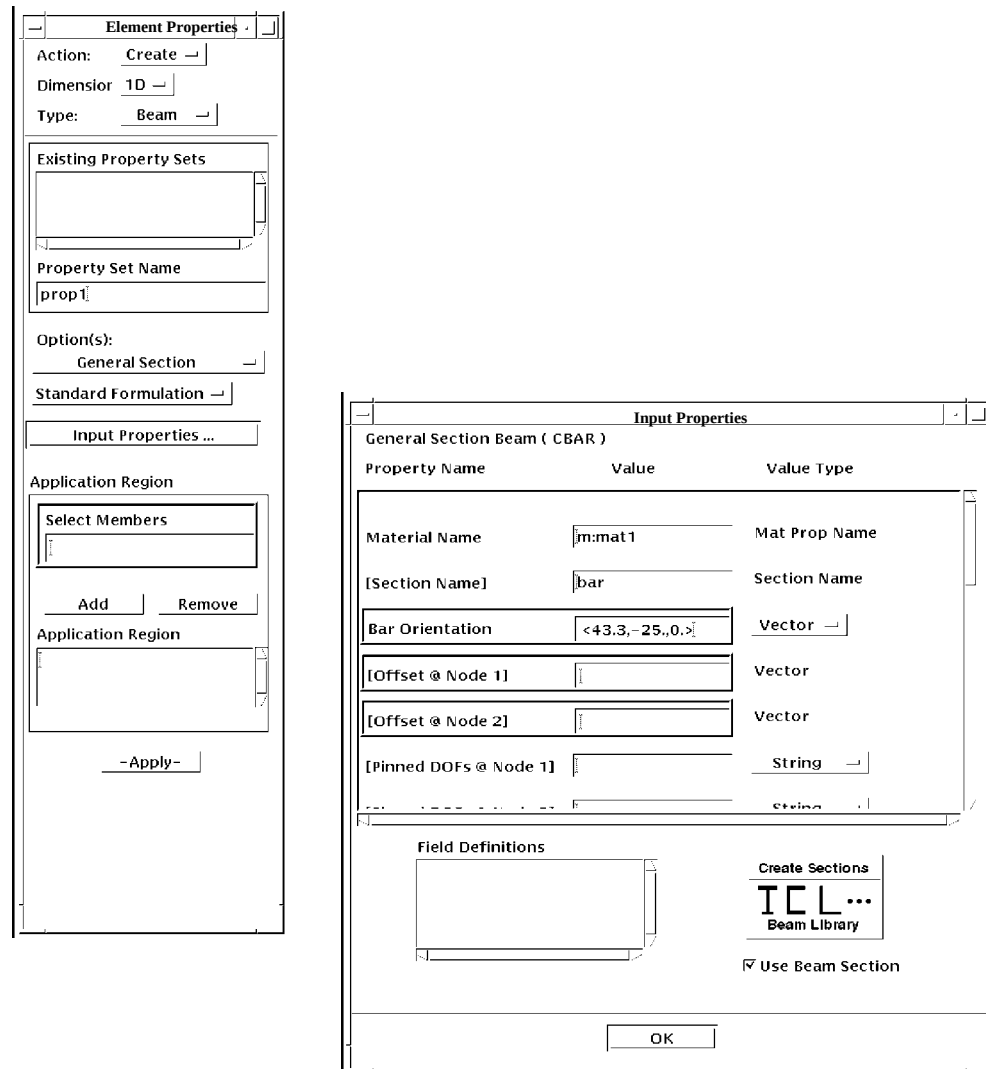


Figure 4-16 Element and Input Property Windows in MSC.Patran

1. To create element properties using the beam cross-section library in MSC.Patran, use the Element Properties form shown in [Figure 4-16](#). Set the Action, Dimension, and Type option menus to Create, 1D, and Beam, respectively. Type in the name for your beam in the Property Set Name field. Under Option(s), use General Section. Click on **Input Properties** (also shown in [Figure 4-16](#)) to access that window.
2. In the Input Properties window, select your material property in Material Name and click on the icon in the lower right hand corner labeled Create Sections to invoke the Beam Library window ([Figure 4-17](#)).

Beam Library

Action: Create

Type: Standard Shape

Existing Sections

*Filter

bar

New Section Name

bar

I

L

T

C

O

●

□

■

□

<

>

Calculate/Display

Write to Report File

OK

Apply

Reset

Cancel

H

W

2.0

4.0

Figure 4-17 Beam Library Window

3. In the Beam Library window, set the Action and Type option menus to Create and Standard Shape, respectively. Type in the name of the cross section in the New Section Name field, or choose section type by scrolling through the icons and selecting one and entering the dimensions for that section type. To verify the input, click on the **Calculate/Display** button, which shows all the section properties that have been generated using that cross section.

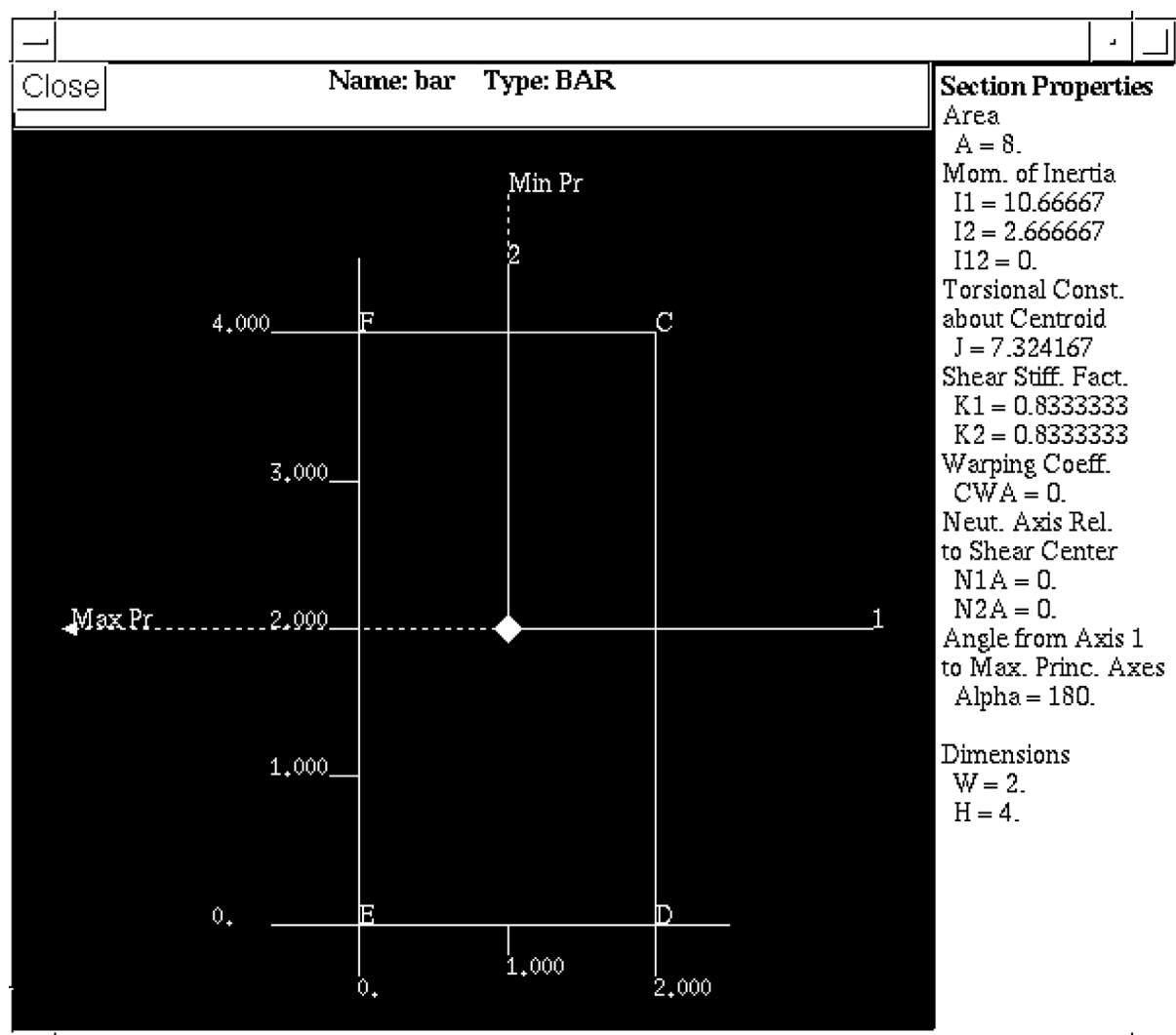


Figure 4-18 Section Display Window

- Since the element y-coordinate corresponds to the 2-axis and the element z-coordinate corresponds to the 1-axis, the vector $\langle 43.3, -25., 0. \rangle$ defines the $\vec{v}$ vector for element number one.

Close the Section Display window ([Figure 4-18](#)); then click on **Apply** to create the property and **OK** to return to the Input Properties window.

- In the Input Properties window, click on the **Use Beam Section** radio button to have MSC.Patran write the MSC.Nastran PBARL Bulk Data entries when the finite element model is generated. If you do not click on **Use Beam Section**, equivalent PBAR properties will be created and optimization on bar dimensions will not be possible. Click on **OK** to return to the Element Properties window.
- In the Element Properties window, insert "element 1" (the element you will apply the properties to) in the Application Region section. At the bottom click on **OK** to accept the data input, and the beam cross-section library definition is complete.
- Repeat steps 4 through 6 for elements number two and three, since each requires a different orientation vector.

8. Write the MSC.Nastran finite element model to a file and the job is ready to be submitted.

The CBAR element assumes that the neutral axis and shear center coincide. For an unsymmetric section, the actual shear center does not coincide with the neutral axis. If this difference is significant, you should use the CBEAM element instead (see the discussion of the CBEAM element later in this section); otherwise, the results may be incorrect.

An additional feature of the CBAR element is the ability to remove some of the connections of individual degrees of freedom from the grid points. This operation is accomplished using the pin flags feature located in the CBAR entry (fields 2 and 3 of the continuation entry). For example, suppose you want to connect two bar elements together with a hinge (or pin joint) as shown in [Figure 4-19](#). This connection can be made by placing an integer 456 in the PB field of CBAR 1 or a 456 in the PA field of CBAR 2.

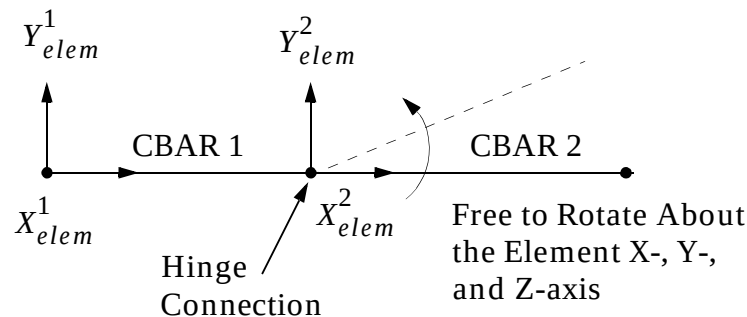


Figure 4-19 A Hinge Connection

The sample bridge model shown in [Figure 4-20](#) is used to illustrate the application of pin flags. In this case, the rotational degree of freedom, θ_z at end B of the braces (grid points 7 and 16) connected to the horizontal span, is released. Note that these degrees of freedoms are referenced in terms of the element coordinate systems. A copy of this input file is shown in [Listing 4-6](#). The deflected shapes for the cases with and without releases are shown in [Figure 4-21](#), and [Figure 4-22](#), respectively. The corresponding abridged stress outputs for the cases with and without releases are shown in [Figure 4-24](#) and [Figure 4-25](#), respectively. Elements 30 and 40 are the two brace elements that are connected to the horizontal span. Note that in the case with releases at ends B, there is no moment transfer to the brace (grid points 7 and 16) at these locations. The moments, however, are transferred across the horizontal span (elements 6, 7, 14, and 15). For clarity, only the pertinent element and grid numbers are shown in [Figure 4-20](#).

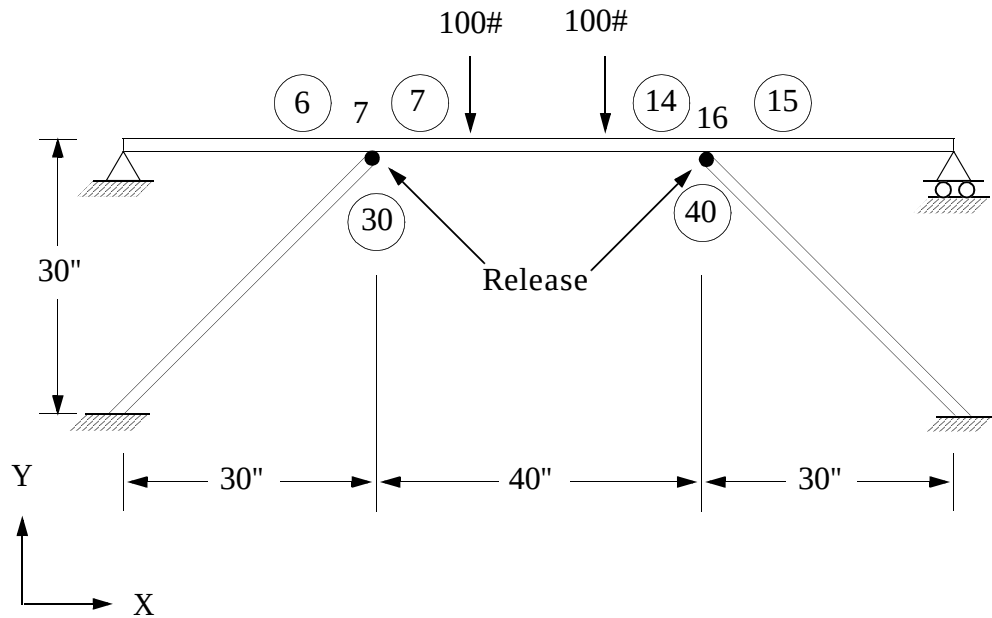


Figure 4-20 Bridge Model Demonstrating the Use of Release

Listing 4-6 Bridge with Release at Brace

```

$
$   bridge1.dat
$
SOL 101
TIME 60
CEND
TITLE = BRIDGE MODEL - RELEASE FOR BRACE AT SPAN INTERSECTION
SUBCASE 1
  SUBTITLE=Default
  SPC = 2
  LOAD = 1
  DISPLACEMENT=ALL
  SPCFORCES=ALL
  STRESS=ALL
BEGIN BULK
PARAM      POST      -1
PBARL      1          1          I
+          A
+          A 1.        1.        1.        .1        .1        .1
MAT1 1      3.E7      .32
CBAR      1          1          1          2          0.        1.        0.
CBAR      2          1          2          3          0.        1.        0.
CBAR      3          1          3          4          0.        1.        0.
CBAR      4          1          4          5          0.        1.        0.
CBAR      5          1          5          6          0.        1.        0.
CBAR      6          1          6          7          0.        1.        0.

```

Listing 4-6 Bridge with Release at Brace (continued)

CBAR	7	1	7	9	0.	1.	0.
CBAR	8	1	9	10	0.	1.	0.
CBAR	9	1	10	11	0.	1.	0.
CBAR	10	1	11	12	0.	1.	0.
CBAR	11	1	12	13	0.	1.	0.
CBAR	12	1	13	14	0.	1.	0.
CBAR	13	1	14	15	0.	1.	0.
CBAR	14	1	15	16	0.	1.	0.
CBAR	15	1	16	18	0.	1.	0.
CBAR	16	1	18	19	0.	1.	0.
CBAR	17	1	19	20	0.	1.	0.
CBAR	18	1	20	21	0.	1.	0.
CBAR	19	1	21	22	0.	1.	0.
CBAR	20	1	22	23	0.	1.	0.
CBAR	21	1	24	25	0.	1.	0.
CBAR	22	1	25	26	0.	1.	0.
CBAR	23	1	26	27	0.	1.	0.
CBAR	24	1	27	28	0.	1.	0.
CBAR	25	1	28	29	0.	1.	0.
CBAR	26	1	29	30	0.	1.	0.
CBAR	27	1	30	31	0.	1.	0.
CBAR	28	1	31	32	0.	1.	0.
CBAR	29	1	32	33	0.	1.	0.
CBAR	30	1	33	7	0.	1.	0.
6							
CBAR	31	1	35	36	0.	1.	0.
CBAR	32	1	36	37	0.	1.	0.
CBAR	33	1	37	38	0.	1.	0.
CBAR	34	1	38	39	0.	1.	0.
CBAR	35	1	39	40	0.	1.	0.
CBAR	36	1	40	41	0.	1.	0.
CBAR	37	1	41	42	0.	1.	0.
CBAR	38	1	42	43	0.	1.	0.
CBAR	39	1	43	44	0.	1.	0.
CBAR	40	1	44	16	0.	1.	0.
6							
\$ Nodes of the Entire Model							
GRID	1		0.	30.	0.		
GRID	2		5.	30.	0.		
GRID	3		10.	30.	0.		
GRID	4		15.	30.	0.		
GRID	5		20.	30.	0.		
GRID	6		25.0000	30.	0.		
GRID	7		30.	30.	0.		
GRID	9		35.	30.	0.		
GRID	10		40.	30.	0.		
GRID	11		45.	30.	0.		
GRID	12		50.	30.	0.		

Listing 4-6 Bridge with Release at Brace (continued)

```

GRID      13      55.      30.      0.
GRID      14      60.      30.      0.
GRID      15      65.      30.      0.
GRID      16      70.      30.      0.
GRID      18      75.      30.      0.
GRID      19      80.      30.      0.
GRID      20      85.      30.      0.
GRID      21      90.      30.      0.
GRID      22      95.      30.      0.
GRID      23     100.      30.      0.
GRID      24       0.       0.       0.
GRID      25       3.       3.       0.
GRID      26       6.       6.       0.
GRID      27       9.       9.       0.
GRID      28     12.0000 12.0000 0.
GRID      29      15.      15.       0.
GRID      30      18.      18.       0.
GRID      31      21.      21.       0.
GRID      32     24.0000 24.0000 0.
GRID      33      27.      27.       0.
GRID      35     100.       0.       0.
GRID      36      97.       3.       0.
GRID      37      94.       6.       0.
GRID      38      91.       9.       0.
GRID      39      88.     12.0000 0.
GRID      40      85.      15.       0.
GRID      41      82.      18.       0.
GRID      42      79.      21.       0.
GRID      43      76.     24.0000 0.
GRID      44      73.      27.       0.
$ Loads for Load Case : Default
SPCADD    2        1        3        4
$ Displacement Constraints of Load Set : disp1
SPC1      1      12345    1
$ Displacement Constraints of Load Set : disp2
SPC1      3      123456  24      35
$ Displacement Constraints of Load Set : disp3
SPC1      4      2345    23
$ Nodal Forces of Load Set : load1
FORCE     1      10      0      100.    0.    -1.    0.
FORCE     1      14      0      100.    0.    -1.    0.
ENDDATA

```

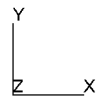
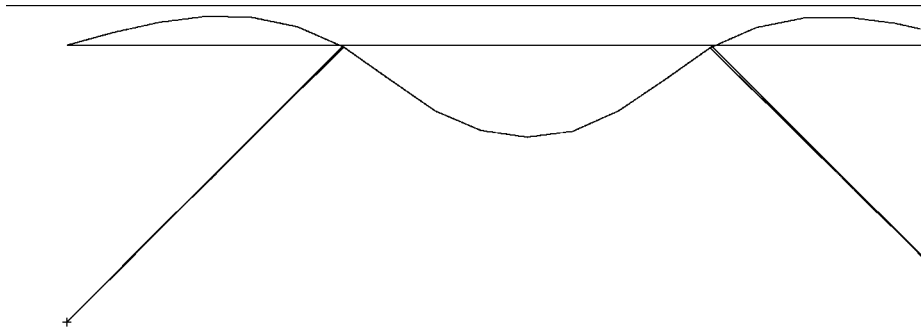


Figure 4-21 Deflected Shape of Bridge with Release at Brace

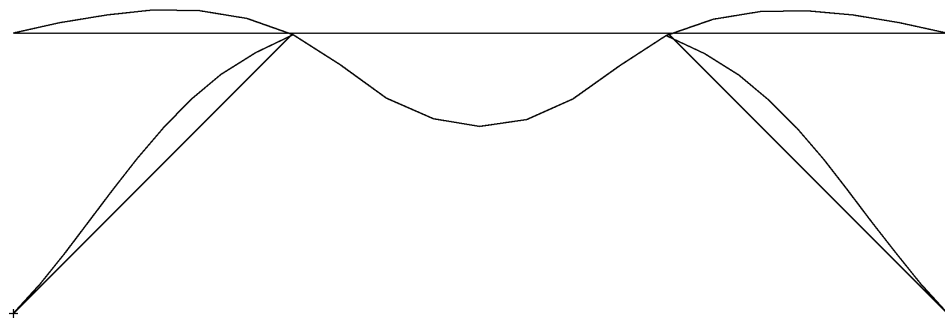


Figure 4-22 Deflected Shape of Bridge without Release at Brace

ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
0 1	.0	.0	.0	.0	1.298114E+00	1.298114E+00	1.298114E+00	
	9.251410E+02	-9.251410E+02	-9.251410E+02	9.251410E+02		9.264391E+02	-9.238429E+02	
0 2	9.251410E+02	-9.251410E+02	-9.251410E+02	9.251410E+02	1.298114E+00	9.264391E+02	-9.238429E+02	
	1.850282E+03	-1.850282E+03	-1.850282E+03	1.850282E+03		1.851580E+03	-1.848984E+03	
	.	.	.	.				
0 6	4.625705E+03	-4.625705E+03	-4.625705E+03	4.625705E+03	1.298114E+00	4.627003E+03	-4.624407E+03	
	5.550846E+03	-5.550846E+03	-5.550846E+03	5.550846E+03		5.552144E+03	-5.549547E+03	
0 7	5.550846E+03	-5.550846E+03	-5.550846E+03	5.550846E+03	-4.153344E+02	5.135511E+03	-5.966180E+03	
	-1.805041E+01	1.805041E+01	1.805041E+01	-1.805041E+01		-3.972839E+02	-4.333848E+02	
	.	.	.	.				
0 14	-4.863751E+01	4.863751E+01	4.863751E+01	-4.863751E+01	-4.153344E+02	-3.666968E+02	-4.639719E+02	
	5.510063E+03	-5.510063E+03	-5.510063E+03	5.510063E+03		5.094729E+03	-5.925397E+03	
0 15	5.510063E+03	-5.510063E+03	-5.510063E+03	5.510063E+03	.0	5.510063E+03	-5.510063E+03	
	4.591719E+03	-4.591719E+03	-4.591719E+03	4.591719E+03		4.591719E+03	-4.591719E+03	
	.	.	.	.				
0 30	2.083161E+00	-2.083161E+00	-2.083161E+00	2.083161E+00	-5.893649E+02	-5.872817E+02	-5.914481E+02	
	-7.759377E-08	7.759377E-08	7.759377E-08	-7.759377E-08		-5.893649E+02	-5.893649E+02	
	.	.	.	.				
0 40	4.021125E+00	-4.021125E+00	-4.021125E+00	4.021125E+00	-5.876757E+02	-5.836545E+02	-5.916968E+02	
	-3.032399E-08	3.032399E-08	3.032399E-08	-3.032399E-08		-5.876757E+02	-5.876757E+02	

Figure 4-23 Abridged Stress Output of Bridge with Release at Brace

ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
0 1	1.265061E-12	-1.265061E-12	-1.265061E-12	1.265061E-12	2.220387E+00	2.220387E+00	2.220387E+00	
	5.686420E+02	-5.686420E+02	-5.686420E+02	5.686420E+02		5.708624E+02	-5.664216E+02	
0 2	5.686420E+02	-5.686420E+02	-5.686420E+02	5.686420E+02	2.220387E+00	5.708624E+02	-5.664216E+02	
	1.137284E+03	-1.137284E+03	-1.137284E+03	1.137284E+03		1.139504E+03	-1.135064E+03	
	.	.	.	.				
0 6	2.843210E+03	-2.843210E+03	-2.843210E+03	2.843210E+03	2.220387E+00	2.845430E+03	-2.840990E+03	
	3.411852E+03	-3.411852E+03	-3.411852E+03	3.411852E+03		3.414073E+03	-3.409632E+03	
0 7	6.634688E+03	-6.634688E+03	-6.634688E+03	6.634688E+03	-4.433274E+02	6.191360E+03	-7.078016E+03	
	1.062523E+03	-1.062523E+03	-1.062523E+03	1.062523E+03		6.191954E+02	-1.505850E+03	
	.	.	.	.				
0 14	1.012320E+03	-1.012320E+03	-1.012320E+03	1.012320E+03	-4.433274E+02	5.689928E+02	-1.455648E+03	
	6.567751E+03	-6.567751E+03	-6.567751E+03	6.567751E+03		6.124424E+03	-7.011079E+03	
0 15	3.384410E+03	-3.384410E+03	-3.384410E+03	3.384410E+03	.0	3.384410E+03	-3.384410E+03	
	2.820342E+03	-2.820342E+03	-2.820342E+03	2.820342E+03		2.820342E+03	-2.820342E+03	
	.	.	.	.				
0 30	2.742704E+03	-2.742704E+03	-2.742704E+03	2.742704E+03	-5.937782E+02	2.148926E+03	-3.336482E+03	
	3.222836E+03	-3.222836E+03	-3.222836E+03	3.222836E+03		2.629058E+03	-3.816614E+03	
	.	.	.	.				
0 40	2.711180E+03	-2.711180E+03	-2.711180E+03	2.711180E+03	-5.912410E+02	2.119939E+03	-3.302421E+03	
	3.183342E+03	-3.183342E+03	-3.183342E+03	3.183342E+03		2.592101E+03	-3.774583E+03	

Figure 4-24 Abridged Stress Output of Bridge without Release at Brace

The CBEAM Element

The CBEAM element provides all of the capabilities of the CBAR element discussed in the previous section, plus the following additional capabilities:

- Different cross-sectional properties may be defined at both ends and at as many as nine intermediate locations along the length of the beam.

- The neutral axis and shear center do not need to coincide. The feature is important for unsymmetric sections.
- The effect of cross-sectional warping on torsional stiffness is included.
- The effect of taper on transverse shear stiffness (shear relief) is included.
- A separate axis for the center of nonstructural mass may be included.
- Distributed torsional mass moment of inertia is included for dynamic analysis.

The format, as shown below for the CBEAM entry, is similar to that of the CBAR entry. The only difference is the addition of the SA and SB fields located in fields 2 and 3 of the second continuation entry. The SA and SB fields are scalar point entries ID used for warping terms. (Warping is discussed later in this section.) The format of the CBEAM entry is as follows:

CBEAM

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

Field	Contents
EID	Unique element identification number.
PID	Property identification number of PBEAM or PBCOMP entry.
GA, GB	Grid point identification numbers of connection points.
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA.
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. The vector $\vec{v}$ is then transferred to end A.
PA, PB	Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees of freedom of the beam. The degrees of freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see the following figure). The beam must have stiffness associated with the PA and PB degrees of freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique integers 1 through 6 with no embedded blanks).

W1A, W2A, W3A Components of offset vectors, measured in the displacement coordinate systems at grid points A and B, from the grid points to the end points of the axis of shear center.

W2A, W2B, W3B

SA, SB Scalar or grid point identification numbers for the ends A and B, respectively. The degrees of freedom at these points are the warping variables $d\theta / dx$.

The PBEAM entry, on the other hand, may be substantially different than the PBAR entry, depending on what features are used. The format of the PBEAM entry is as follows:

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

The next two continuations are repeated for each intermediate station, and SO and X/XB must be specified (see the [MSC.Nastran Quick Reference Guide](#)).

1	2	3	4	5	6	7	8	9	10
	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	DI	D2	E1	E2	F1	F2	

The last two continuations are:

1	2	3	4	5	6	7	8	9	10
	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Field	Contents
PID	Property identification number.
MID	Material identification number.
A(A)	Area of the beam cross section at end A.
I1(A)	Area moment of inertia at end A for bending in Plane 1 about the neutral axis.
I2(A)	Area moment of inertia at end A for bending in Plane 2 about the neutral axis.
I12(A)	Area product of inertia at end A.
J(A)	Torsional stiffness parameter at end A.
NSM(A)	Nonstructural mass per unit length at end A.

Ci(A), Di(A) Ei(A), Fi(A)	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery.
SO	Stress output request option. “YES”Stresses recovered at points Ci, Di, Ei, and Fi on the next continuation. “YESA”Stresses recovered at points with the same y and z location as end A. “NO”No stresses or forces are recovered.
X / XB	Distance from end A in the element coordinate system divided by the length of the element.
A, I1, I2, I12, J, NSM	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x.
Ci, Di, Ei, Fi	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center for the cross section located at X / XB. The values are fiber locations for stress data recovery.
K1, K2	Shear stiffness factor K in $K \cdot A \cdot G$ for Plane 1 and Plane 2.
S1, S2	Shear relief coefficient due to taper for Plane 1 and Plane 2.
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B.
CW(A), CW(B)	Warping coefficient for end A and end B.
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B.
N1(A), N2(A), N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B.

The coordinate system for the CBEAM element, shown in [Figure 4-25](#), is similar to that of the CBAR element. The only difference is that the element x-axis for the CBEAM element is along the shear center of the CBEAM. The neutral axis and the nonstructural mass axis may be offset from the elemental x-axis. (For the CBAR element, all three are coincident with the x-axis.) The vector $\vec{v}$ is defined in the the same manner as it is for the CBAR element.

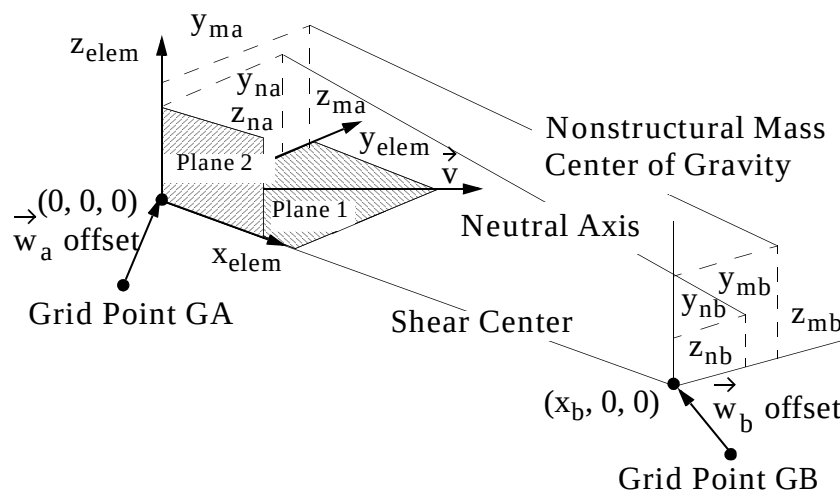


Figure 4-25 CBEAM Element Geometry System

The CBEAM element presents you with more options than any other element in the MSC.Nastran element library. It is not expected that you will employ all of the features of the CBEAM element at the same time. Therefore, you may omit most of the data fields corresponding to the features that are not being used. For those data fields that are left blank, a default value is used, as described in the [MSC.Nastran Quick Reference Guide](#).

An exception to this rule is the data fields for A, I1, and I2. If any of these fields is left blank, a fatal message occurs. While this is acceptable for the CBAR element, it is not acceptable for the CBEAM element. The difference is due to the way that the element stiffness matrix is generated. For the CBAR element, the element stiffness matrix is generated directly from the input data. For instance, if I1 for a CBAR element is zero, then the corresponding element stiffness matrix term is null, which is not necessarily a problem. On the other hand, the CBEAM element uses the input data to generate an element flexibility matrix, which must be inverted to produce the element stiffness matrix. Therefore, positive values for A, I1, and I2 must be entered.

As an example, consider the truss model shown earlier in [Figure 4-5](#). To convert a CBAR model to a CBEAM model, only three changes are needed. Change the CBAR name to CBEAM, change the PBAR name to a PBEAM, and change the location of J from field 7 of the PBAR entry to field 8 of the PBEAM entry.

One difference between the CBAR element and the CBEAM element, that is not obvious, is the default values used for the transverse shear flexibility. For the CBAR element, the default values for K1 and K2 are infinite, which is equivalent to zero transverse shear flexibility. For the CBEAM element, the default values for K1 and K2 are both 1.0, which includes the effect of transverse shear in the elements. If you want to set the transverse shear flexibility to zero, which is the same as the CBAR element, use a value of 0.0 for K1 and K2.

The resulting stress output is shown in [Figure 4-26](#).

S T R E S S E S I N B E A M E L E M E N T S (C B E A M)											
ELEMENT-ID	GRID	STAT DIST/ LENGTH		SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S. -T	M.S. -C
1	1	0.000		4.666807E+02	4.763772E+02	4.552972E+02	4.649937E+02	4.763772E+02	4.552972E+02		
	4	1.000		4.638323E+02	4.644395E+02	4.672348E+02	4.678420E+02	4.678420E+02	4.638323E+02		
2	2	0.000		4.552972E+02	4.649937E+02	4.666807E+02	4.763772E+02	4.763772E+02	4.552972E+02		
	4	1.000		4.672348E+02	4.678420E+02	4.638323E+02	4.644395E+02	4.678420E+02	4.638323E+02		
3	3	0.000		-9.219825E+02	-9.413745E+02	-9.219825E+02	-9.413745E+02	-9.219825E+02	-9.413745E+02		
	4	1.000		-9.310710E+02	-9.322858E+02	-9.310710E+02	-9.322858E+02	-9.310710E+02	-9.322858E+02		

Figure 4-26 CBEAM Stress Output

As shown in **Figure 4-26**, the stress output for the CBEAM element is not the same as that for the CBAR element. For CBAR elements, the SAI and SBI columns are the stresses due to bending only, and the axial stresses is given in a separate column. For the CBEAM element, however, the values at SXC, SXD, SXE, and SXF are the stresses due to the bending and axial forces on the CBEAM at stress locations C, D, E, and F on the cross section. Stress recovery is performed at the end points and any intermediate location is defined on the PBEAM entry.

As another example for the beam element that uses more of the CBEAM features, consider the tapered beam shown in **Figure 4-27**.

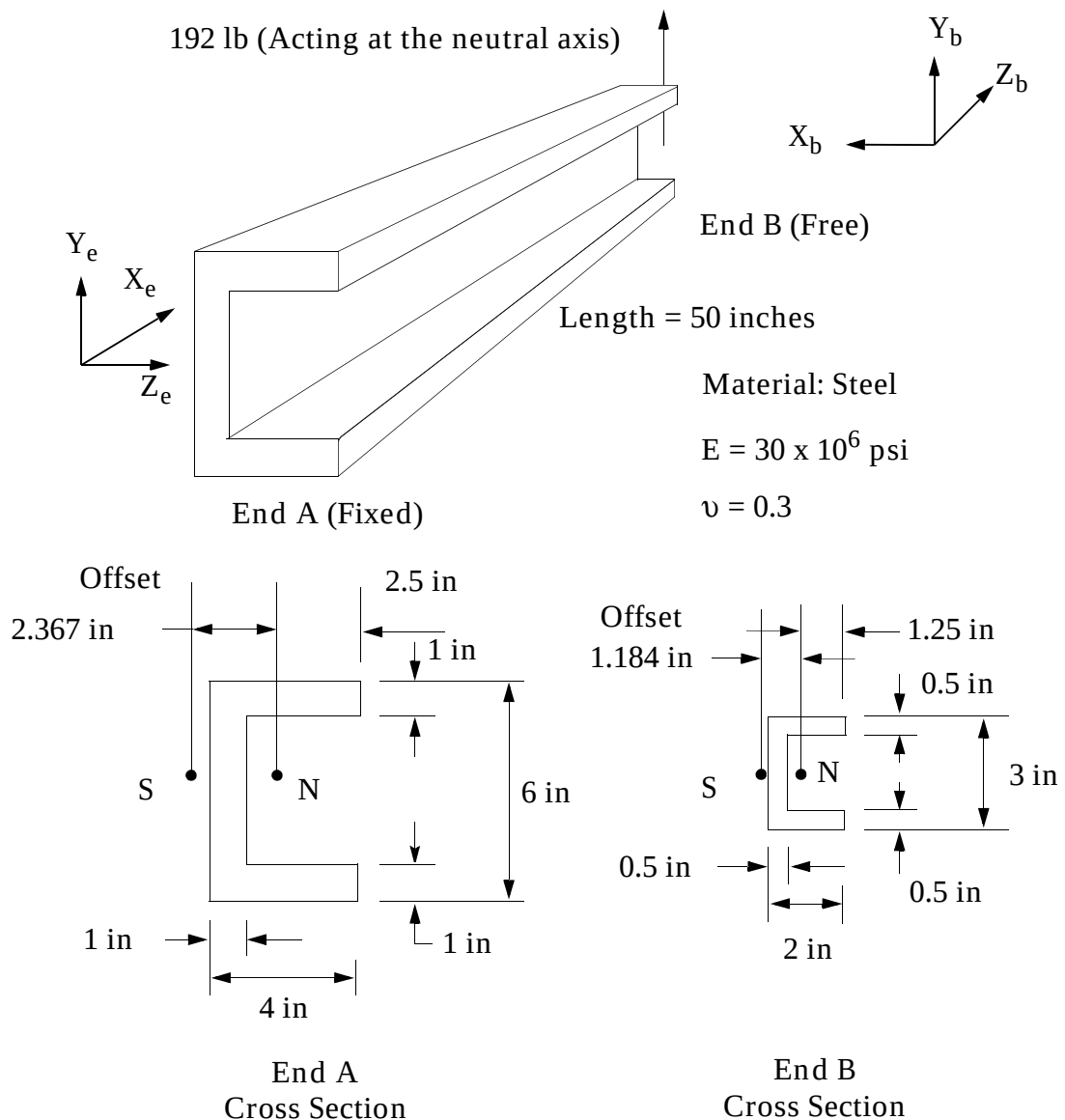


Figure 4-27 Tapered Beam Example

Because the cross section is an open channel section, the shear center of the beam does not coincide with the neutral axis. In general, you must decide in your model planning whether having noncoincident shear and neutral axes is significant for your application. Since for this example the entire structure is a single tapered beam, modeling the offset is important. There are two methods that can be used to model the offset. The first method is to place the shear axis on the line between the end grid points 1 and 2. In this case the neutral axis is offset from the shear axis using the y_{na} , z_{na} , y_{nb} , and z_{nb} offsets (see [Figure 4-25](#)) entered on the PBEAM entry.

The second method to model the noncoincident axes in this example is to place the neutral axis on the line extending from grid point 1 to grid point 2. In this case, the offsets WA and WB, entered on the CBEAM entry, are used to position the shear axis at the appropriate location. The y_{na} , z_{na} , y_{nb} , and z_{nb} offsets entered on the PBEAM entry are then used to position the neutral axis to the appropriate position.

Both methods represent the same CBEAM element but are positioned differently with respect to the grid points. The first method requires one set of offset values to be entered, and the second method requires two sets of offset values. The forces are applied to the grid points. By using the second method for this problem, the loads are then applied at the neutral axis instead of the shear center axis. By doing so, you can observe the twisting of the beam due to a pure vertical load.

The Bulk Data is shown in [Listing 4-7](#). The offsets $WA = (2.367, 0.0, 0.0)$ and $WB = (1.184, 0.0, 0.0)$ are entered on the CBEAM entry to define the locations of the shear axis. The neutral axis is offset from the shear axis using the offsets $y_{na} = 0.0$, $z_{na} = 2.367$, $y_{nb} = 0.0$, and $z_{nb} = 1.184$ entered on the PBEAM entry. It may appear that the offsets do not accomplish the desired goal of placing the neutral axis in the right location because all of the offset are positive. However, keep in mind that the shear center offsets (WA and WB) are in the displacements coordinate system, measured from GA and GB, respectively. The neutral axis offsets are in the CBEAM's element coordinate system.

Nine intermediate stations are used to model the taper. Since the properties are not a linear function of the distance along the beam (A is, but I1 and I2 are not), it is necessary to compute the cross-sectional properties for each of the stations. The properties for the nine stations are entered on the PBEAM entry.

To demonstrate all of the capabilities of the CBEAM element, beam warping is included; however, beam warping is not significant for this problem.

You should note the locations of the stress recovery locations on the PBEAM entry. The stress recovery locations are entered with respect to the shear axis, not the neutral axis (i.e., they are input with respect to the element coordinate system). MSC.Nastran computes the distance from the neutral axis internally for the stress recovery.

Listing 4-7 Tapered Beam Input File

```

$
$  FILENAME - BEAM2.DAT
$
ID      LINEAR, BEAM2
SOL      101
TIME     5
CEND
TITLE = TAPERED BEAM MODEL
DISP = ALL
STRESS = ALL
FORCE = ALL
LOAD = 1
SPC = 1
BEGIN BULK
PARAM    POST      0
PARAM    AUTOSPC YES
$
GRID      1          0.0      0.0      0.0
GRID      2          0.0      0.0     50.0
SPOINT    101      102
SPC        1        1      123456  0.0
CBEAM      1        11      1        2        0.        1.        0.
          2.367      0.        0.        1.184      0.        0.
          101      102
$
$      2      3      4      5      6      7      8      9
$
PBEAM      11      21  12.000  56.000  17.000          3.930
          -3.000      .867  -3.000      4.867      3.000      4.867      3.000      .867
        YES      .100  10.830  45.612  13.847          3.201
          -2.850      .824  -2.850      4.624      2.850      4.624      2.850      .824
        YES      .200      9.720  36.742  11.154          2.579
          -2.700      .780  -2.700      4.380      2.700      4.380      2.700      .780
        YES      .300      8.670  29.232      8.874          2.052
          -2.550      .737  -2.550      4.137      2.550      4.137      2.550      .737
        YES      .400      7.680  22.938      6.963          1.610
          -2.400      .694  -2.400      3.894      2.400      3.894      2.400      .694
        YES      .500      6.750  17.719      5.379          1.244
          -2.250      .650  -2.250      3.650      2.250      3.650      2.250      .650
        YES      .600      5.880  13.446      4.082          .944
          -2.100      .607  -2.100      3.407      2.100      3.407      2.100      .607
        YES      .700      5.070      9.996      3.035          .702
          -1.950      .564  -1.950      3.164      1.950      3.164      1.950      .564
        YES      .800      4.320      7.258      2.203          .509
          -1.800      .520  -1.800      2.920      1.800      2.920      1.800      .520
        YES      .900      3.630      5.124      1.556          .360
          -1.650      .477  -1.650      2.677      1.650      2.677      1.650      .477
        YES      1.000      3.000      3.500      1.062          .246
          -1.500      .434  -1.500      2.434      1.500      2.434      1.500      .434
          .241          -.666          0.
          0.        2.367          0.        1.184
$
MAT1      21      3.+7          .3
$
FORCE      1      2          192.      0.        1.        0.
$
ENDDATA

```

The displacement results, as shown in **Figure 4-28**, include the displacements of the grid points at the end of the CBEAM and scalar points 101 and 102. As mentioned previously, the force is applied directly to the grid point; therefore, the force acts at the neutral axis of the beam. Since the shear center is offset from the neutral axis, a loading of this type should cause the element to twist. This result can be observed in the R3 displacement, which represents the twist of the beam. If the shear center is not offset from the neutral axis, R3 will be zero. The displacements of scalar points 101 and 102 represent the twist due to the warping at ends A and B, respectively.

The forces in the beam are shown in **Figure 4-29**. The total torque and the warping torque acting along the beam is printed. As can be seen, the warping for this case is negligible. Note that the inclusion of warping does not affect any of the other forces in the CBEAM element. The stress recovery output is shown in **Figure 4-30**. The stress output shows only the longitudinal stress; hence, any stress due to torsional or warping is not included.

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	
2	G	.0	1.181932E-02	.0	-3.621016E-04	.0	-1.170325E-03	
101	S	2.322214E-05	2.322214E-05					

Figure 4-28 Displacement Output for the Tapered Beam

F O R C E S I N B E A M E L E M E N T S (C B E A M)									
		STAT DIST/ LENGTH	- BENDING MOMENTS -		- WEB SHEARS -		AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
ELEMENT-ID	GRID		PLANE 1	PLANE 2	PLANE 1	PLANE 2			
0	1								
	1	.000	9.608063E+03	.0	3.199436E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.100	8.647795E+03	.0	3.071564E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.200	7.687526E+03	.0	2.943692E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.300	6.727258E+03	.0	2.815820E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.400	5.766989E+03	.0	2.687948E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.500	4.806720E+03	.0	2.560076E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.600	3.846451E+03	.0	2.432204E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.700	2.886183E+03	.0	2.304332E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.800	1.925914E+03	.0	2.176460E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.900	9.656460E+02	.0	2.048588E+02	.0	.0	-2.272644E+02	-3.206116E-16
	2	1.000	5.377076E+00	.0	1.920716E+02	.0	.0	-2.272644E+02	-3.206116E-16

Figure 4-29 Force Output for the Tapered Beam

S T R E S S E S I N B E A M E L E M E N T S (C B E A M)									
		STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S. - T M.S. - C
ELEMENT-ID	GRID								
1									
	1	.000	5.147177E+02	5.147177E+02	-5.147177E+02	-5.147177E+02	5.147177E+02	-5.147177E+02	
	0	.100	5.403450E+02	5.403450E+02	-5.403450E+02	-5.403450E+02	5.403450E+02	-5.403450E+02	
	0	.200	5.649208E+02	5.649208E+02	-5.649208E+02	-5.649208E+02	5.649208E+02	-5.649208E+02	
	0	.300	5.868400E+02	5.868400E+02	-5.868400E+02	-5.868400E+02	5.868400E+02	-5.868400E+02	
	0	.400	6.033994E+02	6.033994E+02	-6.033994E+02	-6.033994E+02	6.033994E+02	-6.033994E+02	
	0	.500	6.103686E+02	6.103686E+02	-6.103686E+02	-6.103686E+02	6.103686E+02	-6.103686E+02	
	0	.600	6.007398E+02	6.007398E+02	-6.007398E+02	-6.007398E+02	6.007398E+02	-6.007398E+02	
	0	.700	5.630309E+02	5.630309E+02	-5.630309E+02	-5.630309E+02	5.630309E+02	-5.630309E+02	
	0	.800	4.776310E+02	4.776310E+02	-4.776310E+02	-4.776310E+02	4.776310E+02	-4.776310E+02	
	0	.900	3.109516E+02	3.109516E+02	-3.109516E+02	-3.109516E+02	3.109516E+02	-3.109516E+02	
	2	1.000	2.304461E+00	2.304461E+00	-2.304461E+00	-2.304461E+00	2.304461E+00	-2.304461E+00	

Figure 4-30 Stress Output for the Tapered Beam

PBEAML-- An Alternate and Convenient Method for Defining CBEAM Cross Sections

Similar to the CBAR element, you can define the property for the CBEAM element by specifying the cross-sectional dimensions (DIM1, DIM2, etc.) instead of the cross-sectional properties (A, I, etc.) for the cross sections shown in [Figure 4-31](#). The PBEAML entry is used for this purpose and the format for the PBEAML entry is as follows:

Format:

(Note: n = number of dimensions and m = number of intermediate stations)

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	GROUP	TYPE					
	DIM1(A)	DIM2(A)	-etc.-	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	DIM1(1)	
	DIM2(1)	-etc.-	DIMn(1)	NSM(1)	SO(2)	X(2)/XB	DIM1(2)	DIM2(2)	
	-etc.-	DIMn(2)	-etc.-	NSM(m)	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	
		NSM(B)							

Example:

PBEAML	99	21		T					
	12.	14.8	2.5	2.6		NO	0.4	6.	
	7.	1.2	2.6		YES	0.6	6.	7.8	
	5.6	2.3		YES					

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Character; Default = "MSCBML0")
TYPE	Cross-section shape. See Remark 4. (Character: "ROD", "TUBE", "L", "I", "CHAN", "T", "BOX", "BAR", "DBOX", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1" for GROUP = "MSCBML0")
DIMi(j)	Cross-section dimensions at end A, intermediate station j and end B. (Real > 0.0 for GROUP = "MSCBML0")
NSM(j)	Nonstructural mass per unit length. (Default = 0.0)
SO(j),SO(B)	Stress output request option for intermediate station j and end B. (Character; Default = "YES")

Field	Contents
	YES: Stresses recovered at all points on next continuation and shown in Figure 4-31 as C, D, E, and F.
	NO: No stresses or forces are recovered.
X(j)/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (Real>0.0; Default = 1.0)

Remarks:

1. For structural problems, PBEAML entries must reference a MAT1 material entry.
 2. PID must be unique with respect to all other PBEAM and PBEAML property identification numbers.
 3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
 4. See the PBEAM entry description for a discussion of beam-element geometry.
 5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j)/XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.
 6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is "MSCBML0". Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry. See "[Building and Using the Sample Programs](#)" on page 251 of the *MSC.Nastran 2004 Installation and Operations Guide* for a discussion of how to include a user-defined beam library.
 7. For GROUP = "MSCBML0", the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in [Figure 4-31](#). The element coordinate system is located at the shear center.
 8. A function of this entry is to derive an equivalent PBEAM entry. Any sorted echo request will also cause printout and / or punch of the derived PBEAM.
 9. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words.
- None of these limits are exceeded with the MSC beam library, but a user defined beam library could.

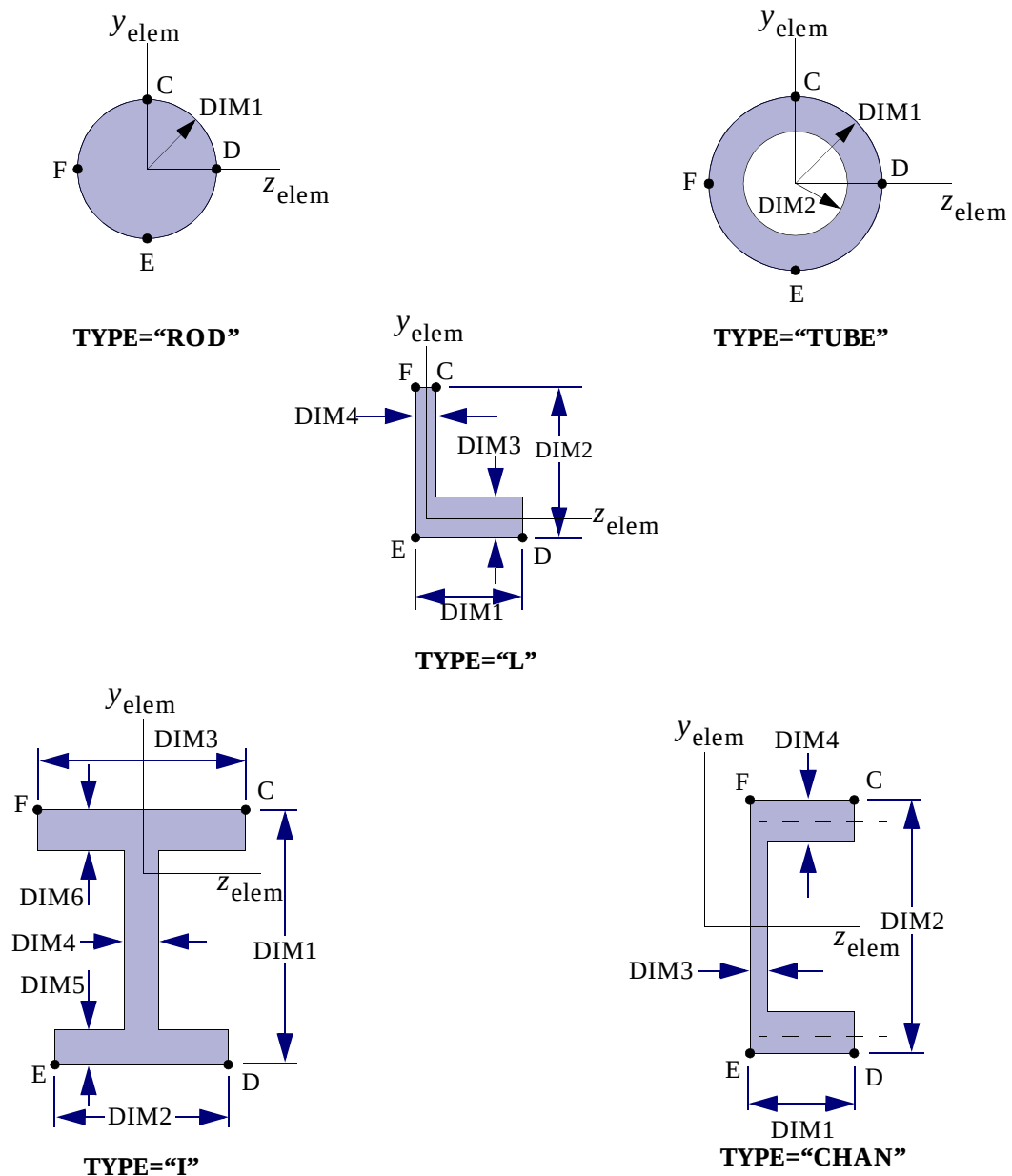


Figure 4-31 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

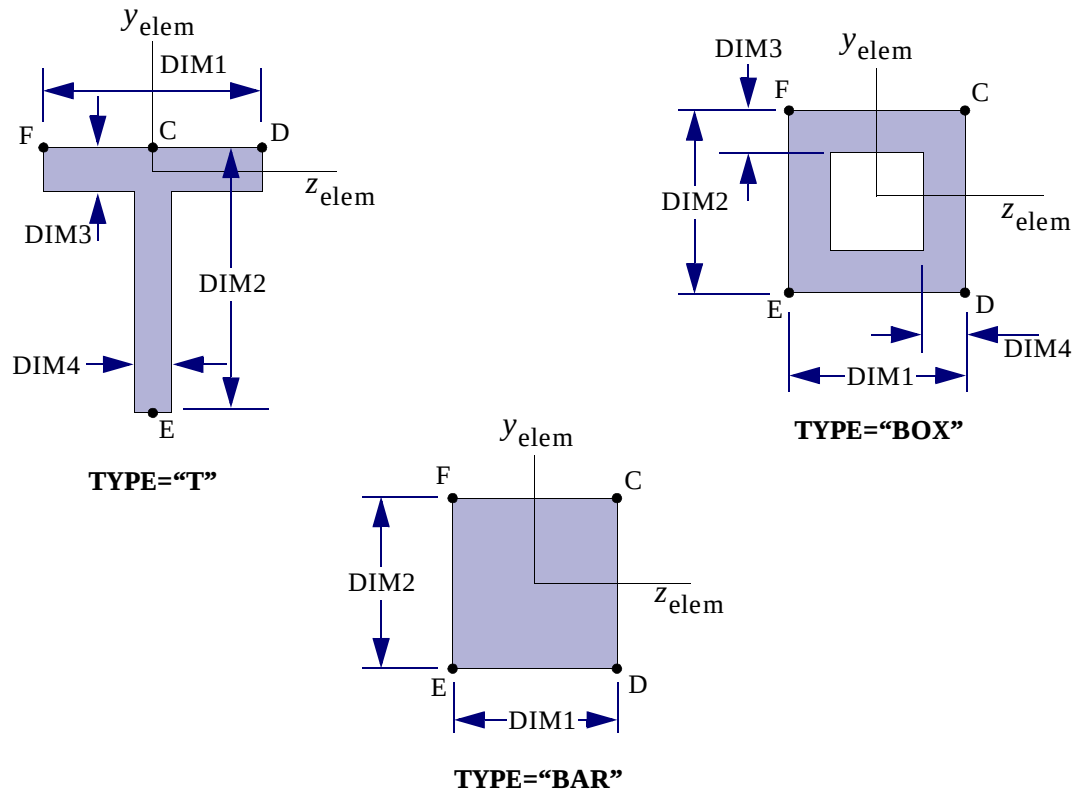
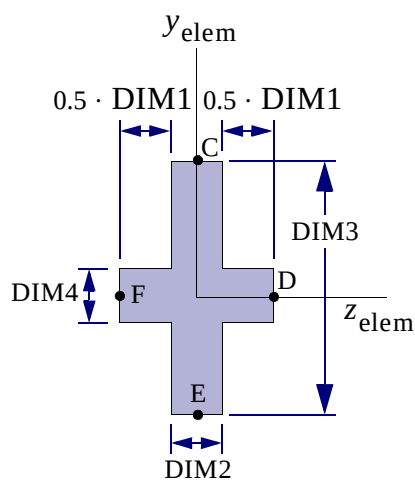
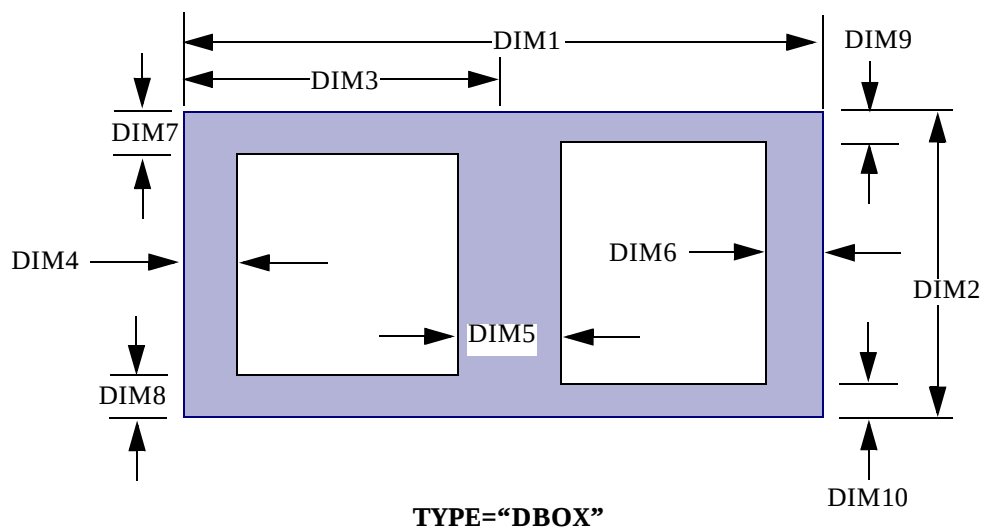
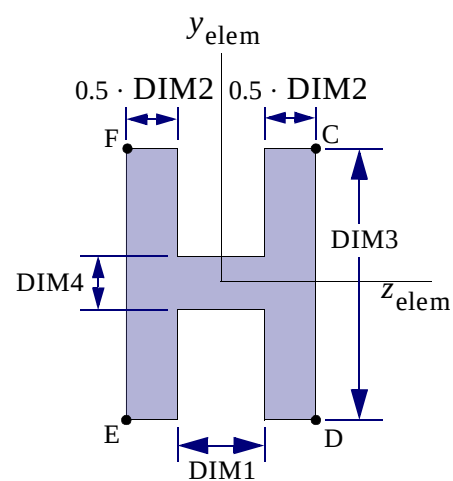


Figure 4-31 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)



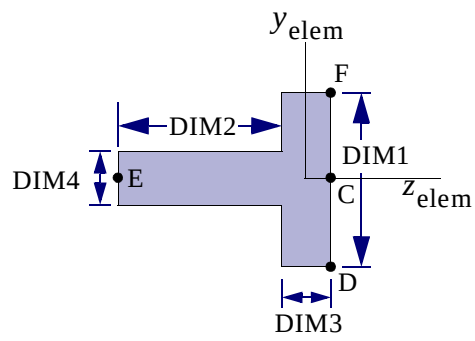
TYPE="CROSS"



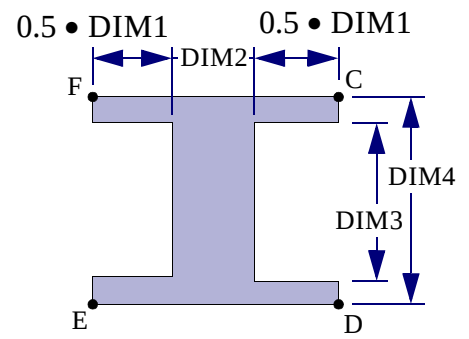
TYPE="H"

Figure 4-31 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

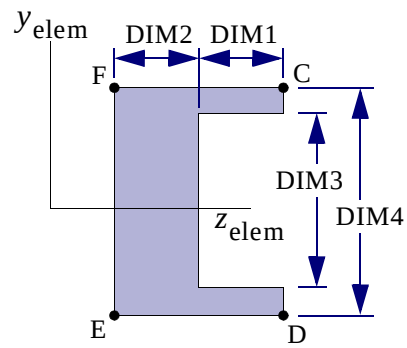
Figure 4-31 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)



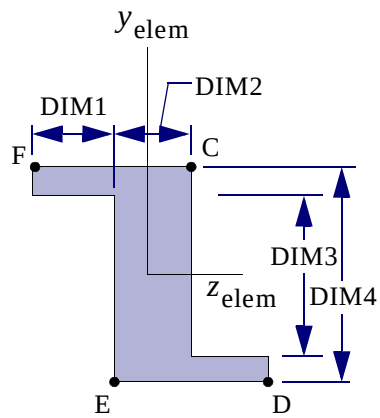
TYPE="T1"



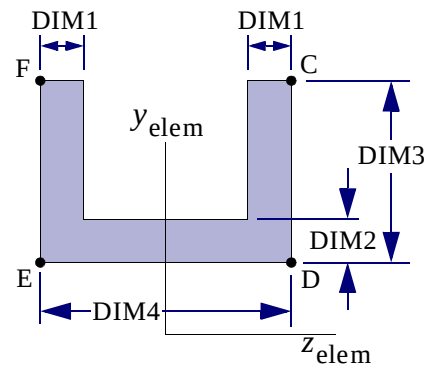
TYPE="I1"



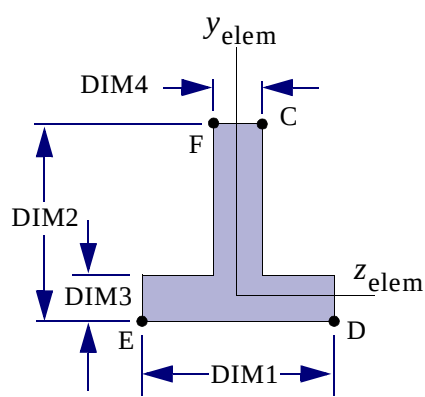
TYPE="CHAN1"



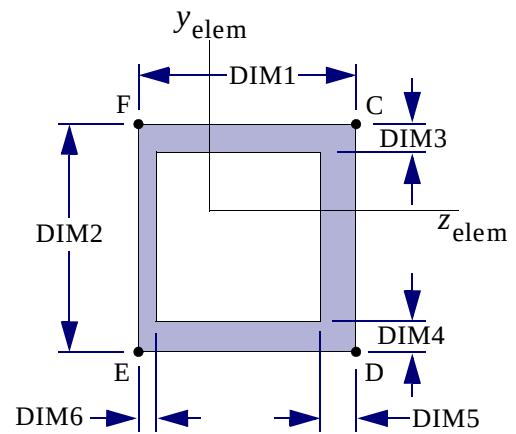
TYPE="Z"



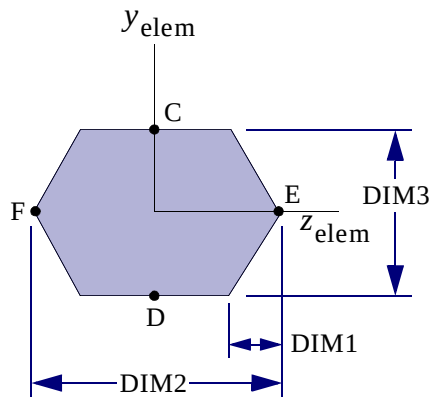
TYPE="CHAN2"



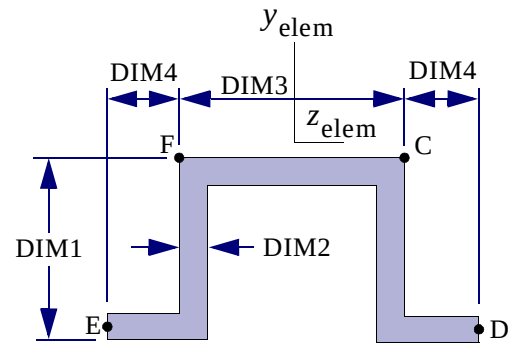
TYPE="T2"



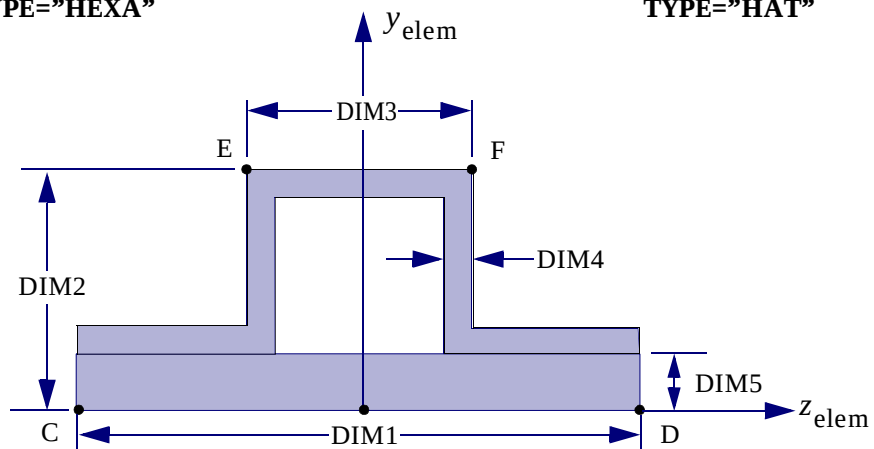
TYPE="BOX1"



TYPE="HEXA"



TYPE="HAT"



TYPE="HAT1"

Figure 4-31 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

The problem shown in [Figure 4-27](#) is rerun using the PBEAML entry. Since the cross section geometry is a channel section, the TYPE field (field five) on the PBEAML entry is assigned the value “CHAN”. Four dimensional values are required at each station that output is desired, or cross sectional properties that cannot be interpolated linearly between the values at the two ends of the CBEAM element. For this example, since only the dimensional values for the two end points and at the middle of the CBEAM element are provided, only output at these locations are available. The complete input and partial output files are shown in [Listing 4-8](#) and [Figure 4-33](#), respectively.

In this case, the values 4.0, 6.0, 1.0, and 1.0 on the first continuation entry represent DIM1, DIM2, DIM3, and DIM4, respectively at end A. The values YES, 0.5, 3.0, 4.5, 0.75, and 0.75 on the first and second continuation entries represent stress output request, value of $X(1)/X_B$, DIM1 at $X(1)/X_B$, DIM2 at $X(1)/X(B)$, DIM3 at $X(1)/X_B$, and DIM4 at $X(1)/X(B)$, respectively at $X(1)/X(B) = 0.5$. The values YES, 1.0, 2.0, 3.0 0.5, and 0.5 on the second and third continuation entries represent stress output request, end B, DIM1 at end B, DIM2 at end B, DIM3 at end B, and DIM4 at end B, respectively.

Listing 4-8 CBEAM Element Defined by Cross-Sectional Dimension

```

$
$  FILENAME - BEAM2N.DAT
$
ID      LINEAR, BEAM2N
SOL      101
TIME      5
CEND
TITLE = TAPERED BEAM MODEL
SUBTITLE = CROSS-SECTION DEFINED BY CHARACTERISTIC DIMENSIONS
DISP = ALL
STRESS = ALL
FORCE = ALL
LOAD = 1
SPC = 1
BEGIN BULK
PARAM    AUTOSPC YES
$
GRID      1              0.0      0.0      0.0
GRID      2              0.0      0.0      50.0
SPOINT    101          102
SPC        1            1      123456  0.0
CBEAM      1            11      1        2      0.        1.        0.
           101          102      2.367    0.        0.        1.184    0.        0.
$
$      2      3      4      5      6      7      8      9
$
$
PBEAML11  21          CHAN
           4.0      6.0      1.0      1.0          YES      0.5      3.0
           4.5      0.75    0.75          YES      1.0      2.0      3.0
           0.5      0.5
$
MAT1      21          3.+7          .3
$
FORCE     1            2              192.    0.        1.        0.
$
ENDDATA

```

```

0
          S T R E S S E S   I N   B E A M   E L E M E N T S           ( C B E A M )
          STAT DIST/
ELEMENT-ID  GRID  LENGTH  SXC      SXD      SXE      SXF      S-MAX      S-MIN      M.S.-T  M.S.-C
0          1
          1      .000  -5.147177E+02  5.147177E+02  5.147177E+02 -5.147177E+02  5.147177E+02 -5.147177E+02
          0      .500  -6.103772E+02  6.103772E+02  6.103772E+02 -6.103772E+02  6.103772E+02 -6.103772E+02
          2      1.000  -2.304461E+00  2.304461E+00  2.304461E+00 -2.304461E+00  2.304461E+00 -2.304461E+00

```

Figure 4-32 Stress Output for the Tapered Beam using Cross-Sectional Dimension

As a side topic to help understand the implementation of warping, it is useful to see the actual equations being used. The basic equation for twist about the shear center of a beam is given by

$$m_0 = \frac{d^2}{dx^2} \left(EC_w \frac{d^2 \theta_x}{dx^2} \right) - \frac{d}{dx} \left(GJ \frac{d\theta_x}{dx} \right) \quad \text{Eq. 4-2}$$

where C_w is the warping coefficient.

The twist of the beam is defined as

$$\phi = \frac{d\theta_x}{dx} \quad \text{Eq. 4-3}$$

Substituting [Eq. 4-2](#) into [Eq. 4-3](#) and transferring the applied internal torsional moments to the end of the beam, the equation for the warping stiffness is reduced to [Eq. 4-4](#).

$$T_x = \frac{d}{dx} \left(EC_w \frac{d\phi}{dx} \right) - JG\phi \quad \text{Eq. 4-4}$$

The scalar points defined on the CBEAM entry are used to represent the ϕ . T_x is the warping torque.

Returning to our taper beam model, it is interesting to see how the single beam element compares to the same member modeled as plate elements and solid elements. [Table 4-1](#) shows the results of modeling the tapered member using a single CBEAM element, plate elements (CQUAD4), and solid elements (CHEXA). The CQUAD4 and CHEXA elements are described in “[Two-Dimensional Elements](#)” on page 127 and “[Three-Dimensional Elements](#)” on page 148, respectively.

Table 4-1 Comparison of the Beam, Plate, and Solid Element Model for the Tapered Beam

Element Type	Number of Elements	Number of DOFs	Y-Disp. at Free End x 10 ⁻² in	θ_z at the Free End x 10 ⁻³	Maximum Normal Stress psi
CBEAM	1	14	1.02	1.36	610
CQUAD4	960	6,174	0.99	0.67	710
CHEXA	3,840	15,435	1.08	0.97	622

The single beam element model with only 14 degrees-of-freedom compares well with the 15,435-degree-of-freedom solid model. The CQUAD4 plate model is included for completeness. Typically, you do not use plate elements for this type of structure. The flanges and web are very thick and do not behave like plates. The stress results shown in [Table 4-1](#) reflect this situation.

The solid model, on the other hand, represents a good use of the CHEXA element. This is discussed later in the solid element section. All of the input file for this example can be located on the delivery media for your review.

The CBEND Element

The CBEND element forms a circular arc that connects two grid points. This element has extensional and torsional stiffness, bending stiffness, and transverse shear flexibility in two perpendicular directions. Typical applications of the CBEND include modeling of pressurized pipe systems and curved components that behave as one-dimensional members.

Specific features of the CBEND element are as follows:

- Principal bending axes must be parallel and perpendicular to the plane of the element (see [Figure 4-33](#)).
- The geometric center of the element may be offset in two directions (see [Figure 4-33](#)).
- The offset of the neutral axis from the centroidal center due to curvature is calculated automatically with a user-override (DN) available for the curved beam form of the element.
- Four methods are available to define the plane of the element and its curvature.
- Three methods are available in the curved pipe form to account for the effect of curvature on bending stiffness and stress.
- The effect of internal pressure on stiffness and stress can be accounted for using two of the three methods mentioned in the previous item.
- Axial stresses can be output at four cross-sectional points at each end of the element. Forces and moments are output at both ends.
- Distributed loads may be placed along the length of the element by means of the PLOAD1 entry.

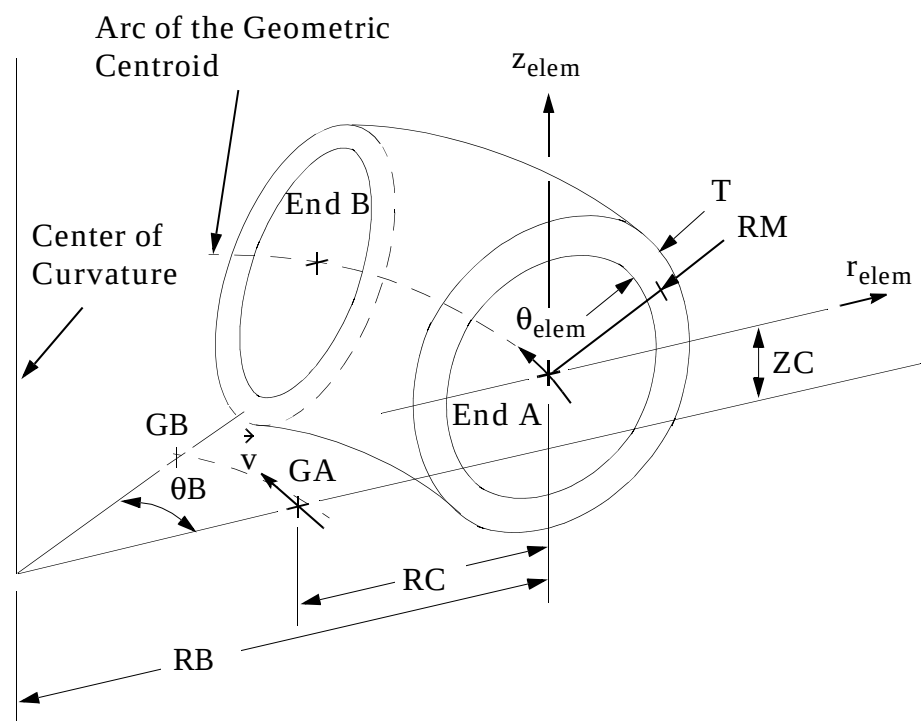
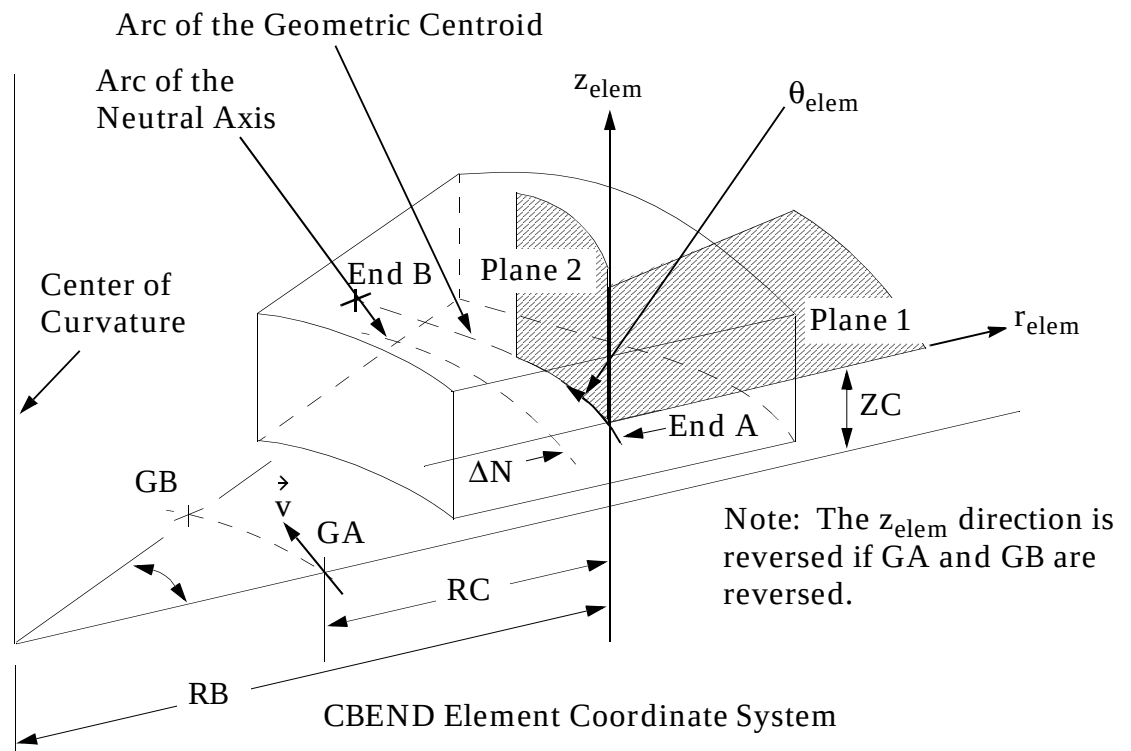
The geometry and properties are entered on the CBEND and PBEND entries, respectively. The transverse shear flexibility can be omitted by leaving the appropriate fields blank on the PBEND entry.

The format of the CBEND entry is as follows:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

Field	Contents
EID	Unique element identification number.
PID	Property identification number of a PBEND entry.
GA, GB	Grid point identification numbers of connection points.
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA.

- G0 Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. The vector $\vec{v}$ is then translated to End A.
- GEOM Flag to select specification of the bend element.



CBEND Circular Cross Section Element Coordinate System

Figure 4-33 The CBEND Element

The PBEND entry has two alternate forms. The first form corresponds to a curved beam of an arbitrary cross section. The second form is used to model elbow and curved pipes.

Like the CBEAM element, the CBEND element must be supplied with positive values for A, I1, and I2. The transverse shear flexibility can be omitted by leaving the appropriate fields blank on the PBEND entry. The format of the PBEND entry is as follows:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	A	I1	I2	J	RB	THETA B	
	C1	C2	DI	D2	E1	E2	F1	F2	
	K1	K2	NSM	RC	ZC	DELTA N			

Alternate Format for Elbows and Curved Pipes:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	FSI	RM	T	P	RB	THETA B	
			NSM	RC	ZC				

Field	Contents
PID	Property identification number.
MID	Material identification number.
A	Area of the beam cross section.
I1, I2	Area moments of inertia in planes 1 and 2.
J	Torsional stiffness.
FSI	Flag selecting the flexibility and stress intensification factors.
RM	Mean cross-sectional radius of the curved pipe.
T	Wall thickness of the curved pipe.
P	Internal pressure.
RB	Bend radius of the line of centroids.
THETAB	Arc angle of element.
Ci, Di, Ei, Fi	The r,z locations from the geometric centroid for stress data recovery.
K1, K2	Shear stiffness factor K in $K \cdot A \cdot G$ for plane 1 and plane 2.
NSM	Nonstructural mass per unit length.
RC	Radial offset of the geometric centroid from points GA and GB.

ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector .
DELTAN	Radial offset of the neutral axis from the geometric centroid; positive is toward the center of curvature.

4.4 Two-Dimensional Elements

This chapter discusses the following groups of two-dimensional elements: the CQUAD4 and CTRIA3 elements, the CQUAD8 and CTRIA6 elements, the CQUADR and CTRIAR elements, the CSHEAR element, and the CRAC2D element.

The CQUAD4, CTRIA3, CQUAD8, CTRIA6, CQUADR, and CTRIAR elements are commonly referred to as the plate and shell elements within MSC.Nastran. These elements differ principally in their shape, number of connected grid points, and number of internal stress recovery points. Each element type can be used to model membranes, plates, and thick or thin shells. Their properties, which are defined using the PSHELL entry, are identical. The important distinction among the elements is the accuracy that is achieved in different applications.

The CQUAD8 and CTRIA6 elements have the same features as the CQUAD4 and CTRIA3 elements, but are not used as frequently. The CQUAD8 and CTRIA6 are higher-order elements that allow for the use of midside nodes in addition to corner nodes. These midside nodes increase the accuracy of the element but are more difficult to mesh. Therefore, most users prefer using the CQUAD4 and CTRIA3 family of elements.

Before discussing the elements in detail, note that for accuracy reasons the quadrilateral elements (CQUAD4 and CQUAD8) are preferred over the triangular elements (CTRIA3 and CTRIA6). The latter are mainly used for mesh transitions or for modeling portions of a structure when quadrilateral elements are impractical.

The CQUAD4 and CTRIA3 Elements

The connectivity of the CQUAD4 and the CTRIA3 elements are entered on the CQUAD4 and CTRIA3 entries, respectively. The format of the CQUAD4 entry is as follows:

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

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHELL or PCOMP entry.
Gi	Grid point identification numbers of connection points.
THETA	Material property orientation angle in degrees.
MCID	Material coordinate system identification number.
ZOFFS	Offset from the surface of grid points to the element reference plane.
Ti	Membrane thickness of element at grid points G1 through G4.

The format of the CTRIA3 element is similar; refer to the **MSC.Nastran Quick Reference Guide**.

The material properties for the CQUAD4 and CTRIA3 are specified with the PSHELL or PCOMP entries. The PCOMP entry is specifically used for composites and is discussed in “**Material Properties and Composites**” in Chapter 5.

For all applications other than composites, the PSHELL entry should be used. The format of the PSHELL entry is as follows:

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

Field	Contents
PID	Property identification number.
MID1	Material identification number for the membrane.
T	Default membrane thickness for T_i .
MID2	Material identification number for bending.
$12I/T^3$	Bending moment of inertia ratio $12I/T^3$. Ratio of the actual bending moment inertia of the shell I to the bending moment of inertia of a homogeneous shell $T^3/12$. The default value is for a homogeneous shell.
MID3	Material identification number for transverse shear.
TS/T	Transverse shear thickness ratio TS/T . Ratio of the shear thickness, (TS), to the membrane thickness of the shell T . The default value is for a homogeneous shell.
NSM	Nonstructural mass per unit area.
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.
MID4	Material identification number for membrane-bending coupling.

As can be seen, the PSHELL entry is used to define the material ID for the membrane properties, the bending properties, the transverse shear properties, the bending-membrane coupling properties, and the bending and transverse shear parameters. By choosing the appropriate materials and parameters, virtually any plate configuration may be obtained.

The most common use of the PSHELL entry is to model an isotropic thin plate. The preferred method to define an isotropic plate is to enter the same MAT1 ID for the membrane properties (MID1) and bending properties (MID2) only and leave the other

fields blank. For a thick plate, you may also wish to enter an MAT1 ID for the transverse shear (MID3). The use of the PSHELL entry to model anisotropic plates is discussed in [“Material Properties and Composites”](#) on page 177.

There are two ways you can input the thickness of the plate elements. The simplest and the recommended way is to enter a constant element thickness in field 4 of the PSHELL entry. If the element has nonuniform thickness, the thickness at each of the corner points is entered on the continuation line of the CQUAD4/CTRIA3 connectivity entry. If you enter the thickness on both the PSHELL entry and the connectivity entry, the individual corner thicknesses take precedence.

Also located on the PSHELL entry are the stress recovery locations Z1 and Z2. By default, Z1 and Z2 are equal to one-half of the plate thickness (typical for a homogeneous plate). If you are modeling a composite plate, you may wish to enter values other than the defaults to identify the outermost fiber locations of the plate for stress analysis.

The element coordinate systems for the CQUAD4 is shown in [Figure 4-34](#). The orientation of the element coordinate system is determined by the order of the connectivity for the grid points. The element z-axis, often referred to as the positive normal, is determined using the right-hand rule (the z-axis is “out of the paper” as shown in [Figure 4-34](#). Therefore, if you change the order of the grid points connectivity, the direction of this positive normal also reverses. This rule is important to remember when applying pressure loads or viewing the element forces or stresses. Often element stress contours appear to be strange when they are displayed by a postprocessor because the normals of the adjacent elements may be inconsistent. Remember that components of forces, moments, and element stresses are always output in the element coordinate system.

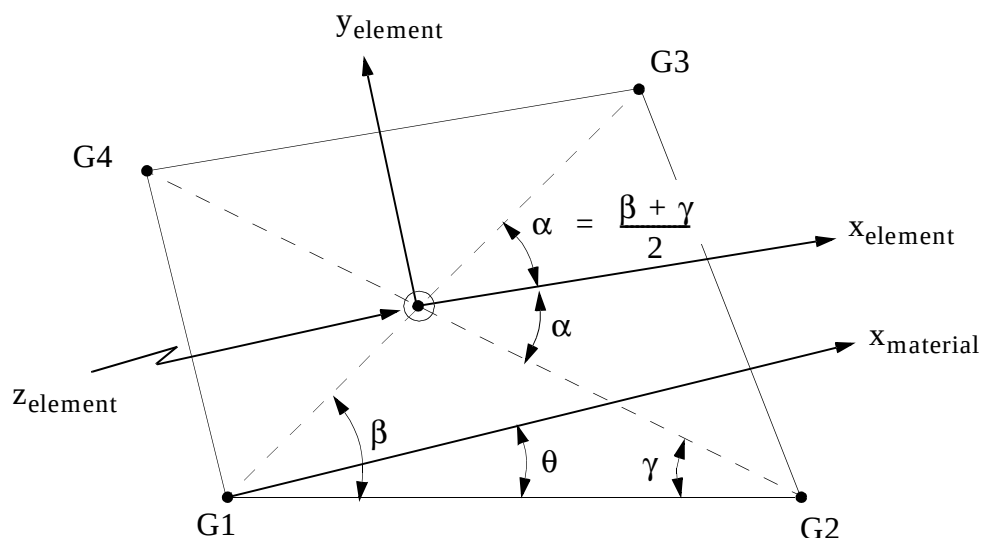


Figure 4-34 CQUAD4 Element Geometry and Coordinate Systems

It can be seen from [Figure 4-34](#) that each element has an element coordinate system and a material coordinate system that may be the same or different. Using a material coordinate system different for the element coordinate system is useful when the material properties are orthotropic or anisotropic. You may wish to review these diagrams when you read [“Material Properties and Composites”](#) on page 177.

Figure 4-35 shows the positive directions of forces, moments, and stresses. These diagrams are particularly helpful in understanding the element force output generated when using the FORCE (or ELFORCE) Case Control command in the Case Control Section. The forces shown are defined to be

- F_x, F_y Normal forces acting on the x and y faces.
- F_{xy} In-plane shear force.
- V_x, V_y Transverse shear forces acting on the x and y faces.
- M_x, M_y Bending moments on the x and y faces.
- M_{xy} Twisting moment.

It is important to note that the forces generated for the elements are forces per unit length. A common error for new users is to assume that the force shown in the output is the total force acting on the element--it is not! It is the force per unit length.

Also shown in **Figure 4-35** are the stresses generated for a plate element. The following summarizes the stresses generated using the STRESS (or ELSTRESS) in the Case Control Section:

- σ_x, σ_y Normal stresses in the x- and y-directions.
- τ_x, σ_y Shear stress on the x face in the y-direction.
- Major and minor principal stress.
- Angle between the the x-axis and the major principal direction. This angle is derived from σ_x, σ_y , and τ_{xy} .
- von Mises equivalent stress (or maximum shear stress if STRESS(MAXS) is used). These stresses are derived from σ_x, σ_y , and τ_{xy} .

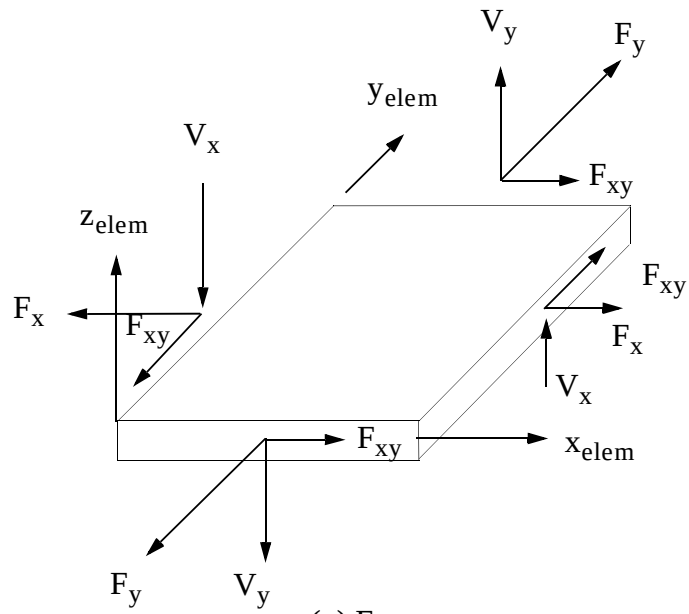
The derived quantities are defined in “**Output Listing**” on page 639. Similar output may be obtained for the element strains using the STRAIN command in the Case Control Section.

By default, the element forces, stresses, and strains are generated for the centroid of the CQUAD4 and CTRIA3 elements only. You have the option to compute and output these quantities at the corner grid points using the corner option for the CQUAD4 element. For example, if you wish to obtain the corner stresses in addition to the centroidal stress for the CQUAD4 elements, you request

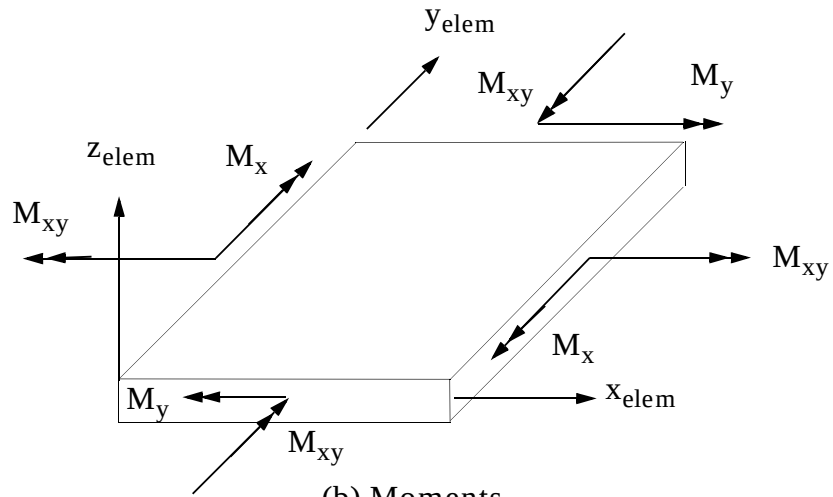
STRESS(CORNER) = n where n is a set of elements
or
STRESS(CORNER) = ALL for all the elements

Hierarchy for Output Type. Currently only one output type is supported per run (i.e., CENTER and CORNER output types cannot be mixed even for different output requests). To determine this output type, you should use the following hierarchy:

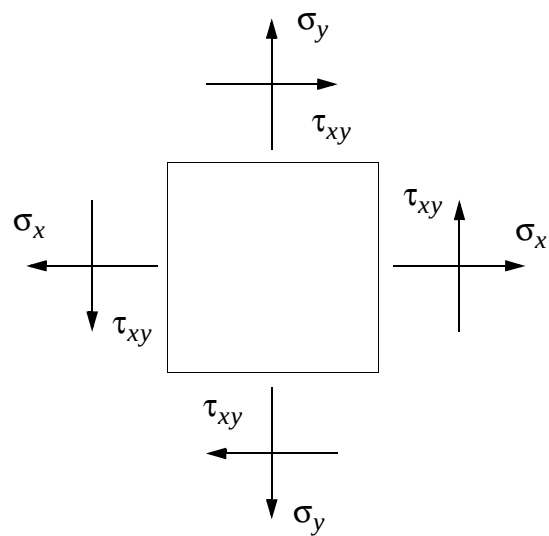
1. Only requests made in the first subcase and above the subcase level are considered when setting the output type. Subcases below the first are not considered for determining the output type (and the output type is then set to CENTER, i.e., the default).
2. The output type of the STRESS request in the first subcase determines the output type for STRESS, STRAIN, and FORCE for the entire run.
3. If there is no STRESS request in the first subcase, then the output type of the STRESS request above the subcase level determines the output type for STRESS, STRAIN, and FORCE for the entire run.
4. If there is no STRESS request above or in the first subcase, then the output type of the STRAIN request in the first subcase determines the output type for STRAIN and FORCE for the entire run.
5. If there is no STRAIN request in the first subcase, then the output type of the STRAIN request above the subcase level determines the output type for STRAIN and FORCE for the entire run.
6. If there is no STRAIN request above or in the first subcase, then the output type of the FORCE request in the first subcase determines the output type for FORCE for the entire run.
7. If there is no FORCE request in the first subcase, then the output type of the FORCE request above the subcase level determines the output type for FORCE for the entire run.



(a) Forces



(b) Moments



(c) Stresses

Figure 4-35 Forces, Moments, and Stresses in Plate Elements

Returning to the CQUAD4 and CTRIA3 element definition, you must specify four grid points for each CQUAD4 and three grid points for each CTRIA3 element. These grid points define the corners of the element. The element, however, is elastically connected to only five of the six degrees of freedom at each of its grid points. The element does not provide direct elastic stiffness to the sixth degrees of freedom, i.e., the rotation about the normal to the surface of the element. Therefore, if a grid point is attached to the CQUAD4 elements only and all of the elements are in the same plane, then the rotational degrees of freedom about the surface normal have zero stiffness. Unless precautions are taken, this zero stiffness results in a singular stiffness matrix and the job fails.

To avoid this potential problem, you have two choices. The first choice is to identify the rotational degrees of freedom and constrain these components using an SPC entry (either field 8 of the GRID entry or an SPC entry). If you choose this approach, make sure that you do not constrain any components that have stiffness attached. To automatically identify and constrain those degrees of freedom in the model that have no stiffness, you can use the parameter AUTOSPC. When this parameter is used, a table is printed in the output file indicating which degrees of freedom in the model are constrained. It is a good practice to review this table to ensure that the components being constrained actually should be constrained and that no components are left unconstrained due to a modeling error. PARAM,AUTOSPC is automatically turned on by default in SOL 101.

The second method of eliminating the singularity associated with the normal rotation degree of freedom is to apply an artificial stiffness term to the degrees of freedom using PARAM K6ROT. It is important to remember when using this parameter that the stiffness being included for the rotational degree of freedom is not a true stiffness and should not be used as such. For example, if you need to connect a CBAR element to the CQUAD4 element, you should not rely on the K6ROT stiffness to transfer the bending moment at the end of the CBAR into the plate. Various methods of transitioning different element types are discussed in “[Modeling Guidelines](#)” on page 311.

An example of using the CQUAD4 element to model a flat cantilever plate is shown in [Figure 4-36](#).

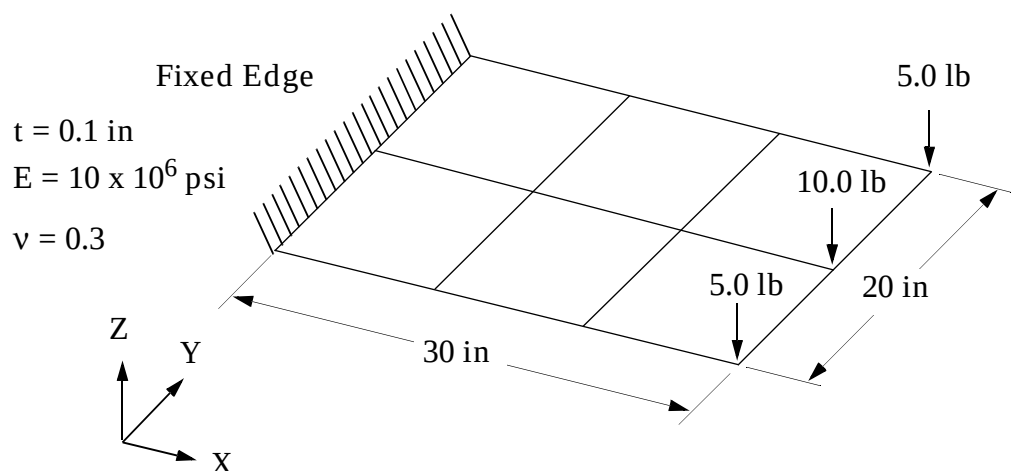


Figure 4-36 Cantilever Plate Model

The input file is shown in [Listing 4-9](#).

Listing 4-9 Cantilever Plate Input File

```

$
$ FILENAME - PLATE2.DAT
$
BEGIN BULK
PARAM      AUTOSPC YES
PARAM      POST    0
$
GRID      1          0.0      -10.      0.0
GRID      2          10.      -10.      0.0
GRID      3          20.      -10.      0.0
GRID      4          30.      -10.      0.0
GRID      5          0.0       0.0       0.0
GRID      6          10.       0.0       0.0
GRID      7          20.       0.0       0.0
GRID      8          30.       0.0       0.0
GRID      9          0.0      10.       0.0
GRID     10          10.      10.       0.0
GRID     11          20.      10.       0.0
GRID     12          30.      10.       0.0
$
CQUAD4    1          1          1          2          6          5
CQUAD4    2          1          2          3          7          6
CQUAD4    3          1          3          4          8          7
CQUAD4    4          1          5          6         10          9
CQUAD4    5          1          6          7         11         10
CQUAD4    6          1          7          8         12         11
$
FORCE     1          12          0         -5.0       0.0       0.0       1.
FORCE     1           4          0         -5.0       0.0       0.0       1.
FORCE     1           8          0        -10.       0.0       0.0       1.
$
SPC1      1          123456    1          5          9
$
PSHELL    1          1          .1          1
$
MAT1      1          1.+7          .3
ENDDATA

```

The parameter AUTOSPC is used to constrain the rotational degrees of freedom normal to the plate. The grid point singularity table output, as shown in [Figure 4-37](#), is generated by PARAM,AUTOSPC. As you see can, the rotational degrees of freedom (DOF 6 in this case) is removed from the *f*-set (Free set) to the *s*-set (SPC set). “[Matrix Operations](#)” on page 455 describes the sets used in MSC.Nastran. The asterisk at the end of the lines indicates that the action was taken.

GRID POINT SINGULARITY TABLE

POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	NEW USET UNION	NEW USET EXCLUSIVE	NEW USET UNION	
2	G	6	0.00E+00	B	F	SB	S	*
3	G	6	0.00E+00	B	F	SB	S	*
4	G	6	0.00E+00	B	F	SB	S	*
6	G	6	0.00E+00	B	F	SB	S	*
7	G	6	0.00E+00	B	F	SB	S	*
8	G	6	0.00E+00	B	F	SB	S	*
10	G	6	0.00E+00	B	F	SB	S	*
11	G	6	0.00E+00	B	F	SB	S	*
12	G	6	0.00E+00	B	F	SB	S	*

Figure 4-37 Parameter AUTOSPC Output.

The displacement and stress output for the plate model are shown in [Figure 4-38](#).

DISPLACEMENT VECTOR

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	-1.395891E+00	-3.679435E-02	2.621396E-01	.0
3	G	.0	.0	-5.132846E+00	-4.282482E-02	4.666319E-01	.0
4	G	.0	.0	-1.013968E+01	-2.199761E-02	5.164736E-01	.0
5	G	.0	.0	.0	.0	.0	.0
6	G	.0	.0	-1.571300E+00	9.460242E-16	2.948987E-01	.0
7	G	.0	.0	-5.344157E+00	-1.331533E-15	4.418921E-01	.0
8	G	.0	.0	-1.024780E+01	-9.773031E-16	5.206971E-01	.0
9	G	.0	.0	.0	.0	.0	.0
10	G	.0	.0	-1.395891E+00	3.679435E-02	2.621396E-01	.0
11	G	.0	.0	-5.132846E+00	4.282482E-02	4.666319E-01	.0
12	G	.0	.0	-1.013968E+01	2.199761E-02	5.164736E-01	.0

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)				VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
1	-5.000000E-02	-1.500000E+04	-3.580141E+03	-6.687832E+02	-86.6598	-3.541109E+03	-1.503903E+04	1.361826E+04	
	5.000000E-02	1.500000E+04	3.580141E+03	6.687832E+02	3.3402	1.503903E+04	3.541109E+03	1.361826E+04	
2	-5.000000E-02	-9.000000E+03	-7.095209E+02	-1.350937E+02	-89.0667	-7.073201E+02	-9.002200E+03	8.670206E+03	
	5.000000E-02	9.000000E+03	7.095209E+02	1.350937E+02	.9333	9.002200E+03	7.073201E+02	8.670206E+03	
3	-5.000000E-02	-3.000000E+03	7.205607E+02	3.975338E+02	83.9688	7.625621E+02	-3.042001E+03	3.486400E+03	
	5.000000E-02	3.000000E+03	-7.205607E+02	-3.975338E+02	-6.0312	3.042001E+03	-7.625621E+02	3.486400E+03	
4	-5.000000E-02	-1.500000E+04	-3.580141E+03	6.687832E+02	86.6598	-3.541109E+03	-1.503903E+04	1.361826E+04	
	5.000000E-02	1.500000E+04	3.580141E+03	-6.687832E+02	-3.3402	1.503903E+04	3.541109E+03	1.361826E+04	
5	-5.000000E-02	-9.000000E+03	-7.095209E+02	1.350937E+02	89.0667	-7.073201E+02	-9.002200E+03	8.670206E+03	
	5.000000E-02	9.000000E+03	7.095209E+02	-1.350937E+02	-.9333	9.002200E+03	7.073201E+02	8.670206E+03	
6	-5.000000E-02	-3.000000E+03	7.205607E+02	-3.975338E+02	-83.9688	7.625621E+02	-3.042001E+03	3.486400E+03	
	5.000000E-02	3.000000E+03	-7.205607E+02	3.975338E+02	6.0312	3.042001E+03	-7.625621E+02	3.486400E+03	

Figure 4-38 Displacement and Stress Output

Plate Element Strain Calculation. In addition to the stress output, you can also request strain output for the plate element. Deformation in the X-Y plane of the plate element at any point C at a distance z in the normal direction to plate middle surface is

$$U = U_o + z\theta_y \quad \text{Eq. 4-5}$$

$$V = V_o - z\theta_x \quad \text{Eq. 4-6}$$

where U, V are the displacements in the element coordinate system, and θ_x, θ_y are the curvatures. The U_o and V_o are the plate mid-surface displacements.

The strain-displacement-middle surface strain and curvatures relationship is given by

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial U_o}{\partial X} \\ \frac{\partial V_o}{\partial Y} \\ \frac{\partial U_o}{\partial Y} + \frac{\partial V_o}{\partial X} \end{Bmatrix} + z \begin{Bmatrix} \frac{\partial \theta_y}{\partial x} \\ -\frac{\partial \theta_x}{\partial y} \\ \frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x} \end{Bmatrix} = \begin{Bmatrix} \epsilon_x^o \\ \epsilon_y^o \\ \gamma_{xy}^o \end{Bmatrix} - z \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix} \quad \text{Eq. 4-7}$$

where the ϵ^o 's and χ 's are the middle surface strains and curvatures, respectively.

You can request strain output as strains at the reference plane and curvatures or strains at locations Z1 and Z2. The following strain output Case Control command

STRAIN = n
or
STRAIN(STRCUR) = n

requests strains and curvatures at the reference plane. Similarly, the following strain output Case Control command

STRAIN(FIBER) = n

requests strains at Z1 and Z2. The example problem in [Listing 4-10](#) contains two identical subcases except for the strain output request format. The output is shown in [Figure 4-39](#). The first and second lines of the strain output for the first subcase represents the mean strains and curvatures, respectively, at the reference plane. The first and second lines of the strain output for the second subcase represents the strains at the bottom (Z1) and top (Z2) fibers, respectively.

In a linear static analysis, the strain output are total strain--mechanical plus thermal strain.

Listing 4-10 Input File Requesting Strain Output

```

$
$ strain2.dat
$
SOL      101
TIME     5
CEND
SPC = 100
DISPLACEMENT = ALL
LOAD = 10
$
SUBCASE 1
  LABEL = MEAN STRAIN AND CURVATURE
  STRAIN      = ALL
$
SUBCASE 2
  LABEL = STRAIN AT FIBER LOCATIONS
  STRAIN(FIBER) = ALL
$
BEGIN BULK
$
CQUAD4  1      1      1      2      3      4
FORCE   10      2      10.    1.    0.    -1.
FORCE   10      3      10.    1.    0.    -1.
GRID    1      0.    0.    0.
GRID    2      10.   0.    0.
GRID    3      10.   10.   0.
GRID    4      0.    10.   0.
MAT1    1      190000. .3
PSHELL  1      1      1.0    1
SPC1    100    123456  1      4
$
ENDDATA

```

```

0      MEAN STRAIN AND CURVATURE                                     SUBCASE 1

      STRAINS IN QUADRILATERAL ELEMENTS ( QUAD 4 )
ELEMENT STRAIN STRAINS IN ELEMENT COORD SYSTEM PRINCIPAL STRAINS (ZERO SHEAR)
  ID. CURVATURE NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR VON MISES
0      1      .0      1.027233E-05 -2.311274E-06 -8.470329E-22 .0000 1.027233E-05 -2.311274E-06 7.734624E-06
      -1.000000E+00 -6.163397E-04 1.386764E-04 1.084202E-19 90.0000 1.386764E-04 -6.163397E-04 4.640774E-04
1
      AUGUST 6, 1996 MSC/NASTRAN 8/ 2/96 PAGE 11

0      STRAIN AT FIBER LOCATIONS                                     SUBCASE 2

      STRAINS IN QUADRILATERAL ELEMENTS ( QUAD 4 )
ELEMENT FIBRE STRAINS IN ELEMENT COORD SYSTEM PRINCIPAL STRAINS (ZERO SHEAR)
  ID. DISTANCE NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR VON MISES
0      1      -5.000000E-01 -2.978976E-04 6.702695E-05 5.336308E-20 90.0000 6.702695E-05 -2.978976E-04 2.243041E-04
      5.000000E-01 3.184422E-04 -7.164950E-05 -5.505714E-20 .0000 3.184422E-04 -7.164950E-05 2.397733E-04

```

Figure 4-39 Strain Output for Plate Elements

Small Deflection Assumption. It is often of concern to the new user that the displacement for each grid point deflection is exclusively in the Z-direction for this problem. Physically, you know that there is a displacement in the X-direction when the Z-displacement is as large as it is for this problem. However, linear analysis assumes small deflections, and as such, there is no deflection in the X-direction.

If the deflection of the plate is significant enough that the in-plane forces cannot be ignored, a nonlinear analysis may be warranted. Such is the case for the cantilever plate model--the deflections at the tip are over 5 inches. To show a comparison, this same problem was run using Solution 106, a nonlinear statics solution. A plot showing the linear and nonlinear results is given in [Figure 4-40](#).

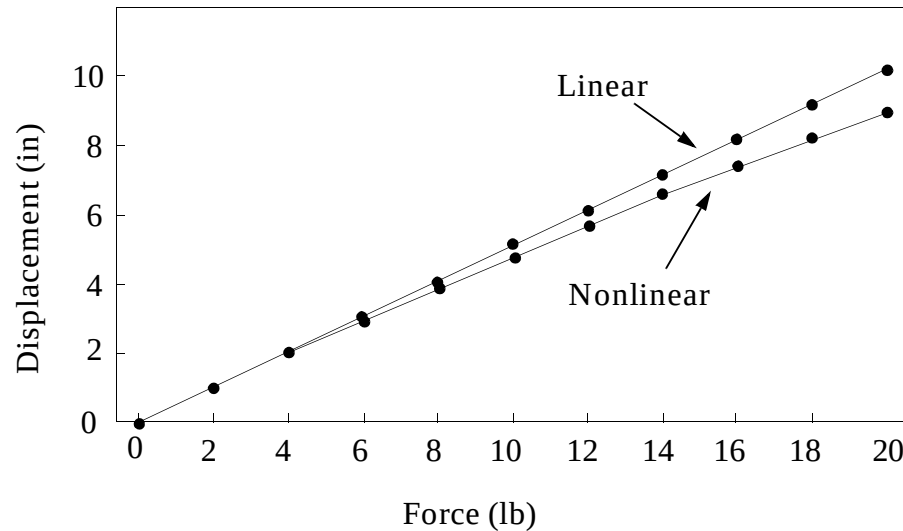


Figure 4-40 Nonlinear vs. Linear Results

This example is not intended to show you how to perform a nonlinear analysis; it serves only to remind you that linear analysis means small deflections and the superposition principle applies. For more information on nonlinear analysis, refer to “” on page 568 of the *MSC.Nastran Reference Manual*. The input file for the nonlinear run, “nonlin.dat”, is located on the delivery media if you wish to see how the nonlinear run is performed.

If the corner output is not used, the stresses are output in the element coordinate system for the center of the element only. For this example, the maximum normal X stress is 7500 psi. However, if you compute the stress at the fixed end using simple beam theory, the stress is 9000 psi. This discrepancy occurs because the 7500 psi stress is computed at a distance of 5 inches from the fixed edge. Although the example seems trivial, most users use a postprocessor to view the results, as shown in [Figure 4-41](#). The contour plot in the [Figure 4-41](#) indicates the maximum stress as 7500 psi in the region close to the fixed edge. If you only look at the contour plot, as many people do, you can easily be misled. To help reduce interpretation errors, the corner stress output discussed earlier may be used. Grid point stresses and stress discontinuity checks are also available. These features are discussed in “[Modeling Guidelines](#)” on page 311 and “[Model Verification](#)” on page 369.

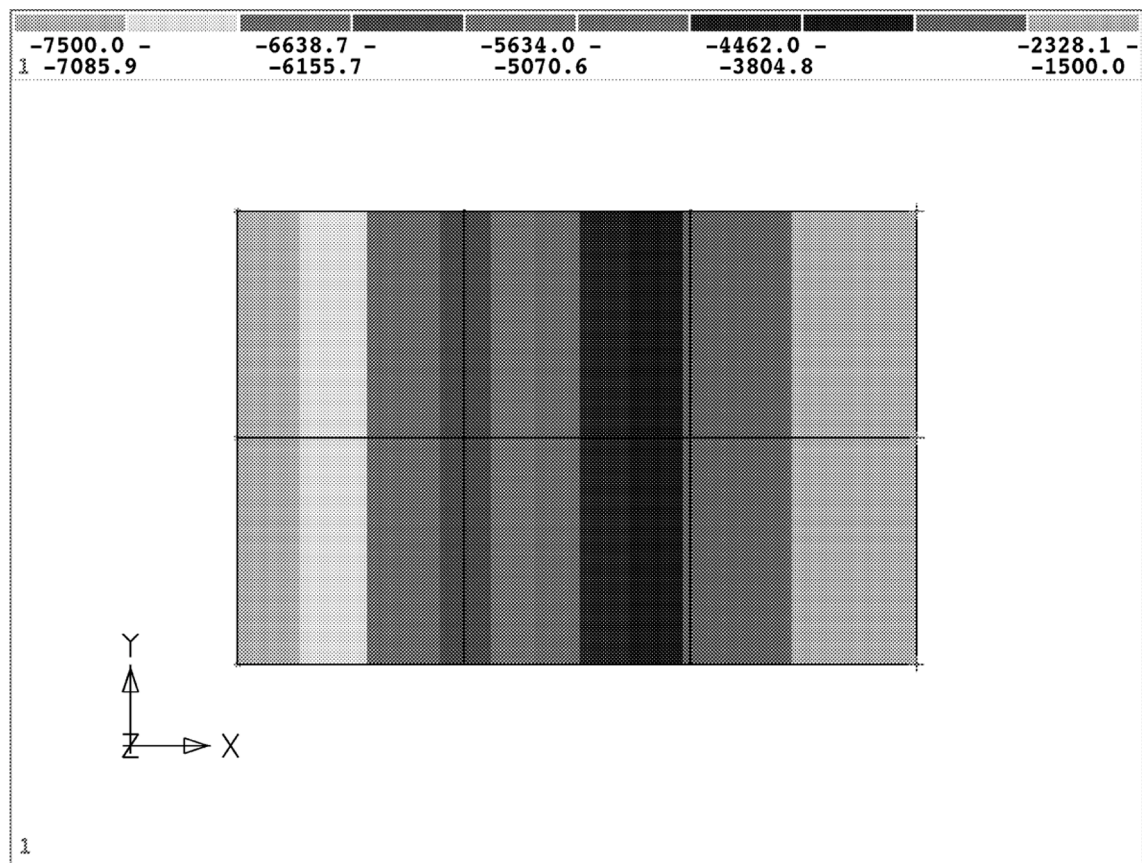


Figure 4-41 Cantilever Plate Stress Contour Plot

The CQUAD8 and CTRIA6 Elements

The CQUAD8 and CTRIA6 elements are similar to the CQUAD4 and CTRIA3 elements except they may also have midside grid points and curved edges. The format of the CQUAD8 entry is as follows:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID	ZOFFS	

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHELL or PCOMP entry.
G1, G2, G3, G4	Identification numbers of connected corner grid points.
G5, G6, G7, G8	Identification numbers of connected edge grid points (optional).
T1	Membrane thickness of element at corner grid points.
THETA	Material property orientation angle in degrees.

MCID	Material coordinate system identification number.
ZOFFS	Offset from the surface of grid points to the element reference plane.

The first four grid points on the CQUAD8 entry define the corners of the element and as such are required. The last four grid points define the midside grid points, any of which can be deleted (principally to accommodate changes in mesh spacing, but deleting midside nodes in general is not recommended). An edge grid point need not lie on the straight line segment joining adjacent corner points. However, it is recommended that it be separated from the midpoint of the line by no more than twenty percent of the length of the line.

The properties of the CQUAD8 and CTRIA6 elements are entered on the PSHELL element as described in the previous section. All of the capabilities described for the CQUAD4 element apply to the CQUAD8 and CTRIA6 elements. The principal advantage of these elements is that they may be more accurate in curved shell applications for the same number of degrees of freedom. The disadvantage is that with the addition of the midside node, they are more difficult to mesh for irregular shape structures.

The element coordinate system is defined implicitly by the location and connection order of the grid points as is the case for the CQUAD4 and CTRIA3. The element coordinate systems for these elements are described in the [*MSC.Nastran Quick Reference Guide*](#).

If the element force, stress, and strain outputs are requested in the Case Control Section, they are always output at both the centroid of the element and at the corner locations. Recalling the element data recovery method for the CQUAD4, if you wish corner values to be output, they must be requested using the corner option. The corner option does not apply to the CQUAD8, CTRIA3, or CTRIA6 element.

Shell Element Force, Stress, and Strain Output in the Material Coordinate System

For the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 elements, a parameter is available to request the element response quantities in the material coordinate system. This capability is limited to element force, stress, and strain responses and provides output in the coordinate system defined using the THETA / MCID field on their associated bulk data entries. Both element center and element corner results are output in the material system. This capability must be used with care because the output of element results in the material coordinate system can produce incorrect results in subsequent calculations that assume the element results are in the element coordinate system. By default, the results for the shell elements are produced in the element coordinate system. The element coordinate system is an artifact of the meshing option chosen to discretize the model and, in the typical case, does not have a physical interpretation. On the other hand, the output of results in a specified coordinate system does have physical meaning and allows the user to scan results across elements and be assured that they have been output in a common coordinate system.

Input

To view results in the material coordinate system, set the parameter, OMID, to yes. The default value for OMID is NO.

The parameter is applicable to forces, strains, and stresses for CTRIA3, CQUAD4, CTRIA6, and CQUAD8 elements in linear analyses. The default value produces these results in the element coordinate system. The parameter is applicable for all subcases and for static and dynamic analyses. PARAM,OMID,YES should not be used if grid point stresses are required or if the results are to be displayed by a postprocessor. The calculation of grid point stresses is based on element stress results with the assumption that the element responses are in the element coordinate system. Postprocessors, such as MSC.Patran, assume element results are in the element coordinate system, thereby producing incorrect values if they are not in that system.

Output

Figure 4-42 contains the strain output from two different runs of the same model, one with PARAM,OMID,YES and the other with PARAM,OMID,NO. Differences are seen in the labeling that indicates how the NORMAL-X, NORMAL-Y, AND SHEAR-XY results are output and in the results themselves. In this case, the material axis is at a 45 degree angle with respect to the element axis and this value is reflected in the difference in the angle listed under the PRINCIPAL STRAINS heading. Similar differences can be seen in the stress output, whereas the force output has an additional label indicating that results are in the material coordinate system when OMID is set to YES. The standard force output does not include this label. See **Figure 4-43**.

ELEMENT		S T R A I N S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)						OPTION
ID	GRID-ID	FIBRE DISTANCE	STRAINS IN MATERIAL COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)		
MISES			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR
1	CEN/4	-5.000000E-02	4.380871E-04	3.387215E-04	8.973553E-04	41.8406	8.398243E-04	-6.301572E-
05	5.820263E-04	5.000000E-02	-4.380871E-04	-3.387215E-04	-8.973553E-04	-48.1594	6.301572E-05	-8.398243E-
04	5.820263E-04	-5.000000E-02	4.592682E-04	3.387215E-04	7.979897E-04	40.7048	8.025165E-04	-4.526860E-
06	5.365264E-04	5.000000E-02	-4.592682E-04	-3.387215E-04	-7.979897E-04	-49.2952	4.526860E-06	-8.025165E-
04	5.365264E-04	5.000000E-02	4.592682E-04	3.387215E-04	7.979897E-04	49.2952	4.526860E-06	8.025165E-

a. Strain Output Fragment with OMID = YES (Key features in bold)

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

Figure 4-42 Effect of Parameter OMID on Element Strain Output

SUBCASE 1										
FORCES IN QUADRILATERAL ELEMENTS (QUAD4)										
OPTION = BILIN										
RESULTS IN MATERIAL COORD SYSTEM										
ELEMENT		- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -		
ID	GRID-ID	FX	FY	FX	MY	MX	MY	QX	QY	
1	CEN/4	.0	.0	.0	9.884681E+01	8.610762E+01	5.752278E+01	-1.243172E+01	2.000000E+02	
1		.0	.0	.0	1.006119E+02	8.787272E+01	4.924231E+01	-1.243172E+01	2.000000E+02	
7		.0	.0	.0	1.071273E+02	9.438809E+01	5.575769E+01	-1.243172E+01	2.000000E+02	
8		.0	.0	.0	9.708171E+01	8.434253E+01	6.580325E+01	-1.243172E+01	2.000000E+02	
5		.0	.0	.0	9.056634E+01	7.782716E+01	5.928787E+01	-1.243172E+01	2.000000E+02	
2	CEN/4	.0	.0	.0	8.610762E+01	9.884681E+01	5.752278E+01	2.000000E+02	-1.243172E+01	
2		.0	.0	.0	8.787272E+01	1.006119E+02	4.924231E+01	2.000000E+02	-1.243172E+01	

Figure 4-43 Element Force Output Labeling with PARAM OMID=YES

The user must keep track of the value of the OMID parameter when working with the .pch file because there is no labeling for the punch output. It is suggested that the user include the value of this parameter in the job TITLE so that it will appear at the head of the .pch file.

Guidelines and Limitations

The current implementation of the OMID parameter is regarded as being incomplete in that it is limited to a subset of elements and has not been integrated with pre-processors and with grid point stress output that assumes that the results are in the element coordinate system. It is intended to provide a quick look at results that are in a consistent coordinate system so that one can easily and meaningfully scan these results across a number of elements. If these data are to be incorporated into design formulae, it removes the requirement that they first be transformed into a consistent coordinate system. Of course, the invariant principal stress/strain results have always provided mesh independent results.

The forces, stress, and strains in the material coordinate system may only be printed or punched and are NOT written to the .op2 or .xdb files. The usual element coordinate system results are written to the .op2 or .xdb file if they are requested (param,post,x).

The CQUADR and CTRIAR Elements

The CQUADR and CTRIAR plate elements take advantage of the normal rotational degrees of freedom (which have no stiffness associated with them in the standard plate elements) to provide improved membrane accuracy. The result is an element with stiffness in all six degrees of freedom at each grid point. However, this additional stiffness is not a true physical stiffness. The shell normal should be turned on when using the CQUADR and CTRIAR elements. The CQUADR and CTRIAR elements do not support differential stiffness. These elements should not be used for buckling analysis (see “[Linear Buckling](#)” on page 485). Also, we recommend that you do not mix elements of different formulations in your model. For example, you should not model part of a structure with the CQUAD4 and CTRIA3 elements and another part with the CQUADR and CTRIAR elements.

The Shear Panel Element (CSHEAR)

The CSHEAR element is a quadrilateral element with four grid points. The element models a thin buckled plate. It supports shear stress in its interior and also extensional force between adjacent grid points. Typically you use the CSHEAR element in situations where the bending stiffness and axial membrane stiffness of the plate is negligible. The use of CQUAD4 element in such situations results in an overly stiff model.

The most important application of the CSHEAR element is in the analysis of thin reinforced plates and shells, such as thin aircraft skin panels. In such applications, reinforcing rods (or beams) carry the extensional load, and the CSHEAR element carries the in-plane shear. This is particularly true if the real panel is buckled or if it is curved. The format of the CSHEAR entry is as follows:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

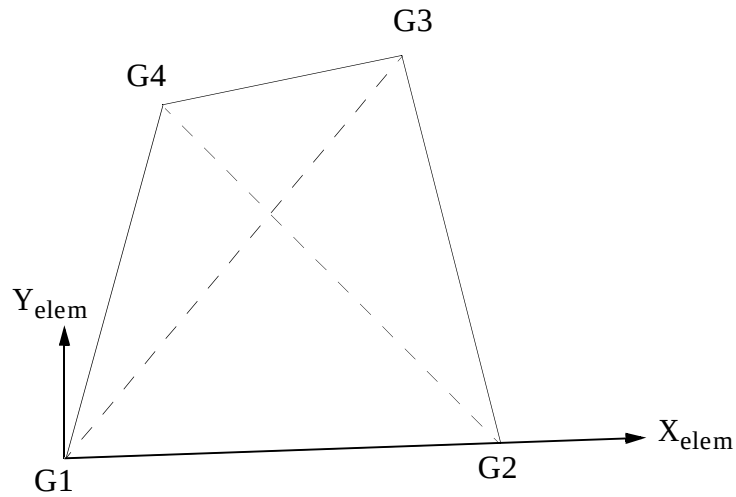
Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHEAR entry.
Gi	Grid point identification numbers of connection points.

The properties of the CSHEAR element are entered on the PSHEAR entry. The format of the PSHEAR entry is as follows:

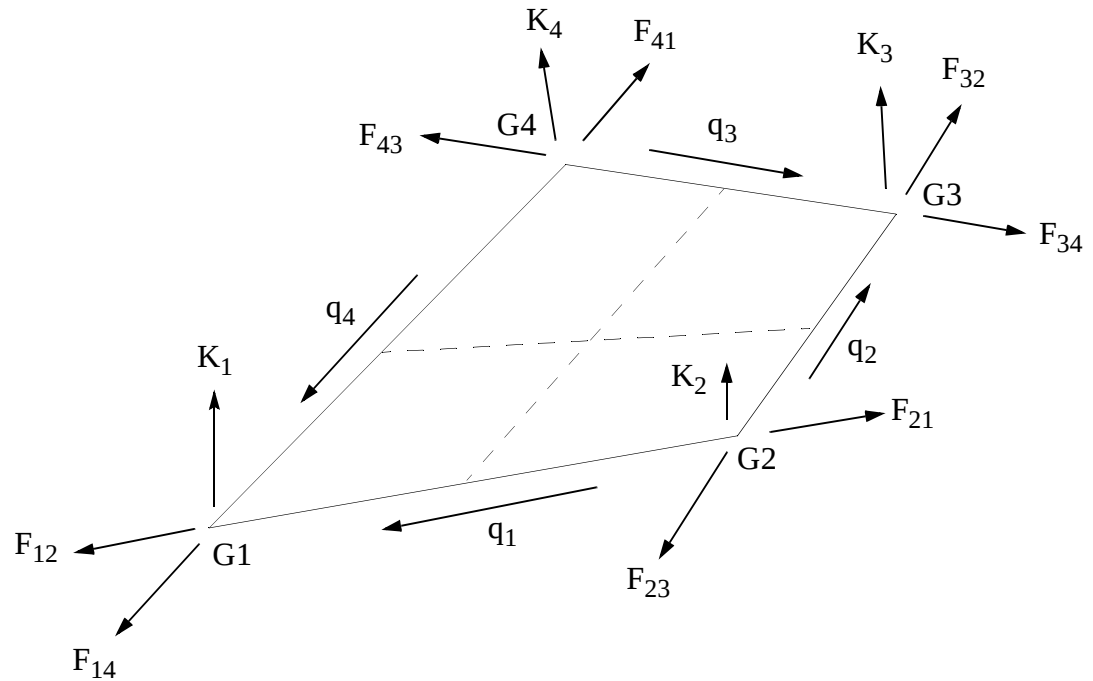
1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

Field	Contents
PID	Property identification number.
MID	Material identification number of a MAT1 entry.
T	Thickness of shear panel.
NSM	Nonstructural mass per unit area.
F1	Effectiveness factor for extensional stiffness along edges 1-2 and 3-4.
F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4.

The output for the CSHEAR element is the components of force at the corners of the element, the shear flows along each of the sides, the average shear stress, and the maximum shear stress. Positive directions for these quantities are identified in [Figure 4-44](#).



(a) Coordinate System



(b) Corner Forces and Shear Flows

Figure 4-44 Coordinate System and Element Forces for the Shear Panel

The optional parameters F1 and F2 on the PSHEAR entry are useful in representing an effective stiffness of the panel for extensional loads by means of equivalent rods on the perimeter of the element. If F1 is less than or equal to 1.01, the areas of the rods on edges 1-2 and 3-4 are set equal to $0.5 \cdot F1 \cdot T \cdot w_1$ where w_1 is the average width of the panel. If F1 is equal to 1.0, the panel is fully effective in the 1-2 direction. If F1 is greater than 1.01, the areas for the rods on edge 1-2 and edge 3-4 are each set equal to $0.5 \cdot F1 \cdot T^2$. The significance of F2 for edges 2-3 and 1-4 is similar.

Two-Dimensional Crack Tip Element (CRAC2D)

The two-dimensional crack tip element is used to model surfaces with a discontinuity due to a crack. The connectivity of the element is entered on the CRAC2D entry. The formats of the CRAC2D and ADUM8 entries are as follows:

1	2	3	4	5	6	7	8	9	10
CRAC2D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

ADUM8	18	0	5	0	CRAC2D				

Field	Contents

EID	Element identification number.
PID	Property identification number of a PRAC2D entry.
Gi	Grid point identification numbers of connection points.

Note the use of the ADUM8 entry. This is due to the fact that the CRAC2D element is not a fully implemented element and as such requires additional input. The element is what is known as a “dummy” element. (This has nothing to do with how smart the element is.) A dummy element is one that has been implemented using one of the prototype element routines. These prototype element routines (which are known as dummy routines) are in every release of MSC.Nastran for the advanced user to add their own elements to the element library. The crack tip elements are added as dummy element for expediency only. The CRAC2D element is dummy element 8; and therefore, the Bulk Data entry ADUM8 must be used with this element.

The properties and the stress evaluation techniques are entered on the PRAC2D entry, and its format is as follows:

1	2	3	4	5	6	7	8	9	10
PRAC2D	PID	MID	T	IPLANE	NSM	GAMMA	PHI		

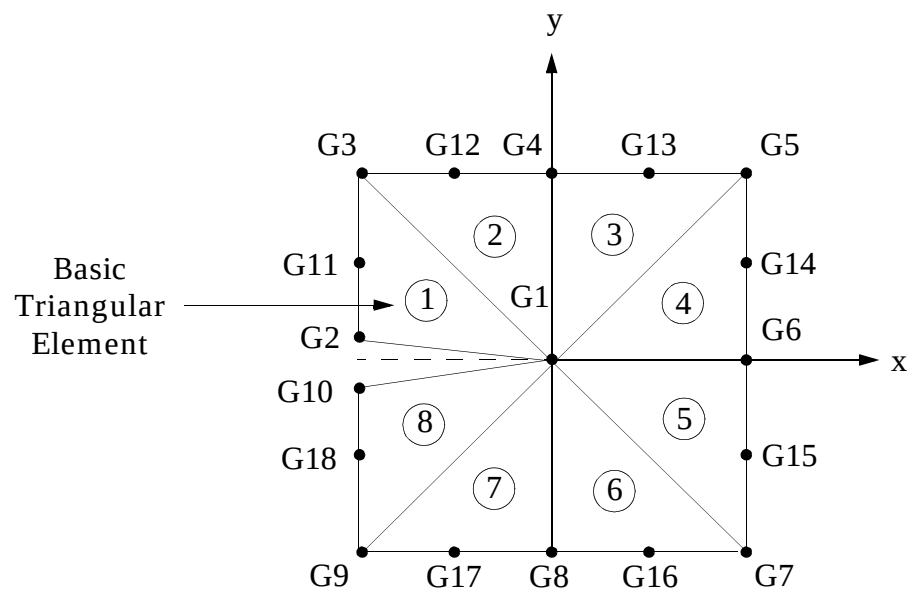
Field	Contents

PID	Property identification number.
MID	Material identification number.
T	Element thickness.
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress.
NSM	Non-structural mass per unit area.

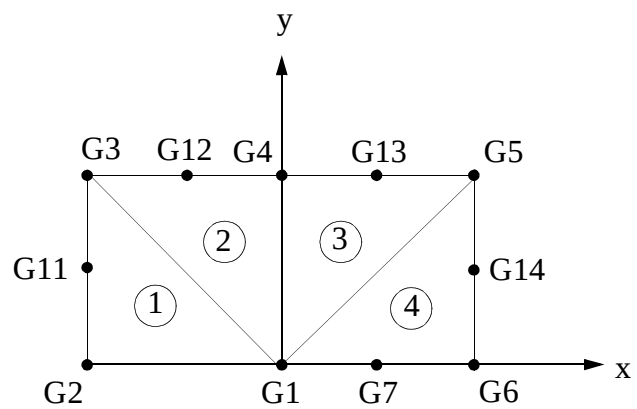
GAMMA	Exponent used in the displacement field.
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated.

The CRAC2D geometry is depicted in [Figure 4-45](#). Two acceptable geometries are shown. Grid points 1 through 10 are required; grid points 11 through 18 are optional. The element may be plane stress or plane strain. You may specify a quadrilateral or a symmetric half option. For the quadrilateral option, MSC.Nastran automatically divides the element into eight basic triangular elements. For the symmetric half-crack option, the program subdivides the element into four basic triangular elements. Only grid points 1 through 7 are required for the symmetric option; grid points 11 through 14 are optional. For the quadrilateral option, the stresses are computed by averaging the stresses from triangles 4 and 5, and the stress intensity factors K_I and K_{II} are computed from triangles 1 and 8. For the symmetric option, the stress is determined from triangle 4, and the stress intensity factor K_I is computed from triangle 1 only.

The CRAC2D element is based on a two-dimensional formulation, but it may be used for three-dimensional planar structures. Deviation from a planar element is checked and any significant deviation results in a printed error message.



(a) Quadrilateral Crack Element



(b) Symmetric Half-Crack Option

Figure 4-45 The CRAC2D Shell Crack Tip Element

4.5 Three-Dimensional Elements

Three-dimensional elements, commonly referred to as solid elements, are used to model structures that cannot be modeled using beam or plate elements. For instance, a solid element is used to model an engine block because of the latter’s three-dimensional nature. If however, you are creating a model of the automobile hood, the best choice is one of the plate elements; you do not want to use the solid elements in this case.

The solid elements discussed in this section are the CHEXA, CPENTA, CTETRA, CTRIAX6, and CRAC3D element. The CTRIAX6 element is an axisymmetric element that is used for axisymmetric analysis only. The CRAC3D element is a three-dimensional crack tip element.

The CHEXA, CPENTA, and CTETRA elements are a family of elements used for general modeling applications. They differ from each other primarily in the number of faces and in the number of connected grid points.

The CHEXA, CPENTA, and CTETRA Elements

The CHEXA element (brick element with six faces) is the most commonly used solid element in the MSC.Nastran element library. The CPENTA (a wedge element with five faces) and CTETRA (a pyramid element with four faces) are used mainly for mesh transitions and in areas where the CHEXA element is too distorted.

The connectivity information for the elements is entered on the CHEXA, CPENTA, and CTETRA entries. For discussion, the format of the CHEXA element is included below. The formats of the CPENTA and CTETRA are similar to that of the CHEXA and therefore are not included.

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

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSOLID or PLSOLID entry.
Gi	Grid point identification numbers of connection points.

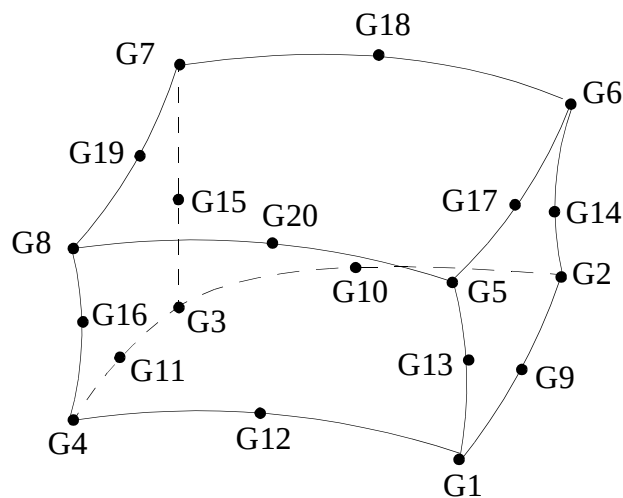


Figure 4-46 CHEXA Element Connection

As can be seen, the CHEXA element is defined by eight corner grid points and up to twelve optional midside grid points. It is important that you observe the ordering of the grid point IDs on the connectivity entries. For example, grid point G12 must lie between grid points G1 and G4. Deviation from the ordering scheme will result in a fatal error, or worse - bad answers. It is not necessary that the midside grid points lie on a straight line between their respective corner grid points. However, it is advisable to keep the edges reasonably straight. In other words, avoid highly distorted elements.

Similar to the case for the CQUAD8 element, any or all of the midside nodes for the CHEXA element may be deleted. However, it is recommended that if midside nodes are used, then all midside grid points be included. The 8-noded or the 20-noded CHEXA should be used in areas where accurate stress data recovery is required.

Solid elements contain stiffness only in the translation degrees of freedom at each grid point. Similar to the normal rotational degrees of freedom for the CQUAD4, you should be aware of the potential singularities due to the rotational degrees of freedom for the solid elements. You may either constrain the singular degrees of freedom manually or you can let MSC.Nastran automatically identify and constrain them for you using the AUTOSPC parameter. The parameter K6ROT does not affect the solid elements. Also any combination of the solid elements with elements that can transmit moments require special modeling. Transitioning elements is discussed in [“Modeling Guidelines”](#) on page 311.

The properties of the solid elements are entered on the PSOLID entry. The format of the PSOLID entry is as follows:

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

Field	Contents

PID	Property identification number.

MID	Identification number of a MAT1, MAT4, MAT5, MAT9, or MAT10 entry.
CORDM	Identification number of the material coordinate system.
IN	Integration network.
STRESS	Location selection for stress output.
ISOP	Integration scheme.
FCTN	Fluid element flag.

The PSOLID entry provides you with an option to define a material coordinate system that can be used to define the material property (an example of this is in “[Material Properties and Composites](#)” in Chapter 5) and also to obtain stresses. The PSOLID entry also provides you with some control over the integration technique used for the element. The CHEXA and the CPENTA are modified isoparametric elements that use selective integration points for different components of strain. There are two different networks of integration points available, each of whose selection depends on the presence of any midside grid points.

The default integration schemes may be overridden with field 7 of the PSOLID entry. In general, it is recommended that you use the default value. An override of the default values is available if you are familiar with element theory and have a particular modeling need; otherwise, use the defaults.

The element coordinate system of the solid elements was modified in Version 67 to improve the repeatability of the coordinate systems of skewed elements when small changes to the model are made. The best way to explain the element coordinate system is to describe how it is generated for the CHEXA. As an example, consider the skewed CHEXA element as shown in [Figure 4-47](#).

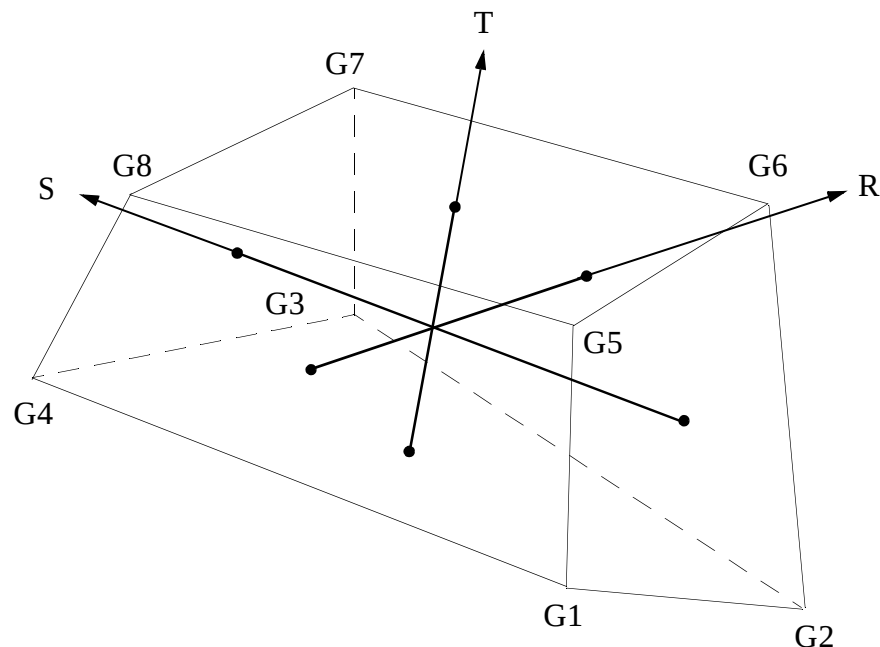


Figure 4-47 CHEXA Element Coordinate System

The first step in generating the element coordinate system is to define three intermediate axes called R, S, and T. The R vector always joins the centroids of the faces G4-G1-G5-G8 and G3-G2-G6-G7. The S vector always joins the faces of G1-G2-G6-G5 and G4-G3-G7-G8. The T vector always joins the faces of G1-G2-G3-G4 and G5-G6-G7-G8. The origin of the coordinate system lies at the intersection of these three vectors. The R, S, and T vectors are not, in general, orthogonal to each other. They are used to define a set of orthogonal vectors R', S', and T' by performing an eigenvalue analysis. The element's x-, y-, and z-axes are then aligned with the same element faces as the R', S', and T' vectors. Since this method does not orient the RST vectors by the grid point IDs, a small perturbation in the geometry does not cause the element coordinate system to change drastically.

This new method for the element coordinate system definition is not biased as the old coordinate system was and provides for more consistent data for postprocessing. However, one drawback is that the actual orientation of the elemental axes is not provided. Therefore, to simplify your work in understanding the stress results, it is advisable not to output stress in the element coordinate system. To output the stress in the basic or any user-defined coordinate system, simply enter the appropriate integer in field 4 of the PSOLID entry. Starting with Version 68, the default coordinate system for the stress output is the basic coordinate system.

As an example of the CHEXA element, consider the tapered beam that was first discussed in “**One-Dimensional Elements**” on page 73. The dimensions of the tapered beam are shown in **Figure 4-15**. Suppose the CBEAM model does not provide the detailed stress contours that you need for your analysis, so you decide to generate a finer solid model. An example of such a model is shown in **Figure 4-48**. The detailed model consists of 3840 CHEXA elements resulting in 15,435 degrees of freedom in the model. Note that four elements are used to model the thickness of the flanges and the web. Four elements are used in order to capture the stress variation through the thickness. A common mistake is not having enough elements across the thickness. Also shown in **Figure 4-48** are deformed plots resulting from the end load.

The input file is not included due to its large size, but it is available (with the name solid1.dat) together with the beam and plate models on the delivery media.

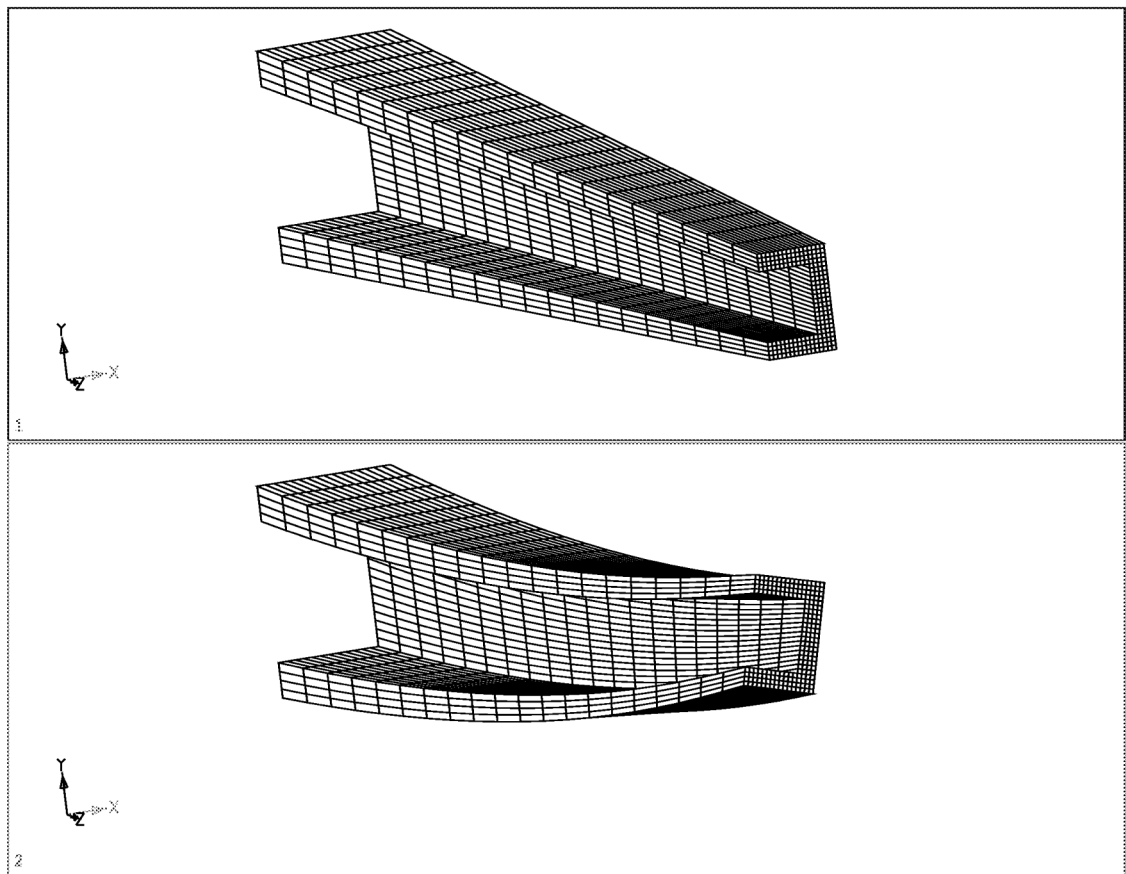


Figure 4-48 Tapered Beam Solid Element Model

The CTETRA element is an isoparametric tetrahedron element with four vertex nodes and up to six additional edge nodes. If you use edge nodes, it is recommended that you include all six nodes. The accuracy of the element degrades if some but not all of the edge grid points are used. It is a good modeling practice to keep the use of the 4-noded CTETRA to a minimum, especially in the areas of high stress.

By default, the stress output for the solid elements is at the center and at each of the corner points. If no midside nodes are used, you may request the stress output at the Gauss points instead of the corner points by setting field 6 of the PSOLID entry to “GAUSS” or 1.

The CTRIAX6 Element

The CTRIAX6 element is a “solid of revolution” element with a triangular cross section. You must supply the three corner grid points and may optionally supply up to three midside nodes. The only application of the CTRIAX6 element is the analysis of axisymmetric structures with axisymmetric loads. This application should not be confused with the cyclic symmetric capability within MSC.Nastran, which can handle both axisymmetric and nonaxisymmetric loading.

Three-Dimensional Crack Tip Element (CRAC3D)

The three-dimensional crack tip element is used to model solids with a discontinuity due to a crack. Like the CRAC2D, the CRAC3D is also a dummy element--you must specify the ADUM9 entry in the Bulk Data Section. The connectivity is entered on the CRAC3D entry. The formats of the CRAC3D and ADUM9 entries are as follows:

1	2	3	4	5	6	7	8	9	10
CRAC3D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20	G21	G22	
	G23	G24	G25	G26	G27	G28	G29	G30	
	G31	G32	G33	G34	G35	G36	G37	G38	
	G39	G40	G41	G42	G43	G44	G45	G46	
	G47	G48	G49	G50	G51	G52	G53	G54	
	G55	G56	G57	G58	G59	G60	G61	G62	
	G63	G64							

Field	Contents

EID	Element identification number.
PID	Property identification number of a PRAC3D entry.
Gi	Grid point identification numbers of connection points.

1	2	3	4	5	6	7	8	9	10
ADUM9	64	0	6	0	CRAC3D				

Grid points 1 through 10 and 19 through 28 are required. All the other grid points are optional. The CRAC3D geometry showing the grid points is depicted in [Figure 4-49](#).

The properties of the CRAC3D element are entered on the PRAC3D entry.

The PRAC3D element property defines the properties of the CRAC3D structural element. The PRAC3D entry has the following format:

1	2	3	4	5	6	7	8	9	10
PRAC3D	PID	MID	GAMMA	PHI					

Field	Contents

PID	Property identification number.
MID	Material identification number.

GAMMA	Exponent used in the displacement field.
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated.

You have two options available, the brick option and the symmetric option. When the brick option is used, MSC.Nastran automatically subdivides the element into eight basic wedges. When you specify the symmetric option, MSC.Nastran subdivides the element into four wedges. For the brick option, the stresses are computed from wedges 4 and 5, and the stress intensity factors K_I and K_{II} are computed from wedges 1 and 8. When the symmetric option is used, the stress is computed from wedge 4, and the stress intensity factor K_I is computed from wedge 1 only.

The CRAC3D element is based upon a 3-D formulation. Both of the faces (formed by grid points 1 through 18 and grid points 19 through 36) and that of the midplane (grid 37 through 46) should be planar. Any significant deviation will result in an error message.

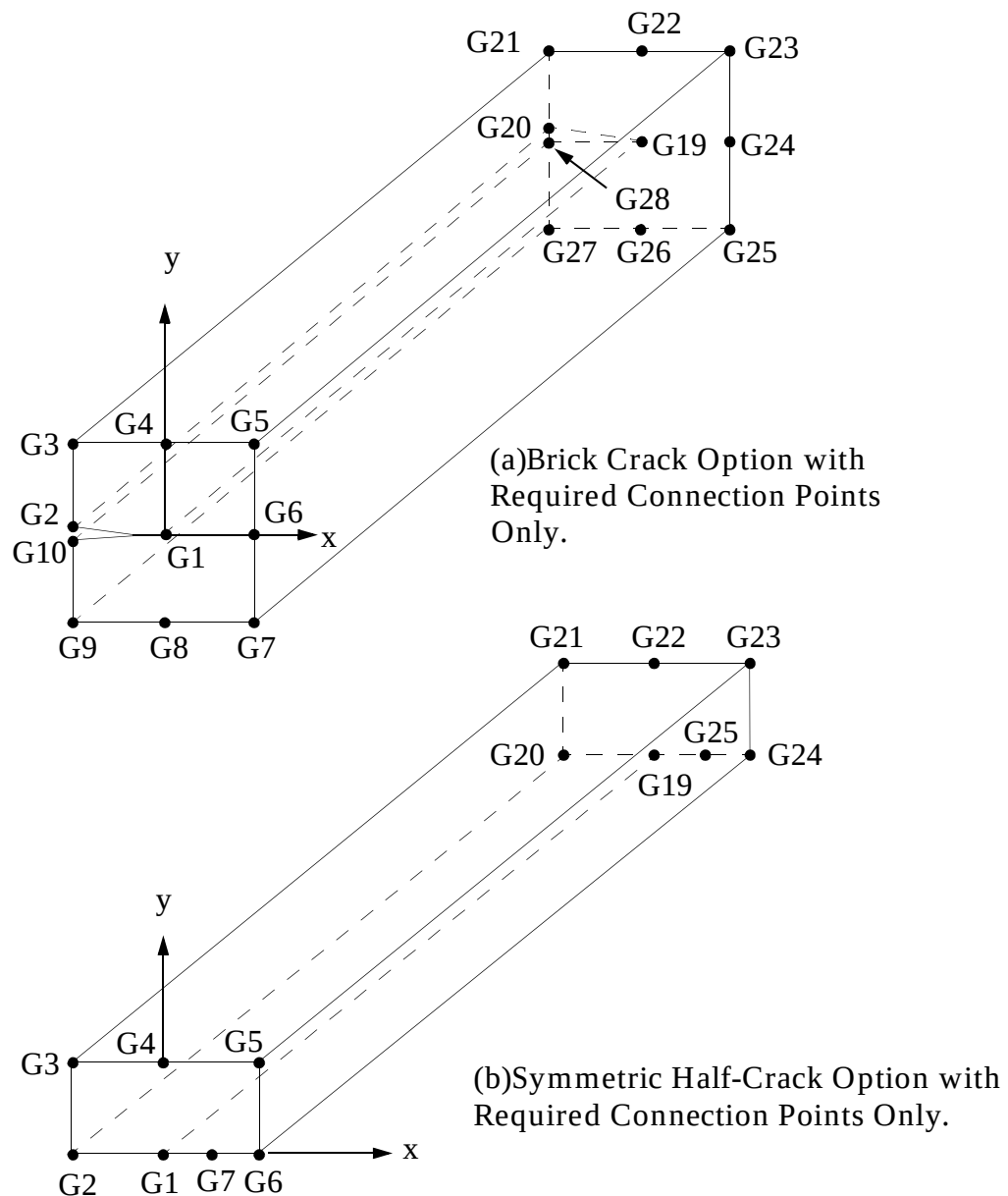


Figure 4-49 CRAC3D Solid Crack Tip Element

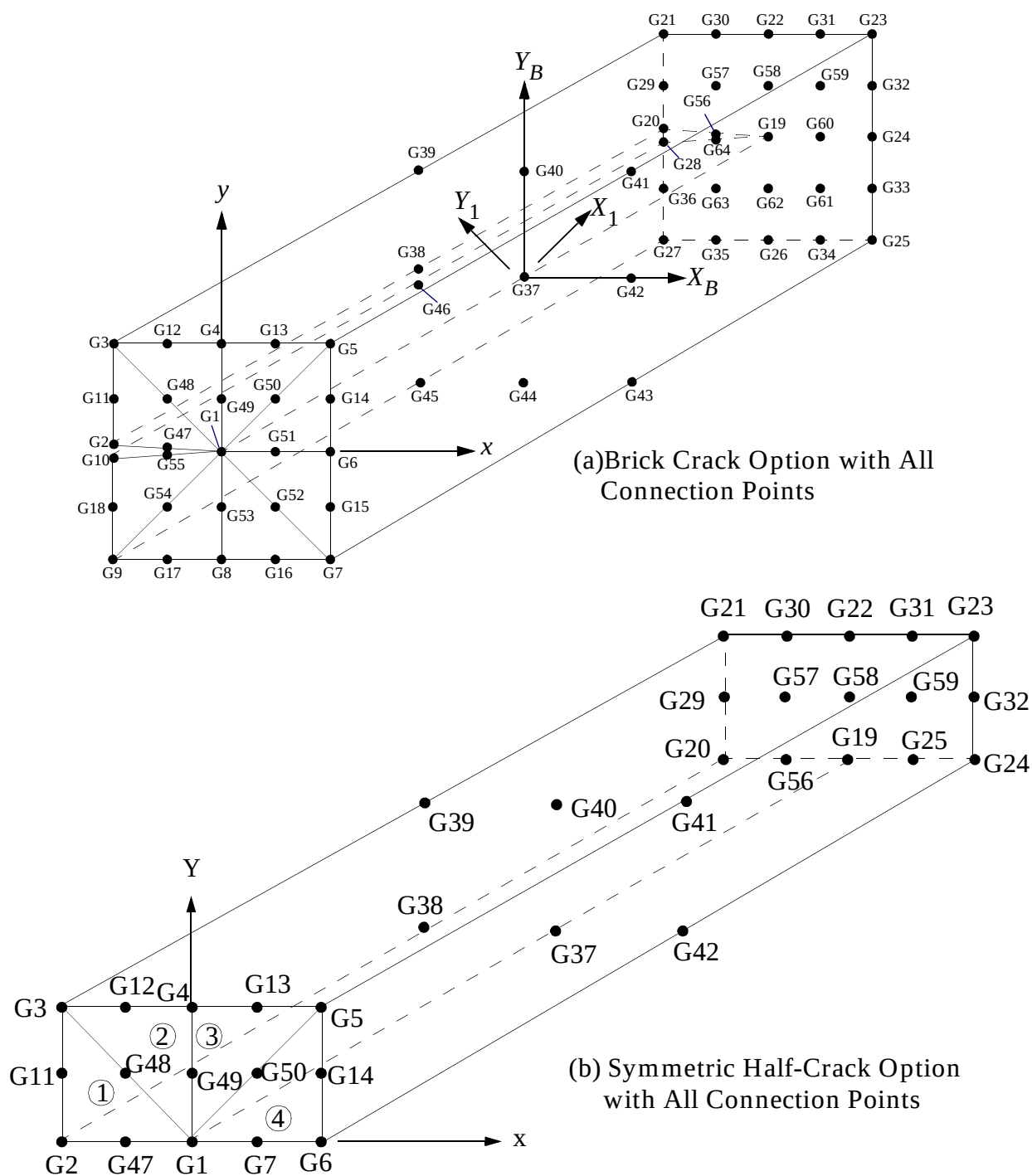


Figure 4-49 CRAC3D Solid Crack Tip Element (continued)

4.6 CBUSH Elements

As discussed in [Figure 4-50](#), if CELASi elements are used and the geometry is not aligned properly, internal constraints may be induced. The CBUSH element contains all the features of the CELASi elements plus avoiding the internal constraint problem. The following example demonstrates the use of the CBUSH element as a replacement for scalar element for static analysis. The analysis joins any two grid points by user-specified spring rates, in a convenient manner without regard to the location or the displacement coordinate systems of the connected grid points. This method eliminates the need to avoid internal constraints when modeling.

The model shown in [Figure 4-50](#) has two sheets of metal modeled with CQUAD4 elements. The sheets are placed next to each other. There are grid points at the common boundary of each sheet of metal, which are joined by spot welds. The edge opposite the joined edge of one of the sheets is constrained to ground. The grid points at the boundary are slightly misaligned between the two sheets due to manufacturing tolerances. There is a nominal mesh size of 2 units between the grid points, with 10 elements on each edge. The adjacent pairs of grid points are displaced from each other in three directions inside a radius of 0.1 units in a pattern that maximizes the offset at one end, approaches zero at the midpoint, and continues to vary linearly to a maximum in the opposite direction at the opposite end.

CELASi elements are used in the first model, and CBUSH elements are used in a second, unconnected model. PLOTTEL elements are placed in parallel with the CELAS2 elements to show their connectivity. The second model is identical to the first model with respect to geometry, constraints, and loading.

A static loading consisting of a point load with equal components in all three directions on the center point opposite the constrained edge is applied. The first loading condition loads only the first model and the second loading condition loads only the second model, allowing comparison of the response for both models in one combined analysis. The input file *bushweld.dat* is delivered in the TPL subdirectory.



Figure 4-50 Spot-Weld Comparison Model

In order to use CELASi elements properly, it is necessary to account for the offsets between the grid points. The most practical method may be to define coordinate systems, which align with a line between each pair of grid points, and then input the CELASi elements along these coordinate systems, which is a tedious, error-prone task. If the grid points are located in non-Cartesian systems or several Cartesian coordinate systems, the task is even more tedious and error prone. Such small misalignment errors are ignored in this model, and the CELAS2 elements are input in the basic coordinate system. The consequence is that internal constraints are built into the model when, for example, the elements are offset in the y-direction, are joined by stiffness in the x- and z-directions, and the element has a rotation about the x or z axis.

CBUSH elements are used to join the plates in the second model. The coordinate system of the elastomer must be defined for each element. The option for defining the elastomer coordinate system, which is useable for all geometry including coincident and very close grid points, is the CID option in field 9 of the CBUSH entry. Since the welds are regarded as isotropic, the basic coordinate system is selected by a value of 0. No other consideration of geometry is needed regardless of the coordinate systems used to define grid point locations or displacement system directions.

Use of PARAM,EST,YES provides the length of each element, a modeling check to ensure that the wrong grid points have not been joined or that the misalignment has not been modeled correctly because the length between connected points is greater than the manufacturing tolerance of 0.1 units. If wrong (nonadjacent) points are inadvertently joined by CELASi elements, large internal constraints can be generated that can be difficult to diagnose. CBUSH elements also appear on structure plots.

The input entries for a spot weld are shown for each modeling method in [Listing 4-11](#). The CBUSH element requires one concise line of nonredundant data per weld, plus a common property entry for all elements, while the CELAS2-based model requires six lines per weld. It would require even more input per weld if the geometry were modeled properly.

Listing 4-11 Spot-Weld Models

```
$ INPUT STREAM FOR ELAS2 MODEL OF ONE SPOT WELD
$ SPOT WELD THE EDGES WITH CELAS ELEMENTS
CELAS2,1, 1.+6, 111000,1, 210000,1
CELAS2,2, 1.+6, 111000,2, 210000,2
CELAS2,3, 1.+6, 111000,3, 210000,3
CELAS2,4, 1.+6, 111000,4, 210000,4
CELAS2,5, 1.+6, 111000,5, 210000,5
CELAS2,6, 1.+6, 111000,6, 210000,6
$ INPUT STREAM FOR BUSH MODEL OF ONE SPOT WELD
$ WELD THE EDGES WITH BUSH ELEMENTS
CBUSH,10000,1, 311000,410000,, , , 0
PBUSH,1, K 1.+6, 1.+6 1.+6 1.+6 1.+6 1.+6
PARAM,EST, 1 $ PRINT THE MEASURE OF ALL ELEMENTS
```

The OLOAD resultants and the SPC-force resultants for the two models ([Figure 4-51](#)) illustrate the effects of internal constraints caused by the misaligned CELASi elements. The first load case is for CELASi modeling and the second line is for CBUSH modeling. When there are no internal constraints, the constraint resultants are equal and opposite to the load resultants. Any unbalance is due to internal constraints.

```

OUTPUT OF LOAD AND SPCFORCE RESULTANTS
SUBCASE 1 IS FOR ELAS ELEMENTS, SUBCASE 2 IS FOR BUSH ELEMENTS

      OLOAD      RESULTANT
      T1         T2         T3         R1         R2         R3
1  4.0000000E+00  4.0000000E+00  4.0000000E+00  1.6000000E+02 -4.0000000E+01 -1.2000000E+02
2  4.0000000E+00  4.0000000E+00  4.0000000E+00  1.6000000E+02 -4.0000000E+01 -1.2000000E+02
. . .

      SPCFORCE RESULTANT
      T1         T2         T3         R1         R2         R3
1 -4.0000000E+00 -4.0000000E+00 -4.0000000E+00 -1.5909122E+02  3.9545681E+01  1.1954554E+02
2 -4.0000000E+00 -4.0000000E+00 -4.0000000E+00 -1.6000000E+02  4.0000000E+01  1.2000000E+02

```

Figure 4-51 Output of Load and SPC-Force Resultants

The resultants in the CBUSH spot weld model balance to the degree of accuracy shown in the printout. The CELASi-based model resultants match to the degree of accuracy shown in the printout for forces, but only to two or three digits for moments. A review of grid point force balance output shows that the grid points attached to CELASi elements are in balance even though they contain internal constraints. This shows the difficulty in diagnosing elements with internal constraints in model check-out condition activities. Evidence of internal constraints are apparent in static analysis when the resultants are not in balance. The analysis does not isolate the elements with internal constraints; instead it merely states that some internal constraints must exist.

Internal constraints provide plausible results for some loading conditions but may provide unexpected results for other loadings or eigenvectors. There is no direct evidence of internal constraints in element forces, grid point forces, SPC forces or other commonly used output data. The internal constraints are hidden SPC moments, which do not appear in SPC force output. Their effect on this model is to reduce the maximum deflection for this loading condition by 0.65%, and the external work (UIM 5293) by 1% when comparing the CELASi model with the CBUSH model due to the (false) stiffening effect of the internal constraints on rotation at the interface between the two sheets of metal.

Another method to model spot welds is through the use of RBAR elements. RBAR elements do not have internal constraints. The CBUSH element should result in lower computation time than the RBAR element when the model contains many spot welds.

In dynamic analysis, the CBUSH elements can be used as vibration control devices that have impedance values (stiffness and damping) that are frequency dependent. See “[Bush Element \(CBUSH\)](#)” on page 225 of the *MSC.Nastran Reference Guide* for further details.

4.7 The GENEL Element

The GENEL element is really not an element in the same sense as the CBAR or CQUAD4 element. There are no properties explicitly defined and no data recovery is performed. The GENEL element is, however, very useful when you want to include in your model a substructure that is difficult to model using the standard elements. The GENEL element is used to describe a substructure that has an arbitrary number of connection grid points or scalar points. The input data entered for the GENEL element may be derived from a hand calculation, another computer model, or actual test data.

There are two approaches that you may use to define the properties of a GENEL element: (1) the stiffness approach, in which case you define the stiffness for the element; and (2) the flexibility approach, in which case you define the flexibility matrix for the element.

1. The stiffness approach:

$$\begin{Bmatrix} f_i \\ f_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ -S^T K & S^T KS \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix}$$

2. The flexibility approach:

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix}$$

where:

$$u_i = [u_{i1}, u_{i2}, \dots, u_{im}]^T$$

$$u_d = [u_{d1}, u_{d2}, \dots, u_{dn}]^T$$

$$KZ = \begin{bmatrix} KZ_{11} & \dots & \dots & \dots \\ KZ_{21} & KZ_{22} & \dots & \dots \\ KZ_{31} & KZ_{32} & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ]$$

$$[K] = [Z] =$$

$$S = \begin{bmatrix} S_{11} & \dots & S_{1n} \\ S_{21} & \dots & \dots \\ S_{31} & \dots & \dots \\ \vdots & \vdots & \vdots \\ S_{m1} & \dots & S_{mn} \end{bmatrix}$$

$$[S] =$$

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body whose rigid body motions are defined by $\{u_i\} = [S]\{u_d\}$.

The required input is the redundant displacement set $\{u_i\}$ list and the lower triangular portion of $[K]$ or $[Z]$ (note: $[Z] = [K]^{-1}$). Additional input may include the determinant $\{u_d\}$ list and $[S]$. If $[S]$ is input, $\{u_d\}$ must also be input.

If $\{u_d\}$ is input but $[S]$ is omitted, $[S]$ is internally calculated. In this case, $\{u_d\}$ must contain six and only six degrees of freedom (translation or rotation, no scalar points). If the $\{u_d\}$ set contains exactly six degrees of freedom, then the $[S]$ matrix computed internally describes the rigid motion at $\{u_i\}$ due to unit values of the components of $\{u_d\}$.

When the $[S]$ matrix is omitted, the data describing the element is in the form of a stiffness matrix (or flexibility matrix) for a redundant subset of the connected degrees of freedom, that is, all of the degrees of freedom over and above those required to express the rigid body motion of the element. In this case, extreme precision is not required because only the redundant subset is input, not the entire stiffness matrix. Using exactly six degrees of freedom in the $\{u_d\}$ set and avoiding the $[S]$ matrix is easier and is therefore recommended. An example of defining a GENEL element without entering an $[S]$ is presented later.

The format of the GENEL entry is as follows:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	-etc.-				

UI_m - The last item in the UI list appears in one of fields 2, 4, 6, or 8.

	"UD"		UD1	CD1	UD2	CD2	-etc.-		

UD_n - The last item in the UD list appears in one of fields 2, 4, 6, or 8.

	"K" or "Z"	KZ11	KZ21	KZ31	-etc.-	KZ22	KZ32		
	-etc.-		KZ33	KZ43	-etc.-				

KZ_{mm} - The last item in the K or Z matrix appears in one of fields 2 through 9.

1	2	3	4	5	6	7	8	9	10
	"S"	S11	S12	-etc.-		S21	-etc.-		

S_{mn} - The last item in the S matrix appears in one of fields 2 through 9.

Field	Contents
EID	Unique element identification number.

UIi, CIi, UDj, CDj	Identification numbers of coordinates in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. Ui and UDi are grid point numbers, and CIi and CDj are the component numbers. If a scalar point is given, the component number is zero.
KZij	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list.
Sij	Values of the [S] matrix ordered by rows, according to the UD list.
“UD”, “K”, “Z”, and “S”	Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices.

For an example of the GENEL element, consider the robotic arm shown in [Figure 4-52](#). The arm consists of three simple bar members with a complex joint connecting members 1 and 2. For the problem at hand, suppose that the goal is not to perform a stress analysis of the joint but rather to compute the deflection of the end where the force is acting. You can make a detailed model of the joint, but it will take a fair amount of time and the results will still be questionable. The ideal choice is to take the joint to the test lab and perform a static load test and use those results directly.

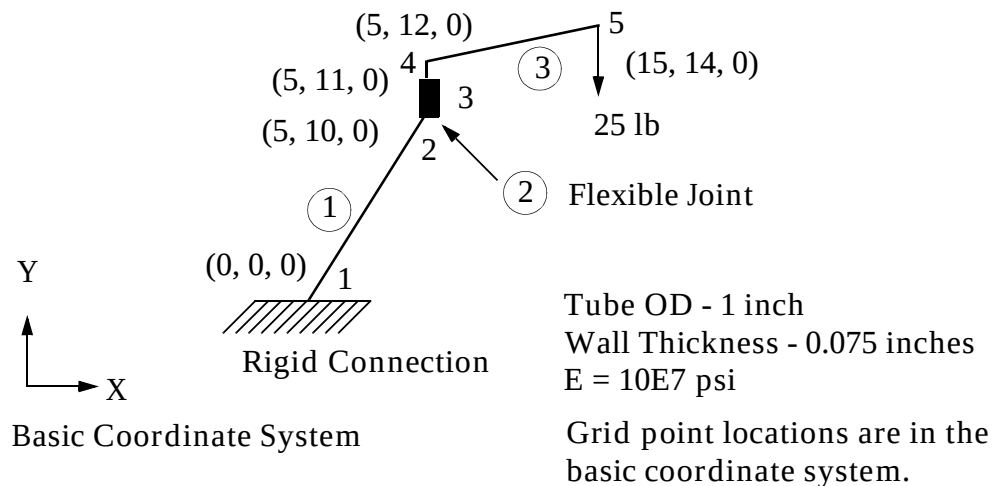


Figure 4-52 Robotic Arm with Joint

The GENEL element is the ideal tool for this job. The joint is tested in the test lab by constraining one end of the joint and applying six separate loads to the other end as shown in [Figure 4-53](#).

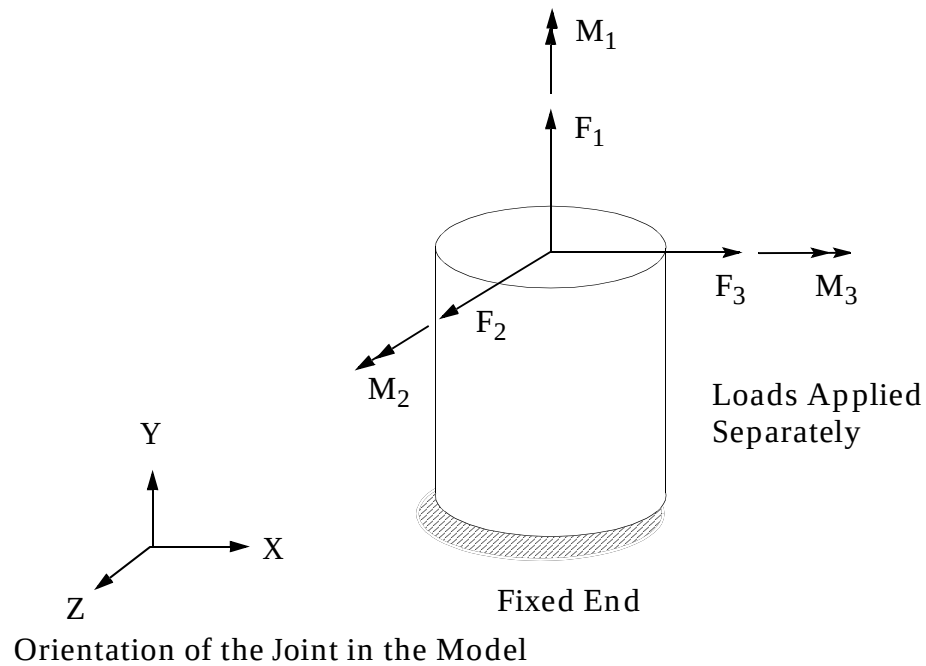


Figure 4-53 The Test Arrangement to Obtain the Flexibility Matrix

Table 4-2 summarizes the displacements measured for each of the applied loads.

Table 4-2 Test Results for the Joint

Deflection (10^{-7}) Due to Unit Loads						
	F_1	F_2	F_3	M_1	M_2	M_3
F_1 DISP	2.2	0	0	0	0	0
F_2 DISP	0	1.58	0	0	0	2.38
F_3 DISP	0	0	1.58	0	-2.38	0
M_1 Rotation	0	0	0	6.18	0	0
M_2 Rotation	0	0	-2.38	0	4.75	0
M_3 Rotation	0	2.38	0	0	0	4.75

Table 4-2 also represents the flexibility matrix of the joint with the rigid body properties removed. By specifying all of the degrees of freedom at grid point 2 as being in the dependent set $\{u_d\}$ and the [S] degrees of freedom at grid point 3 as being in the independent set $\{u_i\}$, the matrix is not required. The input file showing the model with the GENEL is shown in **Listing 4-12**.

Listing 4-12 GENEL Element Input File

```

$
$FILENAME - GENEL1.DAT
$
CORD2R  1      0      0.      0.      0.      1.      0.      0.
        0.      1.      0.
GENEL   99      2      1      2      2      2      3
        2      4      2      5      2      6
        UD      3      1      3      2      3      3
        3      4      3      5      3      6
        Z      2.20E-7 0.0      0.0      0.0      0.0      0.0      1.58E-7
        0.0      0.0      0.0      2.38E-7 1.58E-7 0.0      -2.38E-7 0.0
        6.18E-7 0.0      0.0      4.75E-7 0.0      4.75E-7
$
GRID    1      0.0      0.0      0.0
GRID    2      5.      10.      0.0      1
GRID    3      5.      11.      0.0      1
GRID    4      5.      12.      0.0
GRID    5      15.      14.      0.0
$
CBAR    1      1      1      2      1.0      1.0      0.0
CBAR    2      1      3      4      1.0      1.0      0.0
CBAR    3      1      4      5      1.0      1.0      0.0
$
FORCE   1      5      0      25.      0.0      -1.      0.0
SPC     1      1      123456 0.0
PBAR    1      1      .2179  .02346  .02346  .04692
$
MAT1    1      1.+7      .3

```

The flexibility matrix generated from the test data was properly aligned with the model geometry with the use of a local coordinate (note the CD field of grid points 2 and 3). Using this local coordinate system, the output displacement X-axis corresponds to the F1 direction, etc.

“**Direct Matrix Input**” on page 471 provides you with an alternative approach of inputting structure matrices using the DMIG entries.

4.8 CWELD Connector Element

Introduction

In the past, MSC.Nastran has offered various ways of modeling structural connections and fasteners. Spot welds, seam welds, bolts, screws, and so on can be represented, depending on the modeling goals, either with flexible springs or bars (CBUSH, CBAR), rigid elements (RBAR, RBE2, RBE3), or multipoint constraints (MPC). Though general, these elements are sometimes difficult to use; singularities may be introduced particularly in the out-of-plane rotational direction for shells, rigid body invariance may not be assured, and data preparation and input can be a formidable task in real-world applications. Increasing mesh refinement can also introduce further stiffness errors; point-to-point connections in which effective cross-sectional areas are larger than 20% of the characteristic element lengths can often lead to significant underestimation of connector stiffness.

The new CWELD element and corresponding PWELD property entries have been introduced to address and solve such modeling issues. Connections can now be established with ease between points, elements, patches, or any of their combinations. In short, the new CWELD connector element is general in purpose, easy to generate, less error-prone than any of the preceding methods, and always satisfies the condition of rigid body invariance. Test and analysis correlation has also been shown to be excellent.

Connectivity Definition

Weld element connectivity is defined using the new CWELD Bulk Data entry. (See “[Bulk Data Entries](#)” on page 849 of the *MSC.Nastran Quick Reference Guide* for a complete description of the new entry.) Connections can either be established conventionally, from point-to-point, or in a more advanced fashion between elements and / or patches of grid points. In the case of elements and patches, actual weld attachment points will usually occur within element domains or patches and will be computed automatically, with corresponding automatic creation of necessary grid points and degrees-of-freedom. Element- and patch-based connections, moreover, eliminate the need for congruent meshes. Reference grids that determine spot weld spacing, for example, can be defined beforehand which, when projected (using the CWELD entry) through the surfaces to be attached, uniquely determine the weld elements’ location and geometry. The following is a brief description of the various CWELD connectivity options:

1. Patch-to-Patch

In the context of CWELD element definition, a patch is a surface to which the weld element will connect. Actually, two patches must be defined on the CWELD entry in order to define a valid connection. The attachment locations on these surfaces are determined by vector projection from a single grid point, GS, also referenced on the CWELD entry. The patches are defined either by reference to a shell element ID or by an ordered list of grid points on a surface. See [Figure 4-54](#).

The “ELEMID” option defines a connection between two shell elements:

```
CWELD, EWID, PWID, GS, “ELEMID”, , , , +CWE1
+CWE1, SHIDA, SHIDB
```

EWID is the identification number of the CWELD element and PWID is the identification number of the corresponding PWELD property entry. The projection of a normal vector through grid point GS determines the connection locations on the shell elements SHIDA and SHIDB. The grid point GS does not need to lie on either of the element surfaces.

Instead of shell element IDs, an ordered list of grids could have been used to define a surface patch as seen in the following CWELD example:

```
CWELD, EWID, PWID, GS, "GRIDID", , , "QT", , +CWG1  
+CWG1, GA1, GA2, GA3, GA4, , , , +CWG2  
+CWG2, GB1, GB2, GB3
```

The "GRIDID" option affords a more general approach to surface patch definition, based on an ordered list of grids. Such patches can either be triangular or rectangular with anywhere from three to eight grids points. Ordering and numbering conventions follow directly from the CTRIA3 and CTRIA6 entries for triangular patches, and the CQUAD4 and CQUAD8 entries for rectangular patches (mid side nodes can be omitted). Since the program is generally unable to tell a TRIA6 with two mid side nodes deleted from a QUAD4, it becomes necessary to also indicate the nature of the patches involved, hence the string "QT" in the preceding CWELD example, for "quadrilateral to triangular surface patch connection." (See Remark 5 on the Bulk Data entry, "**CWELD**" on page 1202 of the entry for other options.)

With the patch-to-patch connection, non-congruent meshes of any element type can be connected. Patch-to-patch connections are recommended when the cross sectional area of the connector is larger than 20% of the characteristic element face area.

The patch-to-patch connection can also be used to connect more than two layers of shell elements. For example, if three layers need to be connected, a second CWELD element is defined that refers to the same spot weld grid GS as the first CWELD. Patch B of the first CWELD is repeated as patch A in the second CWELD.

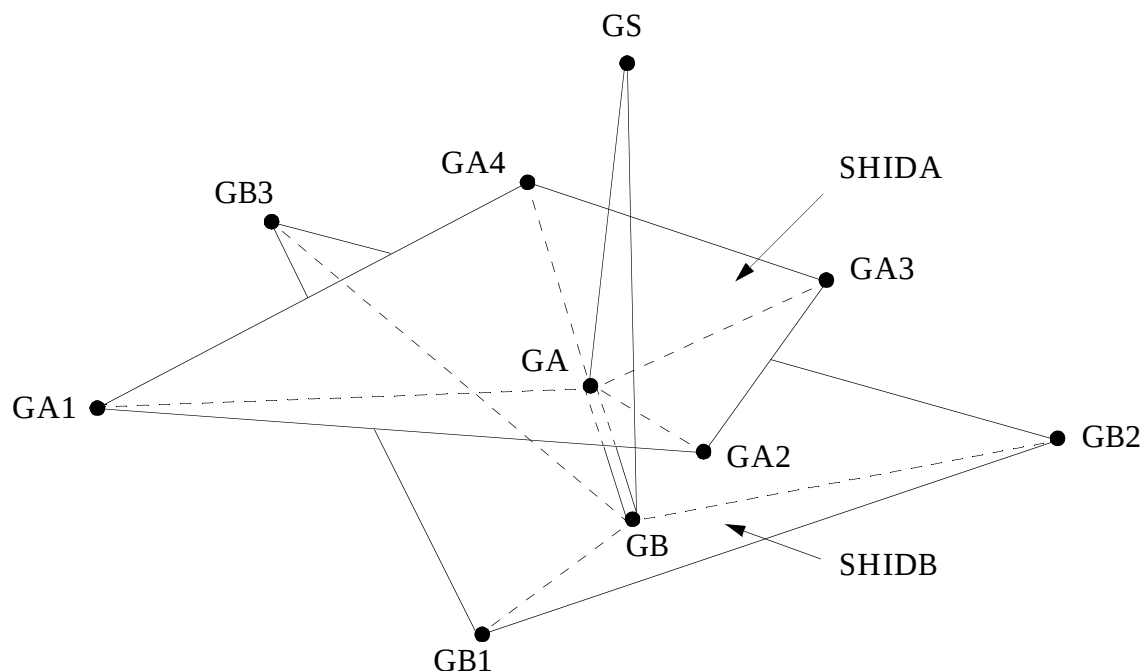


Figure 4-54 Patch-to-Patch Connection Defined with Format GRIDID or ELEMID

2. Point-to-Patch

The “GRIDID” and “ELEMID” formats can also be used to define the connection of a point to a surface patch and again, is a useful method of joining non-congruent meshes. As before, the patch can either be triangular or quadrilateral:

CWELD, EWID, PWID, GS, “GRIDID”, , , “Q”, , +CWP1
+CWP1, GA1, GA2, GA3, GA4

For a point-to-patch connection, the vertex grid point GS of a shell element is connected to a surface patch, as shown in [Figure 4-55](#). The patch is either defined by grid points GA_i or a shell element SHIDA. A normal projection of grid point GS on the surface patch A creates grid GA. The vector from grid GA to GS defines the element axis and length. A shell normal in the direction of the element axis is automatically generated at grid GS. Point-to-patch connections are recommended if a flexible shell (GS) is connected to a stiff part (GA_i).

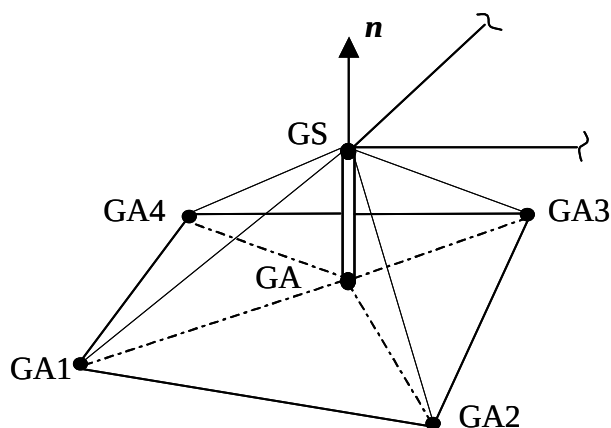


Figure 4-55 Point-to-Patch Connection

3. Point-to-Point

The “ALIGN” format defines a point-to-point connection:

CWELD, EWID, PWID, , “ALIGN”, GA, GB

In the point-to-point connection, two shell vertex grids, GA and GB, are connected as shown in [Figure 4-56](#). The vector from GA to GB determines the axis and length of the connector. Two shell normals are automatically generated for the two shell vertex grids, and both normals point in the direction of the weld axis.

The point-to-point connection can only be used for shell elements, and the two layers of shell meshes must be nearly congruent. The element axis (the vector from GA to GB) has to be nearly normal to the shell surfaces. The point-to-point connection is recommended only when the cross sectional area of the connector does not exceed 20% of the characteristic shell element area.

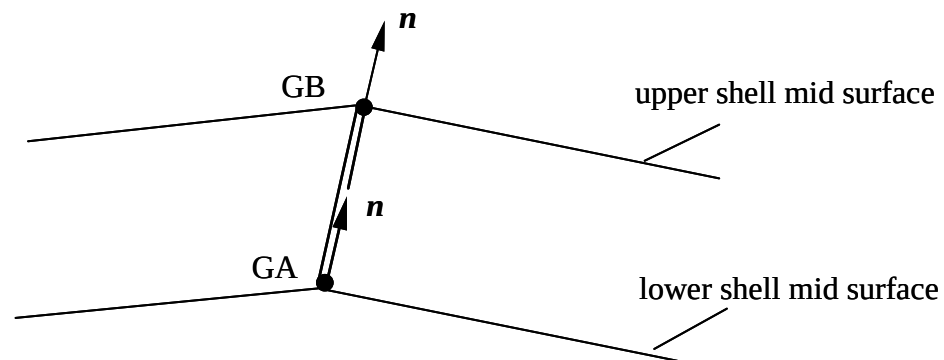


Figure 4-56 Point-to-Point Connection

Property Definition

CWELD element properties are defined with the PWELD entry (see “[Bulk Data Entries](#)” on page 849 of the *MSC.Nastran Quick Reference Guide*):

PWELD, PWID, MID, D, , , MSET, , TYPE

MID references a material entry, D is the diameter of the element, and MSET is used to indicate whether the generated constraints will explicitly appear in the *m*-set or be reduced out at the element level prior to global stiffness matrix assembly. Further parameters and flags will more than likely be introduced in the future as additional options are implemented.

CWELD element constraint generation for patch-based connections are under some measure of user control. By default, the constraints are expressed as multi-point constraint equations. This is the MSET=ON option. For output purposes, the additional *m*-set constraints are labeled “RWELD”, with identification numbers beginning with the ID specified on the PARAM, OSWELM entry.

Constraint equations can be incorporated directly into the element stiffness matrices. The MSET=OFF option instead applies the CWELD element constraint set locally, without generating additional m -set equations. The local constraint effects are reduced out prior to system stiffness matrix assembly resulting in a more compact set of equations, and the expected benefits in numerical performance and robustness. The MSET = OFF option is only available for the patch-to-patch connection (GRIDID or ELEMID).

TYPE identifies the type of connection: “Blank” for a general connector, and “SPOT” for a spot weld connector. The connection type influences effective element length in the following manner: If the format on the CWELD entry is “ELEMID”, and TYPE= “SPOT”, then regardless of the distance GA to GB, the effective length L_e of the spot weld is set to $L_e = 1/2(t_A + t_B)$ where t_A and t_B are the thickness of shell A and B, respectively. If TYPE is left blank (a general connector), then the length of the element is not modified as long as the ratio of length to diameter is in the range $0.2 \leq L/D \leq 5.0$. If L is below the range, the effective length is set to $L_e = 0.2D$. If L is above this range, the effective length is instead set to $L_e = 5.0D$.

Finite Element Representation of the Connector

For all connectivity options, the CWELD element itself is modeled with a special shear flexible beam-type element of length L and a finite cross-sectional area which is assumed to be a circle of diameter D , as shown in [Figure 4-57](#). Other cross sections will be added in the future. The length L is the distance from GA to GB. An effective length is computed if TYPE=“SPOT” or if grids GA and GB are coincident. The element has twelve (2x6) degrees-of-freedom.

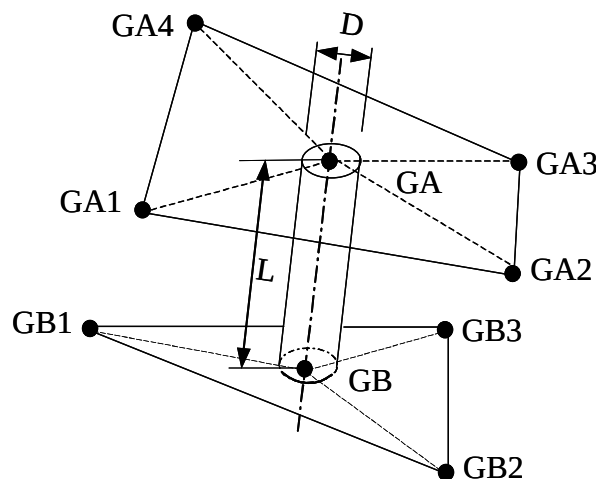


Figure 4-57 CWELD Element

For the point-to-patch and patch-to-patch connection, the degrees-of-freedom of the spot weld end point GA are constrained as follows: the 3 translational and 3 rotational degrees-of-freedom are connected to the 3 translational degrees-of-freedom of each grid GA_i with constraints from Kirchhoff shell theory,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_A = \sum N_i(\xi_A, \eta_A) \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_i \quad \text{Eq. 4-8}$$

$$\begin{aligned} \theta_x^A &= \frac{\partial w}{\partial y} = \sum N_{i,y} \cdot w_i \\ \theta_y^A &= -\frac{\partial w}{\partial x} = -\sum N_{i,x} \cdot w_i \\ \theta_z^A &= \frac{1}{2} \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) = \frac{1}{2} (\sum N_{i,x} \cdot v_i - \sum N_{i,y} u_i) \end{aligned} \quad \text{Eq. 4-9}$$

These 6 equations are written in the local tangent system of the surface patch at point GA. The two tangent directions are x and y, and the normal direction is z. N_i are the shape functions of the surface patch; ξ_A and η_A are the normalized coordinates of GA; u, v, w are the displacements; and θ_x , θ_y , θ_z are the rotations in the local tangent system at GA. For the patch-to-patch connection, another set of 6 equations similar to [Eq. 4-8](#) and [Eq. 4-9](#) is written to connect grid point GB to GBi. The patch-to-patch connection results in 12 constraint equations.

In summary, with MSET="ON", the CWELD element consists of a two node element with 12 degrees-of-freedom. In addition, 6 constraint equations are generated for the point-to-patch connection or 12 constraint equations for the patch-to-patch connection. The degrees-of-freedom for GA and GB are put into the dependent set (*m*-set).

If a patch-to-patch connection is specified and MSET="OFF", the 12 constraint equations are instead included in the stiffness matrix, and the degrees-of-freedom for GA and GB are condensed out. No *m*-set degrees-of-freedom are generated and the subsequent, sometimes costly, *m*-set constraint elimination is avoided. The resulting element is 3xN degrees-of-freedom, where N is the total number of grids GAi plus GBi. This maximum total number of grids is 16, yielding an element with a maximum 48 degrees-of-freedom.

Results Output

The element forces are output in the element coordinate system, see [Figure 4-58](#). The element x-axis is in the direction of GA to GB. The element y-axis is perpendicular to the element x-axis and is lined up with the closest axis of the basic coordinate system. The element z-axis is the cross product of the element x- and y-axis. The CWELD element force output and sign convention is the same as for the CBAR element.

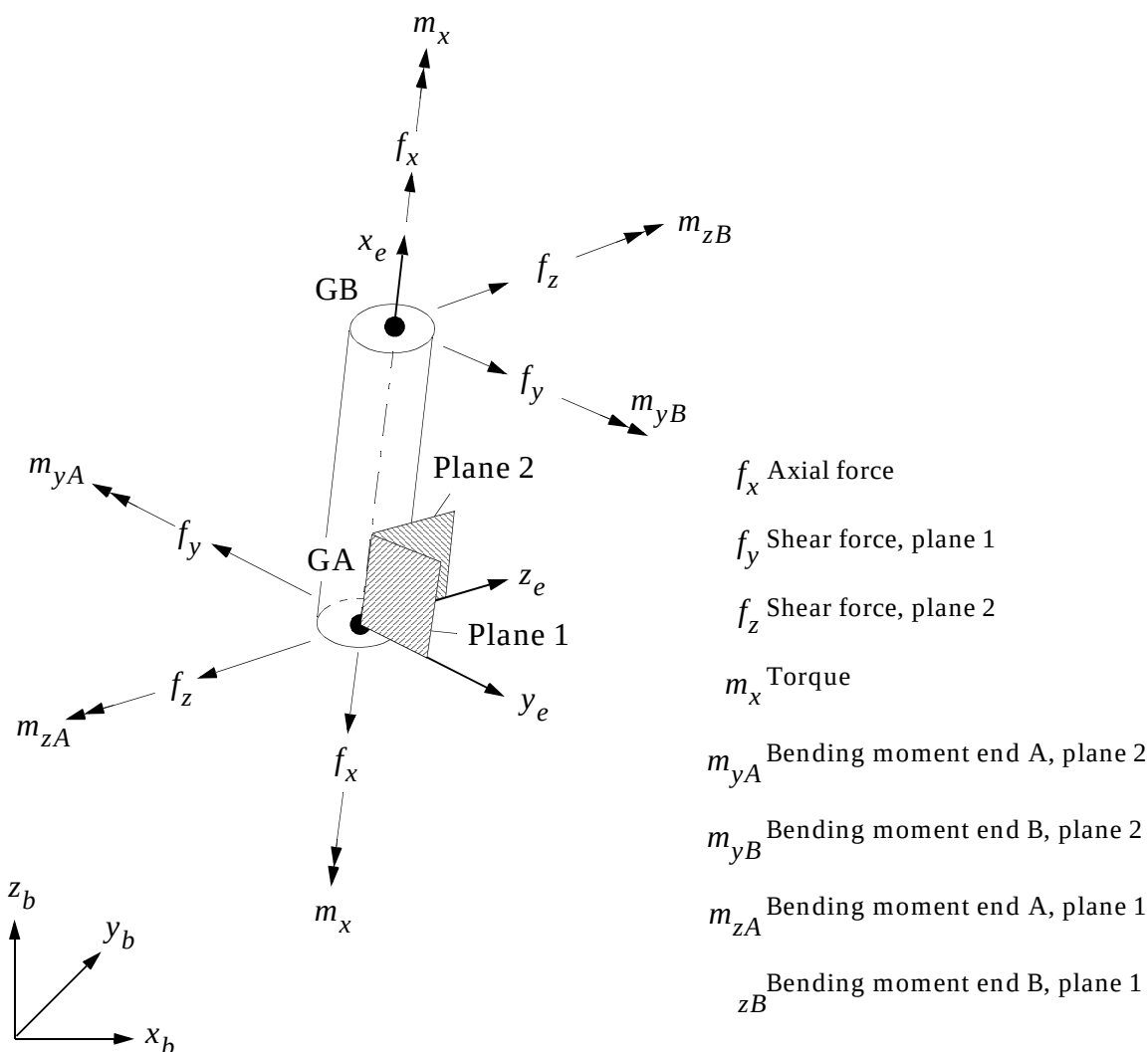


Figure 4-58 Element Coordinate System and Sign Convention of Element Forces

Example (CWELD101a-b.DAT)

In the following example, two cylindrical shell segments are connected with 4 CWELD elements at the 4 corners of the overlapping sections, see [Figure 4-59](#). The results of the patch-to-patch connection are compared to the results of the point-to-point connection. For the patch-to-patch connection, we place the four CWELD elements on the corner shells of the overlapping area. For the point-to-point connection, we take the inner vertex points of the corner shells. The deflection at grid point 64 for the patch-to-patch connection is lower (stiffer) than from the point-to-point connection (1.6906 versus 1.9237). The difference is significant in this example because of the coarse mesh and because the connection of the two shells is modeled with only 4 welds. In most practical problems, the patch-to-patch connection produces stiffer results than the point-to-point connection.

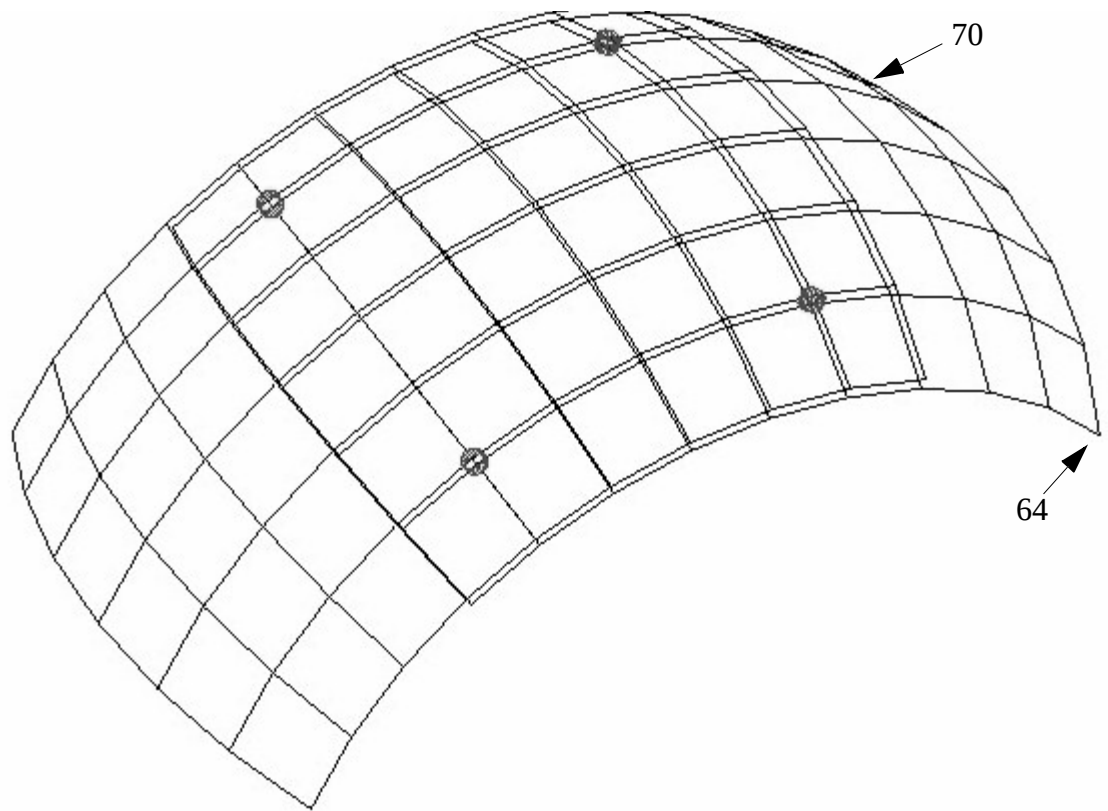


Figure 4-59 Two Spherical Segments with Spot Welds at the Four Corners

Common Input

```

$
$ Nodes of the connected elements
$
GRID      1      21.6506 37.5   -25.    1
GRID      2      23.4923 40.6899-17.101 1
GRID      6      23.4923 40.6899 17.101 1
GRID      7      21.6506 37.5    25.    1
GRID      8      14.8099 40.6899-25.    1
GRID      9      16.0697 44.1511-17.101 1
GRID     13      16.0697 44.1511 17.101 1
GRID     14      14.8099 40.6899 25.    1
GRID     36     -14.81   40.6899-25.    1
GRID     37     -16.0697 44.1511-17.101 1
GRID     41     -16.0697 44.1511 17.101 1
GRID     42     -14.81   40.6899 25.    1
GRID     43     -21.6507 37.5   -25.    1
GRID     44     -23.4924 40.6899-17.101 1
GRID     48     -23.4924 40.6899 17.101 1
GRID     49     -21.6507 37.5    25.    1
GRID     92     22.1506 38.3661-25.    1
GRID     93     23.9923 41.5559-17.101 1
GRID     97     23.9923 41.5559 17.101 1
GRID     98     22.1506 38.366  25.    1
GRID     99     15.1519 41.6296-25.    1
GRID    100     16.4117 45.0908-17.101 1
GRID    104     16.4117 45.0908 17.101 1
GRID    105     15.1519 41.6296 25.    1
GRID    127     -15.152  41.6296-25.    1
GRID    128     -16.4118 45.0908-17.101 1
GRID    130     -17.4431 47.9243 2.435-5 1
GRID    131     -17.1833 47.2105 8.68244 1
GRID    132     -16.4118 45.0908 17.101 1
GRID    133     -15.152  41.6296 25.    1
GRID    134     -22.1506 38.366 -25.    1
GRID    135     -23.9923 41.5559-17.101 1
GRID    139     -23.9923 41.5559 17.101 1
GRID    140     -22.1506 38.366  25.    1
GRID    141      0.      0.      0.      2
$
$ Grid points GS for spot weld location
$
GRID    142      16.2407 44.6209-17.101
GRID    143      16.2407 44.6209 17.101
GRID    144     -16.2407 44.6209-17.101
GRID    145     -16.2407 44.6209 17.101
$
$ Referenced Coordinate Frames
$
CORD2C   1      0.      0.      0.      0.      0.      1.
          1.217-8 1.      0.
CORD2R   2      0.      0.      0.      0.      0.      1.
          1.      0.      0.
$
$ CQUAD4 connectivities
$
CQUAD4   1      1      1      2      9      8
CQUAD4   6      1      6      7      14     13
CQUAD4  31      1      36     37     44     43
CQUAD4  36      1      41     42     49     48
CQUAD4  73      1      92     93     100    99
CQUAD4  78      1      97     98     105   104
CQUAD4  103     1     127    128     135   134
CQUAD4  108     1     132    133     140   139
$
$ shell element property t=1.0
$
PSHELL   1      1      1.      1          1
$
$ Material Record : Steel
$
MAT1     1      210000.      .3      7.85-9
$
$ PWELD property with D= 2.0
$
PWELD    200     1 2.

```

Input for the Patch-to-Patch Connection

```

$
$ CWELD using ELEMID option, outer diagonal elements
$
CWELD    109     200     142     ELEMID
          1      73
CWELD    110     200     143     ELEMID
          6      78
CWELD    111     200     144     ELEMID
          31     103
CWELD    112     200     145     ELEMID
          36     108

```

Input for the Point-to-Point Connection

```
$
$CWELD ALIGN option
$
CWELD 109 200 ALIGN 9 100
CWELD 110 200 ALIGN 13 104
CWELD 111 200 ALIGN 37 128
CWELD 112 200 ALIGN 41 132
```

Output of Element Forces

0									
SUBCASE 1									
T S (C W E L D)					F O R C E S I N W E L D E L E M E N T				
ELEMENT		BEND-MOMENT END-A			BEND-MOMENT END-B				
		- SHEAR -			AXIAL				
ID		PLANE 1 (MZ)		PLANE 2 (MY)	FORCE FX	PLANE 1 (MZ)		PLANE 2 (MY)	
109	PLANE 1 (FY)	PLANE 2 (FZ)				TORQUE MX			
	-2.908656E+02	-1.447359E+02		1.586358E+02		7.728133E+01		1.390482E+01	
110		2.135744E+02		-8.345229E+01	0.0				
	2.908643E+02	2.135769E+02		1.586297E+02		-7.728750E+01		1.390158E+01	
111		-1.447281E+02		-8.346558E+01	0.0				
	3.254667E+00	-2.363556E+02		-3.536731E+02		-2.396099E+02		-2.361487E+02	
112		1.175355E+02		-1.198793E+01	0.0				
		2.363570E+02		-3.536719E+02		2.396101E+02		-2.361489E+02	
	-3.253387E+00	1.175340E+02		-1.198794E+01	0.0				

Guidelines and Remarks

- The CWELD element is available in all solution sequences.
- Material or geometric nonlinear capabilities have not yet been implemented.
- SNORM Bulk Data entries are not necessary.
- PARAM,K6ROT is not necessary.
- The patch-to-patch connection is sufficiently accurate if the ratio of the cross sectional area to the surface patch area is between 10% and 100%, see [Figure 4-60](#).
- With the GRIDID or ELEMID option, the cross section of the connector may cover up to 8 grid points if a quadrilateral surface patch with mid side nodes is defined, see [Figure 4-60](#).
- The CWELD element does not support buckling analysis.

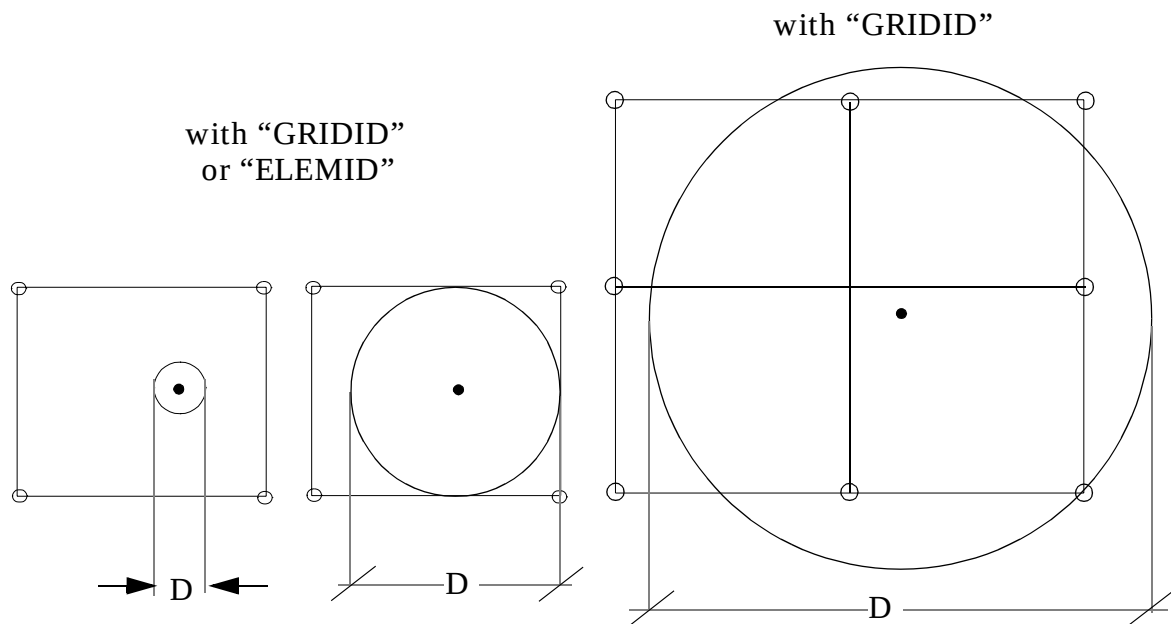


Figure 4-60 Range of Cross-sectional Area versus Element Size for the Patch-to-Patch Connection

- For the patch-to-patch and the point-to-patch connection, the projected grid points GA or GB may lie on an edge of the surface patches or may coincide with a grid point GAI or GBI. The connection is valid as long as GA and GB lie within the surface A and B, respectively. If GA or GB lie outside the surface but inside a tolerance of 5% of the element length, then they are moved on to the surface. In extreme cases, the patch-to-patch type connects elements that are not overlapping (see [Figure 4-61](#)). Although the connection is valid, the CWELD may become too stiff.

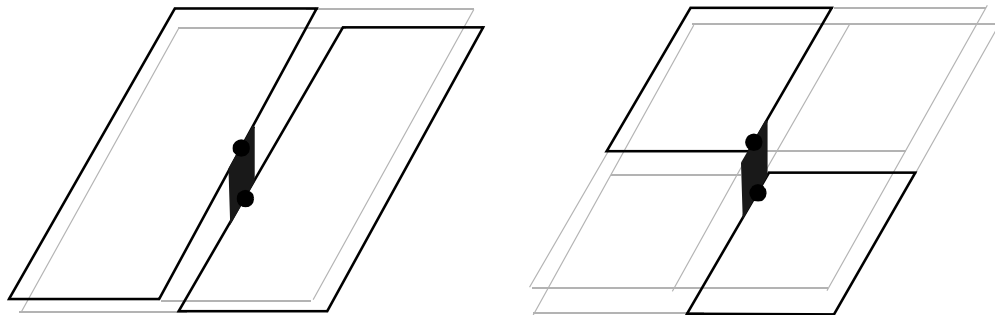


Figure 4-61 Non-overlapping Elements in the Patch-to-Patch Connection

Material Properties and Composites

- Introduction to Material Properties
- Isotropic Material (MAT1)
- Two-Dimensional Anisotropic Material (MAT2)
- Axisymmetric Solid Orthotropic Material (MAT3)
- Two-Dimensional Orthotropic Material (MAT8)
- Three-Dimensional Anisotropic Material (MAT9)
- The PSHELL Entry
- The Composite Element (PCOMP)
- Equivalent Composite Honeycomb Section

5.1 Introduction to Material Properties

MSC.Nastran supports isotropic, orthotropic and anisotropic materials. All of the examples up to this point have involved isotropic material properties only. The purpose of this chapter is to describe all of the material property types that are available to you for linear static analysis.

The material definitions discussed in this chapter include:

- Isotropic materials (MAT1 entry) -- An isotropic material property is defined as a material having the same properties in each direction. This material may be used with all MSC.Nastran linear elements.
- Two-dimensional anisotropic material (MAT2 entry) -- Material definition for the plate and shell elements. The in-plane material properties are defined with respect to an element material coordinate system. Transverse shear material properties may be included.
- Axisymmetric solid orthotropic material (MAT3) -- A three-dimensional material property for axisymmetric analysis only.
- Two-dimensional orthotropic material (MAT8) -- Defines an orthotropic material property for plate and shell elements. Transverse shear material properties may also be included.
- Three-dimensional anisotropic material (MAT9) -- Defines an anisotropic material property of solid elements. The MAT9 entry may also be used to define a three-dimensional orthotropic material.

Also presented in this chapter is the use of the composite analysis in MSC.Nastran. For two-dimensional analysis involving composite material properties, two options are available. The first method is to use the PSHELL entry directly with the appropriate material definitions and geometric parameters. This method is sufficient for simple composites; however, for most composites, determining the appropriate material definitions is a tedious process. The second method available to you is the PCOMP entry. The input on the PCOMP entry consists of the layer-by-layer material definition. MSC.Nastran computes the equivalent PSHELL entries for you. An additional benefit of the PCOMP entry is the ability to obtain individual layer stresses and strains, inter-laminar stress, and failure indices.

For three-dimensional composite analyses, you must develop the anisotropic material matrix yourself. This material matrix is typically entered using the MAT9 entry.

Examples are presented in this chapter showing the use of the PCOMP entry and the MAT9 entry for composite analysis.

5.2 Isotropic Material (MAT1)

The isotropic material, defined by the MAT1 entry, is the most commonly used material property. An isotropic material is defined as a material that has the same material properties in any direction. Furthermore, the isotropic material is fully described by only two material constants. These two constants may be any combination of E, G, and ν . You may specify all three of these constants if desired, but remember, it only takes two of the constants to define the material. When you enter only two constants, the third is computed from the following relationship:

$$G = \frac{E}{2(1 + \nu)} \quad \text{Eq. 5-1}$$

If you enter all three constants and they do not satisfy this relationship, all three constants are used, but that may not be your intention. Therefore, a warning message is printed indicating that the isotropic relationship has been violated. It is recommended that you only input two of the three constants. For line elements, E is the modulus of elasticity, and G is the shear modulus for torsion and transverse shear if it is present in the element. For plate and solid elements, E, G, and ν are used to develop a material matrix for the element.

The MAT1 entry may also be used to define the mass density, coefficient of thermal expansion, and stress limits. The mass properties are only required in static analysis when a gravity loading or rotating force is used; however, they are useful for model checkout with any loading condition (of course, they are very important for dynamic analysis). The format of the MAT1 Bulk Data entry is as follows:

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

Field	Contents
MID	Material identification number.
E	Young's modulus.
G	Shear modulus.
NU	Poisson's ratio.
RHO	Mass density.
A	Thermal expansion coefficient.
TREF	Reference temperature.
ST, SC, SS	Stress limits for tension, compression, and shear.
MCSID	Material coordinate system identification number.

Note that GE shown in field 9 was not described. The GE is a material damping that has no significance for static analysis. Information regarding material damping can be found in the ***MSC.Nastran Basic Dynamics User's Guide***.

5.3 Two-Dimensional Anisotropic Material (MAT2)

The MAT2 entry is used to specify a general anisotropic two-dimensional stress-strain relationship of the form shown in [Eq. 5-2](#) and [Eq. 5-3](#). The MAT2 entry can only be used with plate and shell elements. The reference temperature is given by T_{REF} and the thermal expansion coefficients are A1, A2, and A3. The component directions X and Y refer to the element material coordinate system, which is explicitly defined for each element. The material coordinate system for the CQUAD4 element is shown in [Figure 4-34](#). The in-plane stress-strain relationship is described by [Eq. 5-2](#). [Eq. 5-3](#) defines the transverse shear stress-transverse shear strain relationship.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{Bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \end{Bmatrix} \quad \text{Eq. 5-2}$$

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \quad \text{Eq. 5-3}$$

The format of the MAT2 Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID								

Field	Contents
MID	Material identification number.
Gij	The material property matrix.
RHO	Mass density.
Ai	Thermal expansion coefficient vector.
TREF	Reference temperature.
ST, SC, SS	Stress limits for tension, compression, and shear.
MCSID	Material coordinate system identification number.

When you perform composite analysis using the PCOMP entries, MAT2 entries are generated automatically. The PCOMP entry is discussed in “[The Composite Element \(PCOMP\)](#)” on page 199.

5.4 Axisymmetric Solid Orthotropic Material (MAT3)

The MAT3 entry is used to define an orthotropic three-dimensional relationship in a cross-sectional coordinate system (x, θ, z). You can only use the MAT3 with the axisymmetric CTRIAX6 element. The axisymmetric solid orthotropic material is defined by [Eq. 5-4](#).

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_\theta \\ \epsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{\theta x}}{E_\theta} & \frac{-\nu_{zx}}{E_z} & 0 \\ \frac{-\nu_{x\theta}}{E_x} & \frac{1}{E_\theta} & \frac{-\nu_{z\theta}}{E_z} & 0 \\ \frac{-\nu_{xz}}{E_x} & \frac{-\nu_{\theta z}}{E_\theta} & \frac{1}{E_z} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - T_{REF}) \begin{Bmatrix} A_x \\ A_\theta \\ A_z \\ 0 \end{Bmatrix}$$

Eq. 5-4

To preserve symmetry, the following relationships must hold:

$$\frac{\nu_{x\theta}}{E_x} = \frac{\nu_{\theta x}}{E_\theta}; \frac{\nu_{xz}}{E_x} = \frac{\nu_{zx}}{E_z}; \frac{\nu_{\theta z}}{E_\theta} = \frac{\nu_{z\theta}}{E_z}$$

Eq. 5-5

The format of the MAT3 Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	ETH	EZ	NUXTH	NUTHZ	NUZX	RHO	
			GZX	AX	ATH	AZ	TREF	GE	

Field	Contents
MID	Material identification number.
EX, ETH, EZ	Young’s moduli in the x, θ, and z directions, respectively.
xx	Poisson’s ratios.
RHO	Mass density.
GZX	Shear modulus.
AX, ATH, AZ	Thermal expansion coefficients.
TREF	Reference temperature.

5.5 Two-Dimensional Orthotropic Material (MAT8)

The MAT8 entry is used to define a two-dimensional orthotropic stress-strain relationship as shown in [Eq. 5-6](#) and [Eq. 5-7](#). The MAT8 entry can only be used with the plate and shell elements. [Eq. 5-6](#) defines the in-plane stress-strain relationship. The transverse shear stress-transverse shear strain relationship is defined by [Eq. 5-7](#).

$$\begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & \frac{-\nu_{12}}{E_1} & 0 \\ \frac{-\nu_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} + (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ 0 \end{Bmatrix} \quad \text{Eq. 5-6}$$

$$\begin{Bmatrix} \tau_{1z} \\ \tau_{2z} \end{Bmatrix} = \begin{bmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{bmatrix} \begin{Bmatrix} \gamma_{1z} \\ \gamma_{2z} \end{Bmatrix} \quad \text{Eq. 5-7}$$

The format of the MAT8 Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						

Field	Contents
MID	Material identification number.
E1	Modulus of elasticity in the longitudinal direction.
E2	Modulus of elasticity in the lateral direction.
NU12	Poisson's ratio ((ϵ_2/ϵ_1) for uniaxial loading in 1-direction).
G12	In-plane shear modulus.
G1Z	Transverse shear modulus for shear in the 1-Z plane.
G2Z	Transverse shear modulus for shear in the 2-Z plane.
RHO	Mass density.
Ai	Thermal expansion coefficient in the i-direction.
TREF	Reference temperature.
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction.

Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction.
S	Allowable stress or strain for in-plane shear.
GE	Structural damping coefficient.
F12	Interaction term used in the Tsai-Wu theory.
STRN	Request for maximum strain theory.

The composite tube structure shown in **Figure 5-1** is used to illustrate the modeling of an orthotropic material. The longitudinal direction is the fiber direction and is oriented at an angle of 45 degrees relative to the tube axes as shown. The tube is held fixed at one end, and a uniform axial load is applied at the other end.

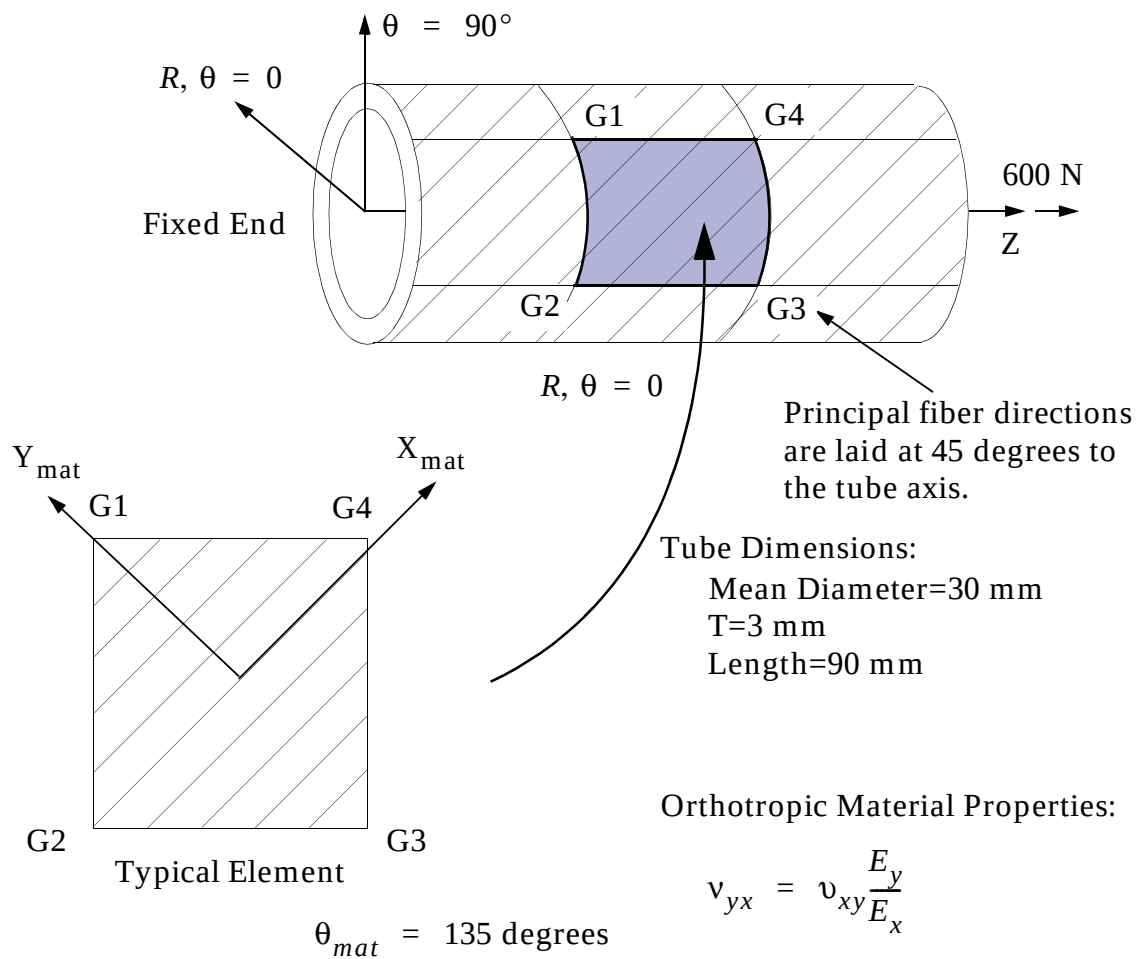


Figure 5-1 Composite Tube with Orthotropic Properties

A sample of the tube material has been tested, and the material properties are determined in the longitudinal (X_{mat}) and transverse (Y_{mat}) directions. The material properties are shown in **Table 5-1**.

Table 5-1 Composite Tube Material Properties

	Longitudinal Direction	Transverse Direction	Shear
E	$25 \cdot 10^4 \frac{N}{mm^2}$	$10 \cdot 10^4 \frac{N}{mm^2}$	--
v	0.4	0.16	--
G ₁₂	--	--	$80,000 \frac{N}{mm^2}$

The input file is shown in [Listing 5-1](#). Normally you would not make a model this coarse for production purposes, but the small number of elements is useful for demonstration purposes. The geometry has been defined with a local cylindrical coordinate system for convenience. This does not affect the orientation of the material coordinate system.

Listing 5-1 Orthotropic Tube Input File

```

$
$ GEOMETRY DEFINED AND OUTPUT IN THE CYL COORD SYS 1
$
CORD2C  1      0      0.0      0.0      0.0      0.0      0.0      1
        1.      0.0      0.0
$
GRID    110     1      15.      150.     30.      1
GRID    111     1      15.      150.     60.      1
GRID    114     1      15.     -150.     30.      1
GRID    115     1      15.     -150.     60.      1
$
CQUAD4  8       1      110      114      115      111      135.
$
PSHELL  1       1       3.       1
$
MAT8    1       250000. 100000. .4      80000.  80000.  80000.
$

```

Note: Only element 8 and its connected grid points are shown.

The finite element model for the tube is shown in [Figure 5-2](#). Element 8 is highlighted to demonstrate how the material coordinate system is used. The material coordinate system is oriented at an angle of 135 degrees (field 8 of the CQUAD4 entry). The 135 degrees is with respect to a line drawn from G1 (which has a grid ID of 110) to G2 (which has a grid ID of 114). The resulting material coordinate system is therefore the same as that shown in [Figure 5-1](#). The finite element model is shown in [Figure 5-2](#). The lower plot in [Figure 5-2](#) is the deformed shape. It is interesting to note that the tube rotates about the z-axis even though a uniform axial load is applied. This rotation is due to the orthotropic material properties and the fiber direction oriented in a different direction than the applied load.

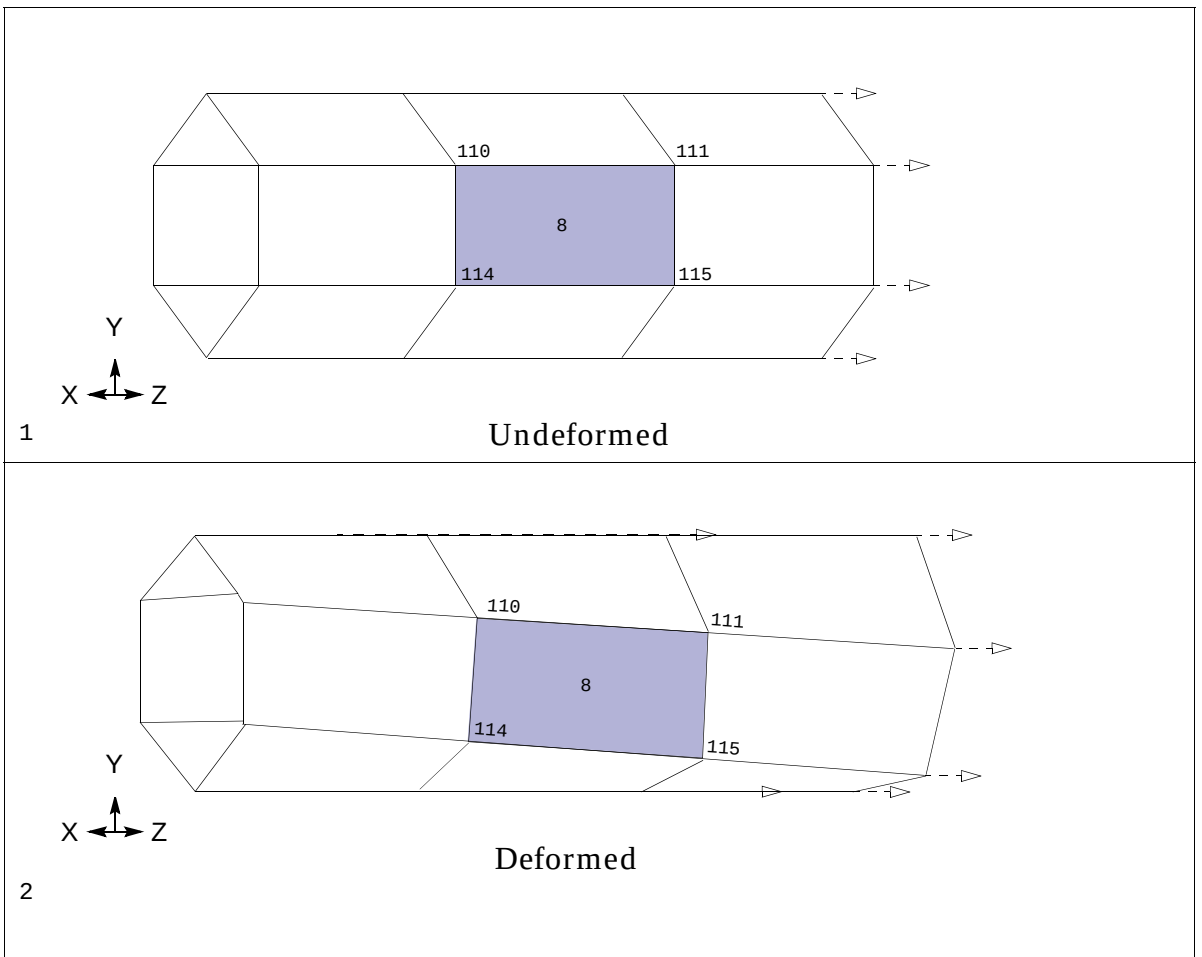


Figure 5-2 Finite Element Model of the Orthotropic Tube

5.6 Three-Dimensional Anisotropic Material (MAT9)

The MAT9 entry is used to define an anisotropic material property for the CHEXA, CPENTA, and CTETRA solid elements. The three-dimensional anisotropic material is defined by [Eq. 5-8](#).

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = [G] \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix} \quad \text{Eq. 5-8}$$

where:

$$[G] = \begin{bmatrix} G_{11} & & & & & \\ G_{12} & G_{22} & & & & \\ G_{13} & G_{23} & G_{33} & & & \\ G_{14} & G_{24} & G_{34} & G_{44} & & \\ G_{15} & G_{25} & G_{35} & G_{45} & G_{55} & \\ G_{16} & G_{26} & G_{36} & G_{46} & G_{56} & G_{66} \end{bmatrix} \quad \text{symmetric}$$

The format of the MAT9 Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

Field	Contents
MID	Material identification number.
Gij	Elements of the 6 · 6 symmetric material property matrix in the material coordinate system.
RHO	Mass density.
Ai	Thermal expansion coefficient.
TREF	Reference temperature.

When you use the MAT9 entry, it is advisable to define a material coordinate system on the PSOLID entry (using field 4). For solid elements, stresses are output in the material coordinate system, which by default, is the basic coordinate system. In general, for solid elements, it is not easy to determine the orientation of the element coordinate system.

If you want to use the MAT9 entry to define an orthotropic material, the terms of the material matrix can be generated by using [Eq. 5-9](#):

$$\begin{aligned}
 G_{11} &= \frac{1 - \nu_{yz}\nu_{zy}}{E_y E_z \Delta} & G_{12} &= \frac{\nu_{yz} + \nu_{zx}\nu_{yz}}{E_y E_z \Delta} & G_{13} &= \frac{\nu_{zx} + \nu_{yx}\nu_{zy}}{E_y E_z \Delta} \\
 G_{22} &= \frac{1 - \nu_{xz}\nu_{zx}}{E_x E_z \Delta} & G_{23} &= \frac{\nu_{zy} + \nu_{xy}\nu_{zx}}{E_x E_z \Delta} \\
 G_{33} &= \frac{1 - \nu_{xy}\nu_{yx}}{E_x E_y \Delta}
 \end{aligned} \tag{Eq. 5-9}$$

where:

ν_{ij} = Poisson's ratios

E_x, E_y, E_z = Young's modulus in the x-, y- and z-directions

G_{xy}, G_{yz}, G_{zx} = shear moduli

$$D = \frac{1 - \nu_{xy}\nu_{yx} - \nu_{yz}\nu_{zy} - \nu_{xz}\nu_{zx} - 2\nu_{yx}\nu_{zy}\nu_{xz}}{E_x E_y E_z}$$

also

$$G_{44} = G_{xy}$$

$$G_{55} = G_{yz}$$

$$G_{66} = G_{zx}$$

and

$$G_{14} = G_{15} = G_{16} = 0.0$$

$$G_{24} = G_{25} = G_{26} = 0.0$$

$$G_{34} = G_{35} = G_{36} = 0.0$$

$$G_{45} = G_{46} = G_{56} = 0.0$$

The following example illustrates the use of the anisotropic material property using the MAT9 material entry. This example consists of three parts as follows:

1. Develop a material property matrix using a pseudo-anisotropic model consisting of a single CHEXA element with some bars along the edges and across the diagonals.
2. Verify that the material matrix developed in part 1 is correct.
3. Use the material matrix in a distorted element.

This three-part approach is analogous to constructing an anisotropic material test coupon and then testing that coupon in the test lab to obtain a strain matrix. From the strain matrix you develop the material matrix. Then you verify the material matrix by modeling the test coupon. Finally, you use the newly developed material matrix in the actual structure.

Part 1 -- Develop the Material Matrix

To simulate a three-dimensional anisotropic material, a single CHEXA element is used with bars of various cross-sectional areas and bending stiffnesses along the edges and across the diagonals as shown in [Figure 5-3\(a\)](#). The cross sections of the bars are not important, but if you are interested in the actual dimensions, refer to the input file “anis1.dat” located on the delivery media. The dimensions of the cube are $1 \cdot 1 \cdot 1$ inches. To constrain the cube, one of the corner grid points is fixed in all six component directions.

Six self-equilibrating load cases are used to represent each of the six stress components. Each direction was applied as a separate subcase. A self-equilibrating normal load consists of applying equal and opposite forces to opposite faces of the cube (i.e., forces F_1 , F_2 , and F_3). For the shear load, a self-equilibrating load consists of applying four forces around the cube in order to place the cube in a state of pure shear. Note that for all self-equilibrating load cases, the net resultant should be zero; hence, the SPC force should also be zero. Because of the dimensions chosen for CHEXA, each load case represents a unit stress, and the resulting strains are the strains due to unit stress.

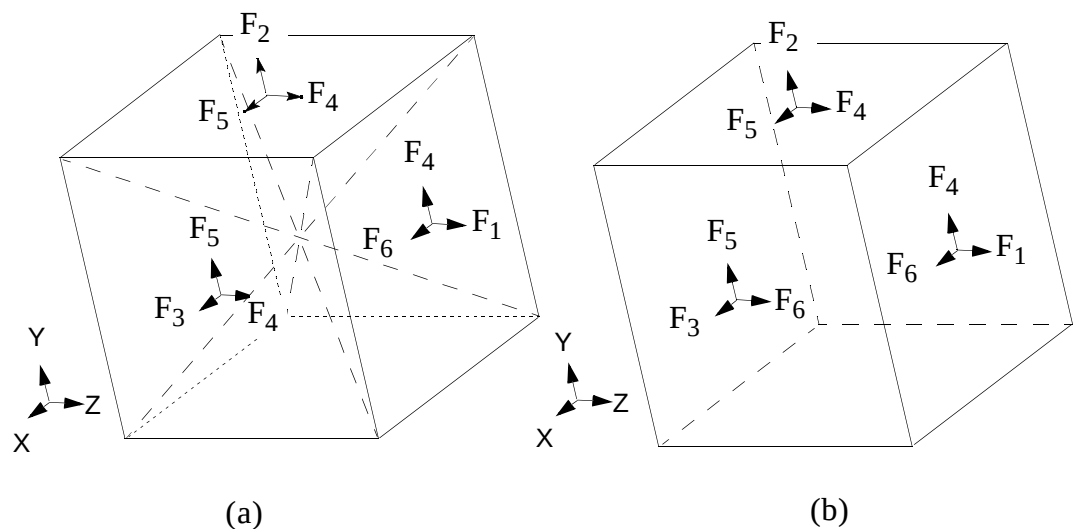


Figure 5-3 Pseudo-Anisotropic Model

The strains at the center of the CHEXA were extracted from the output file and summarized in [Table 5-2](#).

Table 5-2 Strains Due to Unit Loads on Pseudo Anisotropic Model

		Applied Unit Load					
		FX	FY	FZ	FXY	FYZ	FZX
Resulting Strains (10 ⁻⁶)	ϵ_X	0.155	-0.034	-0.056	-0.016	0.002	-0.011
	ϵ_Y	-0.034	0.072	-0.018	-0.005	0.001	-0.003
	ϵ_Z	-0.056	-0.018	0.107	-0.008	0.003	-0.005
	γ_{XY}	-0.016	-0.005	-0.008	0.197	-0.011	0.009
	γ_{YZ}	0.002	0.001	0.003	-0.011	0.219	-0.021
	γ_{ZX}	-0.011	-0.003	-0.005	0.009	-0.021	0.180

[Table 5-2](#) represents a strain matrix that, when inverted, is the material matrix $[G]$ introduced in [Eq. 5-8](#). Note that since the strain matrix is symmetric, the material matrix $[G]$ is also symmetric.

The result is the material matrix $[G]$ for our anisotropic material as shown in [Eq. 5-10](#).

$$[G] = \begin{bmatrix} 10.67 & 6.91 & 6.89 & 1.25 & -0.09 & 0.86 \\ 6.91 & 19.03 & 6.90 & 1.27 & -0.12 & 0.83 \\ 6.89 & 6.90 & 14.28 & 1.23 & -0.15 & 0.82 \\ 1.25 & 1.27 & 1.23 & 5.27 & 0.21 & -0.11 \\ -0.09 & -0.12 & -0.15 & 0.21 & 4.63 & 0.52 \\ 0.86 & 0.83 & 0.82 & -0.11 & 0.52 & 5.72 \end{bmatrix} \cdot 10^6 \quad \text{Eq. 5-10}$$

Part 2 -- Verify the Material Matrix

Now that the material matrix has been generated, the next step is to verify that the matrix is entered on the MAT9 entry correctly. To do this, we return to the 1 · 1 · 1 inch cube, but this time the analysis will not include the CBAR entries as shown in [Figure 5-3\(b\)](#). The same six load cases are applied as in Part 1.

If the material matrix is correct, the strains produced should be numerically the same as those shown in [Table 5-2](#).

The input file for the verification model is shown in [Listing 5-2](#).

Listing 5-2 Anisotropic Verification Input File

```
$
$ FILENAME - ANIS2.DAT
$
ID      LINEAR,ANIS2
SOL      101
TIME     5
CEND
TITLE = ANISOTOPIC TEST ELEMENT
SPC = 1
SET 1 =13
STRAIN = 1
SPCFORCE=ALL
SUBCASE 1
    LOAD = 1
SUBCASE 2
    LOAD = 2
SUBCASE 3
    LOAD = 3
SUBCASE 4
    LOAD = 4
SUBCASE 5
    LOAD = 5
SUBCASE 6
    LOAD = 6
EGIN BULK$
GRID    1          0.0      0.0      0.0
GRID    2          1.       0.0      0.0
GRID    3          0.0      1.       0.0
GRID    4          1.       1.       0.0
GRID    6          0.0      0.0      1.
GRID    8          1.       0.0      1.
GRID   10          0.0      1.       1.
GRID   12          1.       1.       1.
```

Listing 5-2 Anisotropic Verification Input File (continued)

```

CHEXA  13      1      1      6      8      2      3      10      +
+      12      4
$
PLOAD4 1      13      1.          2      12      +
+      1.
PLOAD4 1      13      1.          10      1      +
+      -1.
PLOAD4 2      13      1.          10      4      +
+      0.0      1.
PLOAD4 2      13      1.          6      2      +
+      0.0      -1.
PLOAD4 3      13      1.          6      12      +
+      0.0      0.0      1.
PLOAD4 3      13      1.          1      4      +
+      0.0      0.0      -1.
PLOAD4 4      13      1.          10      4      +
+      1.      0.0
PLOAD4 4      13      1.          6      2      +
+      -1.      0.0
PLOAD4 4      13      1.          12      2      +
+      1.
PLOAD4 4      13      1.          1      10      +
+      -1.
PLOAD4 5      13      1.          10      4      +
+      1.
PLOAD4 5      13      1.          6      2      +
+      -1.
PLOAD4 5      13      1.          6      12      +
+      1.
PLOAD4 5      13      1.          1      4      +
+      -1.
PLOAD4 6      13      1.          6      12      +
+      1.
PLOAD4 6      13      1.          1      4      +
+      -1.
PLOAD4 6      13      1.          2      12      +
+      1.
PLOAD4 6      13      1.          1      10      +
+      -1.
$
PSOLID 1      2      0
$
MAT9    2      10.67+6  6.91+6  6.89+6  1.25+6  -0.09+6  0.86+6  19.03+6
        6.90+6  1.27+6  -0.12+6  0.83+6  14.28+6  1.23+6  -0.15+6  0.82+6
        5.27+6  0.21+6  -0.11+6  4.63+6  0.52+6  5.72+6
SPC1    1      123      6
SPC1    1      12      1
SPC1    1      2      2
ENDDATA

```

You should notice a few items in the verification model input file. The six loads are applied separately, each with their own subcase. The constraints are applied to three separate grid points and are non-redundant. The non-redundant constraint set allows the element to expand in all directions without imposing any constraint forces. However, rigid body modes are constrained, which is a requirement for static analysis. The last item to note is the use of the basic coordinate system for the material definition.

The resulting center strains for the verification model are shown in [Figure 5-4](#). As can be seen, there is good agreement compared to the strain matrix shown in [Table 5-2](#).

S T R A I N S I N H E X A H E D R O N S O L I D E L E M E N T S (H E X A)												
ELEMENT-ID	CORNER GRID-ID	-----CENTER AND CORNER POINT STRAINS-----					DIR. COSINES			MEAN PRESSURE	VON MISES	
		NORMAL		SHEAR		PRINCIPAL	-A-	-B-	-C-			
13												

Note: Only the center strains are shown here to reduce space. The complete output also gives the corner strains.

Figure 5-4 Verification Model Strain Output

Part 3 - Use the Anisotropic Material in the Actual Structure

The last step in the example is to use the material properties to model an actual structure as shown in [Figure 5-5](#). The structure is very coarse since it is used for demonstration purposes only. The model consists of four elements with their material coordinate system oriented 45 degrees with respect to the basic coordinate system.

Note: Y_{mat} is along Y_b and
 X_{mat} and Z_{mat} are
rotated 45 degrees with
respect to X_b and Z_b

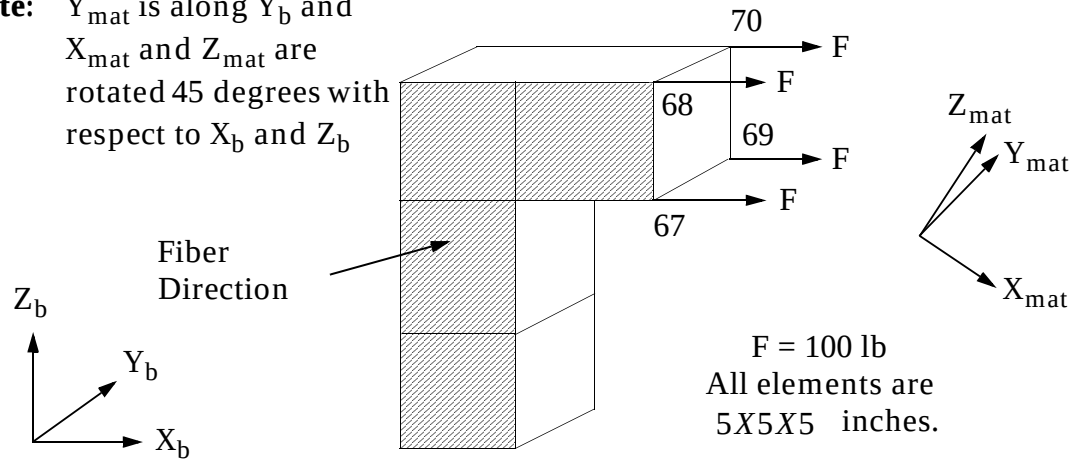


Figure 5-5 Anisotropic Solids Model

The input file is shown in [Listing 5-3](#). Note the local coordinate system used to define the material orientation. The displacement results are shown in [Figure 5-6](#).

Listing 5-3 Structure with Anisotropic Material Input File

```

$
$ FILENAME - ANIS3.DAT
$
ID      LINEAR, ANIS3
SOL     101
TIME    5
CEND
TITLE = USING THE MATERIAL MATRIX
LOAD = 1
SPC = 1
DISP = ALL
BEGIN BULK
PARAM   POST      0
PARAM   AUTOSPC YES
$
$
$ MATERIAL COORDINATE SYSTEM
$
CORD2R  1      0      0.0    0.0    0.0    .70710680.0    .7071068+
        .70710680.0    -.707107
GRID    51      0.0    5.     0.0
GRID    52      5.     5.     0.0
GRID    53      0.0    5.     5.
GRID    54      5.     5.     5.
GRID    55      0.0    0.0    0.0
GRID    56      5.     0.0    0.0
GRID    57      0.0    0.0    5.
GRID    58      5.     0.0    5.
GRID    59      0.0    5.    10.
GRID    60      5.     5.    10.
GRID    61      0.0    0.0    10.
GRID    62      5.     0.0    10.
GRID    63      0.0    5.    15.
GRID    64      5.     5.    15.
GRID    65      0.0    0.0    15.
GRID    66      5.     0.0    15.
GRID    67      10.    5.    10.
GRID    68      10.    5.    15.
GRID    69      10.    0.0    10.
GRID    70      10.    0.0    15.
$
CHEXA   1      1      51      55      56      52      53      57      +
+      58      54
CHEXA   2      1      53      57      58      54      59      61      +
+      62      60
CHEXA   3      1      59      61      62      60      63      65      +
+      66      64
CHEXA   4      1      60      62      69      67      64      66      +
+      70      68
$
$
FORCE   1      69      0      100.    1.     0.0    0.0
FORCE   1      67      0      100.    1.     0.0    0.0
FORCE   1      68      0      100.    1.     0.0    0.0
FORCE   1      70      0      100.    1.     0.0    0.0
$
PSOLID  1      2      1      GRID      SMECH
$
MAT9    2      10.67+6 6.91+6 6.89+6 1.25+6 -0.09+6 0.86+6 19.03+6
        6.90+6 1.27+6 -0.12+6 0.83+6 14.28+6 1.23+6 -0.15+6 0.82+6
        5.27+6 0.21+6 -0.11+6 4.63+6 0.52+6 5.72+6
$
SPC     1      51      123456 0.0
SPC     1      52      123456 0.0
SPC     1      56      123456 0.0
SPC     1      55      123456 0.0
$
ENDDATA

```

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
51	G	0.0	0.0	0.0	0.0	0.0	0.0
52	G	0.0	0.0	0.0	0.0	0.0	0.0
53	G	1.232738E-04	3.251065E-07	8.268995E-05	0.0	0.0	0.0
54	G	1.204173E-04	1.783952E-06	-8.797066E-05	0.0	0.0	0.0
55	G	0.0	0.0	0.0	0.0	0.0	0.0
56	G	0.0	0.0	0.0	0.0	0.0	0.0
57	G	1.002035E-04	2.286116E-05	7.448530E-05	0.0	0.0	0.0
58	G	9.797833E-05	-2.485534E-05	-7.772400E-05	0.0	0.0	0.0
59	G	3.605538E-04	1.530884E-05	1.199663E-04	0.0	0.0	0.0
60	G	3.610890E-04	-1.436989E-05	-1.294236E-04	0.0	0.0	0.0
61	G	3.284150E-04	1.772471E-05	1.189002E-04	0.0	0.0	0.0
62	G	3.284143E-04	-1.659684E-05	-1.265683E-04	0.0	0.0	0.0
63	G	6.271876E-04	1.717129E-05	1.256454E-04	0.0	0.0	0.0
64	G	6.300415E-04	-1.526032E-05	-1.351714E-04	0.0	0.0	0.0
65	G	5.944516E-04	1.910164E-05	1.262980E-04	0.0	0.0	0.0
66	G	5.975869E-04	-1.427919E-05	-1.336644E-04	0.0	0.0	0.0
67	G	3.671078E-04	-5.154453E-05	-4.031157E-04	0.0	0.0	0.0
68	G	6.358424E-04	-4.903843E-05	-4.012304E-04	0.0	0.0	0.0
69	G	3.339844E-04	-4.717636E-05	-4.023441E-04	0.0	0.0	0.0
70	G	6.039479E-04	-4.626662E-05	-3.997554E-04	0.0	0.0	0.0

Figure 5-6 Output for Structure with Anisotropic Material

5.7 The PSHELL Entry

The PSHELL entry was introduced in “[MSC.Nastran Elements](#)” on page 65 with the discussion of the elements. The discussion of the material fields was limited to modeling an isotropic plate. For isotropic materials, use the same MAT1 identification for MID1 and MID2, and leave the MID3 and MID4 fields blank. The purpose of this section is to describe the use of the material fields if your plate structure is not isotropic.

The PSHELL entry is used to define the properties of the plate elements. With the PSHELL element, you have several options available, providing you with a versatile tool for modeling a wide variety of applications. With the appropriate selection of plate thickness, material properties, and bending and transverse shear parameters, you can model practically any plate you can imagine.

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

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

Field	Contents
PID	Property identification number.
MIDi	Material identification numbers.
T	Default membrane thickness for Ti on the connection entry.
$12I/T^3$	Bending stiffness parameter.
TS/T	Transverse shear thickness divided by the membrane thickness.
NSM	Nonstructural mass per unit area.
Z1, Z2	Fiber distances for stress calculations.

On the PSHELL entry, there are four material ID fields available as shown in [Table 5-3](#).

Table 5-3 Material Properties on the PSHELL Entry

ID	PSHELL Field	Purpose
MID1	3	Membrane material property.
MID2	5	Bending material property.
MID3	7	Transverse shear stiffness material property.
MID4	4 (cont.)	Material property used to define coupling between the membrane and the bending deformation.

If you wish to include membranes only, use MID1 only. If you wish to include bending only, use MID2 and optionally MID3. When MID3 is used, thick plate theory (which includes the transverse shear flexibility) is used to develop the element stiffness matrix. In general, for thin or curved surfaces, it is advisable not to use MID3.

The material type used for any of the material fields (MID1, MID2, MID3, or MID4) may be MAT1, MAT2, or MAT8. The PCOMP entry, which is the subject of the next section, provides a convenient way of entering appropriate material properties for a plate consisting of layers of unidirectional fibers.

The MID4 field on the PSHELL entry is used to input a coupling relationship between the in-plane forces and bending moments. This coupling only occurs in plates that are nonsymmetric about the neutral plane or where the neutral plane is offset from the grid points. Typical applications include reinforced skins and aluminum bonded to fiberglass. The MID4 field should be left blank if the element cross section is symmetric.

5.8 The Composite Element (PCOMP)

MSC.Nastran provides a property definition specifically for performing composite analysis. You specify the material properties and orientation for each of the layers and MSC.Nastran produces the equivalent PSHELL and MAT2 entries. Additional stress and strain output is generated for each layer and between the layers.

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

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA 1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA 3	SOUT3	etc.				

Field	Contents
PID	Property identification number.
Z0	Distance from the reference plane to the bottom surface.
NSM	Nonstructural mass per unit area.
SB	Allowable shear stress of the bonding material.
FT	Failure theory.
TREF	Reference temperature.
LAM	“Blank”, “SYM”, “MEM”, “BEND” option.
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer.
Ti	Thicknesses of the various plies.
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element.
SOUTi	Stress or strain output request.

A two-dimensional composite material is defined as a stacked group of laminae arranged to form a flat or curved plate or shell. Each lamina may be considered as a group of unidirectional fibers. The principal material axes for the lamina are parallel and perpendicular to the fiber directions. The principal directions are referred to as “longitudinal” or the 1-direction of the fiber and as “transverse” or the 2-direction for the perpendicular direction (matrix direction).

A laminate is a stack of these individual lamina arranged with the principal directions of the each lamina oriented in a particular direction as shown in [Figure 5-7](#).

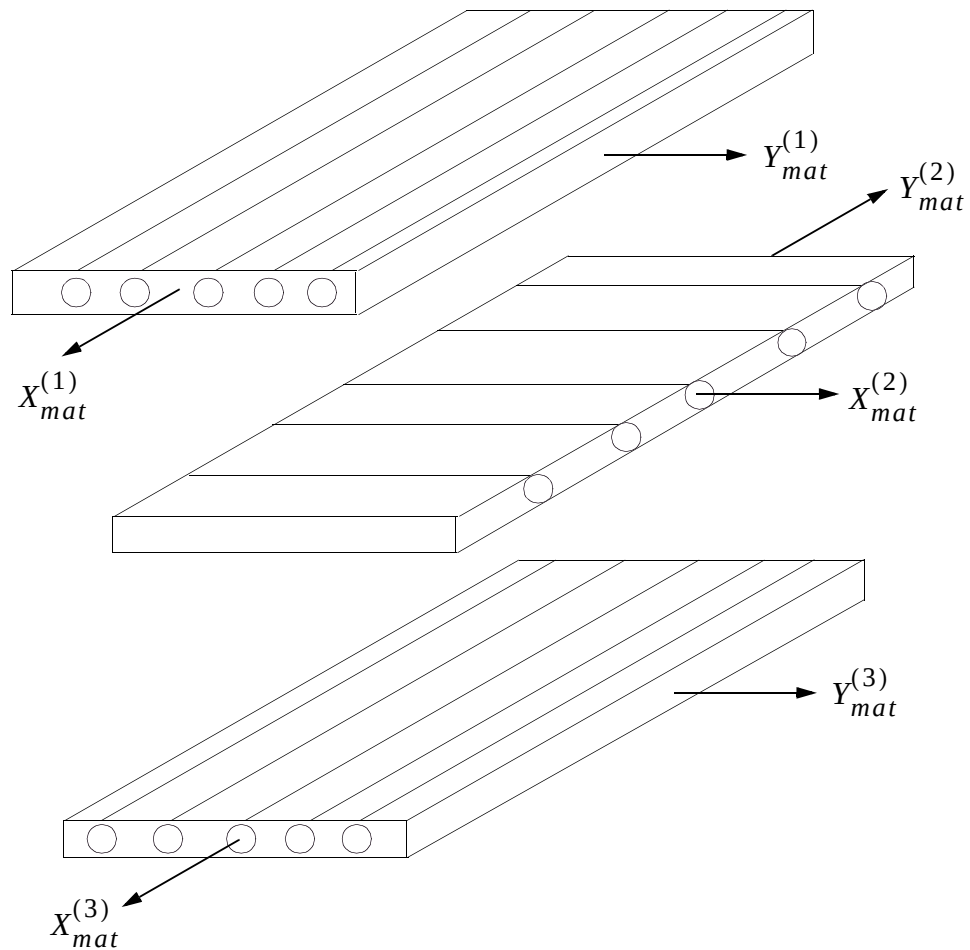


Figure 5-7 Laminae Arranged to Form a Laminate

The laminae are bonded together with a thin layer of bonding material that is considered to be of zero thickness. Each lamina can be modeled as an isotropic material (MAT1), two-dimensional anisotropic material (MAT2), or orthotropic material (MAT8). The assumptions inherent in the lamination theory are as follows:

- Each lamina is in a state of plane stress.
- The bonding is perfect.
- Two-dimensional plate theory can be used.

Using the material properties for each of the lamina, equivalent PSHELL and MAT2 entries are generated as shown in [Figure 5-8](#).

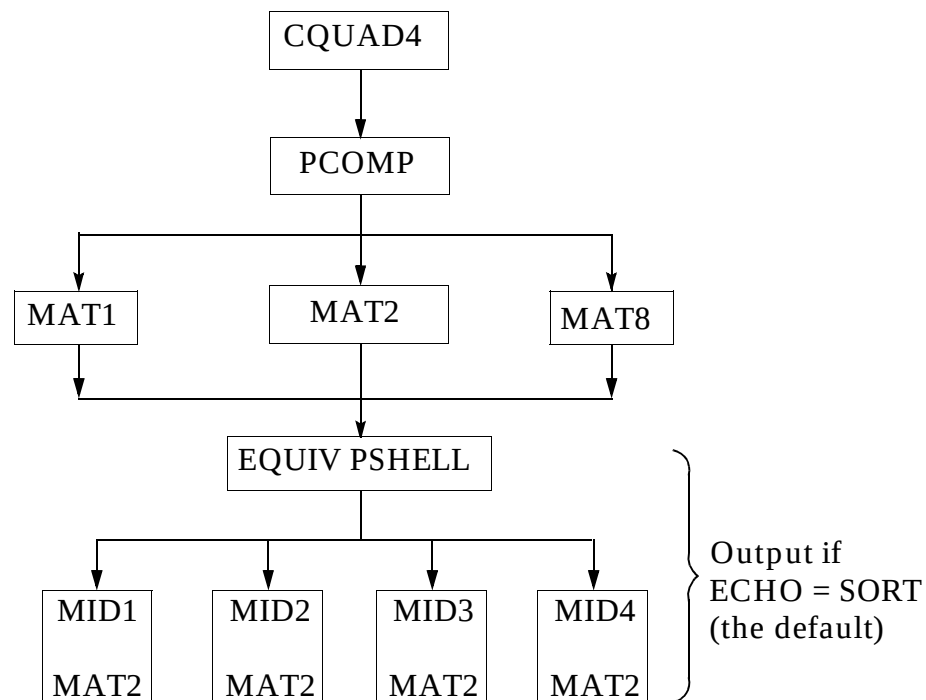


Figure 5-8 Equivalent PSHELL and MAT2 Entries Are Generated

The output you may request for a composite analysis includes:

- Stresses and strains for the equivalent plate.
- Force resultants.
- Stresses and /or strains in the individual laminate including approximate interlaminar shear stresses in the bonding material output.
- A failure index table.

If you want stress and /or the failure indices for the composite elements, ELSTRESS must be requested in the Case Control Section for the appropriate elements. Also, if you want the failure index table, you must enter the stress limits for each lamina on the appropriate material entry, the shear stress limit S_b , and the failure theory method FT on the PCOMP entries.

As shown in [Figure 5-8](#), each PCOMP (together with the material entries for each lamina) is processed to form an equivalent PSHELL and four MAT2 entries. The equivalent PSHELL and MAT2 entries are printed in the output. If you wish to use them for future runs, they may be punched to an ASCII file. The ID numbers of the MAT2s are important. The MID1 material has an ID in the range of 100000000 to 199999999. The MID2 material has an ID in the range of 200000000 to 299999999, etc. These ranges are used to inform MSC.Nastran that the material is part of a composite analysis. If you are using the equivalent properties in a future analysis instead of using the PCOMP entry and you are entering a thermal coefficient of expansion, do not change the ID numbers. Also, if you use the equivalent PSHELL and MAT2s, you will not be able to obtain the laminae stress or the failure index table. For the failure index, you have a choice of four failure theories: Hill's theory, Hoffman's theory, Tsai-Wu's theory, and the maximum strain theory.

As an example of the PCOMP entry, consider the cantilevered honeycomb plate shown in [Figure 5-9](#). Although the honeycomb structure is not considered a composite layup, it can be analyzed effectively using the PCOMP entry.

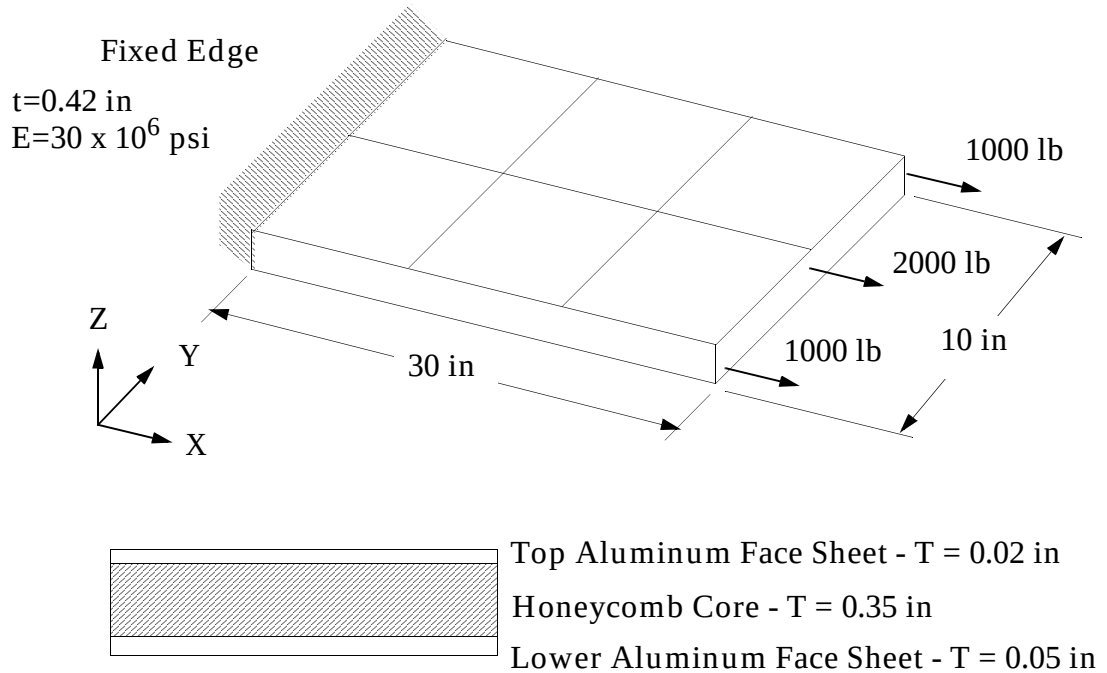


Figure 5-9 Honeycomb Cantilever Plate

The material properties of the honeycomb section are given in [Table 5-4](#). Note that the face sheets are not of the same thickness. The result is a nonsymmetric plate, see [Figure 5-10](#), which can be noted by the MAT2 entry for the MID4 field of the equivalent PSHELL.

Table 5-4 Honeycomb Material Properties

Material	Modulus of Elasticity (10^6 psi)	Tensile Limit (10^3 psi)	Compression Limit (10^3 psi)	Shear Limit (10^3 psi)
Aluminum Face Sheets	10.0	35	35	23
Core	0.0001	0.05	0.3	0.2
Bonding Material	--	--	--	0.1

Part of the input file for the honeycomb plate is shown in [Listing 5-4](#). The load on the plate is a 4000-pound uniform load acting in the plane of the plate. The output requests include the element force, laminae stress, laminae strain, and failure index table.

A selected portion of the output is shown in [Figure 5-10](#).

Listing 5-4 Honeycomb Plate

```

$
$  FILENAME - PCOMP1.DAT
$
CQUAD4  1      100    1      2      6      5
CQUAD4  2      100    2      3      7      6
CQUAD4  3      100    3      4      8      7
CQUAD4  4      100    5      6     10      9
CQUAD4  5      100    6      7     11     10
CQUAD4  6      100    7      8     12     11
$
FORCE   1      12      0      1.     1000.0  0.0    0.0
FORCE   1      4       0      1.     1000.0  0.0    0.0
FORCE   1      8       0      1.     2000.0  0.0    0.0
$
SPC1    1      123456  1      5      9
$
PCOMP   100
        120    0.05    0.     YES    130    0.35    0.0    YES
        120    0.02    0.     YES
$
MAT1    120    10.+6      0.3
        35.E3    35.E3    23.E3
MAT1    130    100.      0.3
        50.     300.     200.

```

```

*** USER INFORMATION MESSAGE 4379, THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING PSHELL AND MAT2 CARDS.
PSHELL      100      100000100      4.2000E-01      200000100      1.0000E+00      300000100      1.0000E+00      0.0000E+00
      -2.1000E-01      2.1000E-01      400000100
MAT2      100000100      1.8316E+06      5.4948E+05      0.0000E+00      1.8316E+06      0.0000E+00      6.4106E+05      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      200000100      4.4895E+06      1.3469E+06      0.0000E+00      4.4895E+06      0.0000E+00      1.5713E+06      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      300000100      3.9189E+01      0.0000E+00      0.0000E+00      3.9189E+01      0.0000E+00      0.0000E+00      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      400000100      3.2705E+05      9.8115E+04      0.0000E+00      3.2705E+05      0.0000E+00      1.1447E+05      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0

DISPLACEMENT VECTOR

POINT ID. TYPE T1 T2 T3 R1 R2 R3
      8 G 1.004209E-02 2.155610E-17 -3.155122E-01 -3.609077E-17 2.089842E-02 .0
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 9

FORCES IN QUADRILATERAL ELEMENTS (QUAD4)

ELEMENT - MEMBRANE FORCES - - BENDING MOMENTS - - TRANSVERSE SHEAR FORCES -
ID FX FY FXY MX MY MXY QX QY
      4 2.000000E+02 2.784165E+01 -9.330303E+00 -4.593548E-15 -3.536165E-03 4.504795E-03 -7.892992E-17 -9.747001E-04
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 10

HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 11

STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRESSES IN FIBRE AND MATRIX DIRECTIONS INTER-LAMINAR STRESSES PRINCIPAL STRESSES (ZERO SHEAR) MAX
ID ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
      4 1 2.08187E+03 2.89631E+02 -9.68898E+01 -2.03948E-16 -2.51853E-03 -3.09 2.08710E+03 2.84409E+02 9.01344E+02
      4 2 3.49114E-02 4.86145E-03 -1.63057E-03 -2.03936E-16 -2.51839E-03 -3.10 3.49996E-02 4.77323E-03 1.51132E-02
      4 3 4.79470E+03 6.67919E+02 -2.24262E+02 -2.82614E-24 -3.48998E-11 -3.10 4.80685E+03 6.55768E+02 2.07554E+03
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 12

FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT FAILURE PLY FP=FAILURE INDEX FOR PLY FB=FAILURE INDEX FOR BONDING FAILURE INDEX FOR ELEMENT FLAG
ID THEORY ID (DIRECT STRESSES/STRAINS) (INTER-LAMINAR STRESSES) MAX OF FP,FB FOR ALL PLIES
      4 STRAIN 1 .0570 -1 .0000
      2 .0007 -1 .0000
      3 .1313 -1 .1313
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 13

HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 1996 MSC/NASTRAN 6/28/96 PAGE 14

STRAINS IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRAINS IN FIBRE AND MATRIX DIRECTIONS INTER-LAMINAR STRAINS PRINCIPAL STRAINS (ZERO SHEAR) MAX
ID ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
      4 1 1.99499E-04 -3.34931E-05 -2.51914E-05 -5.30264E-23 -6.54819E-10 -3.09 2.00177E-04 -3.41721E-05 2.34350E-04
      4 2 3.34529E-04 -5.61196E-05 -4.23949E-05 -5.30233E-18 -6.54781E-05 -3.10 3.35676E-04 -5.72665E-05 3.92943E-04
      4 3 4.59433E-04 -7.70492E-05 -5.83081E-05 -7.34797E-31 -9.07396E-18 -3.10 4.61012E-04 -7.86288E-05 5.39641E-04

```

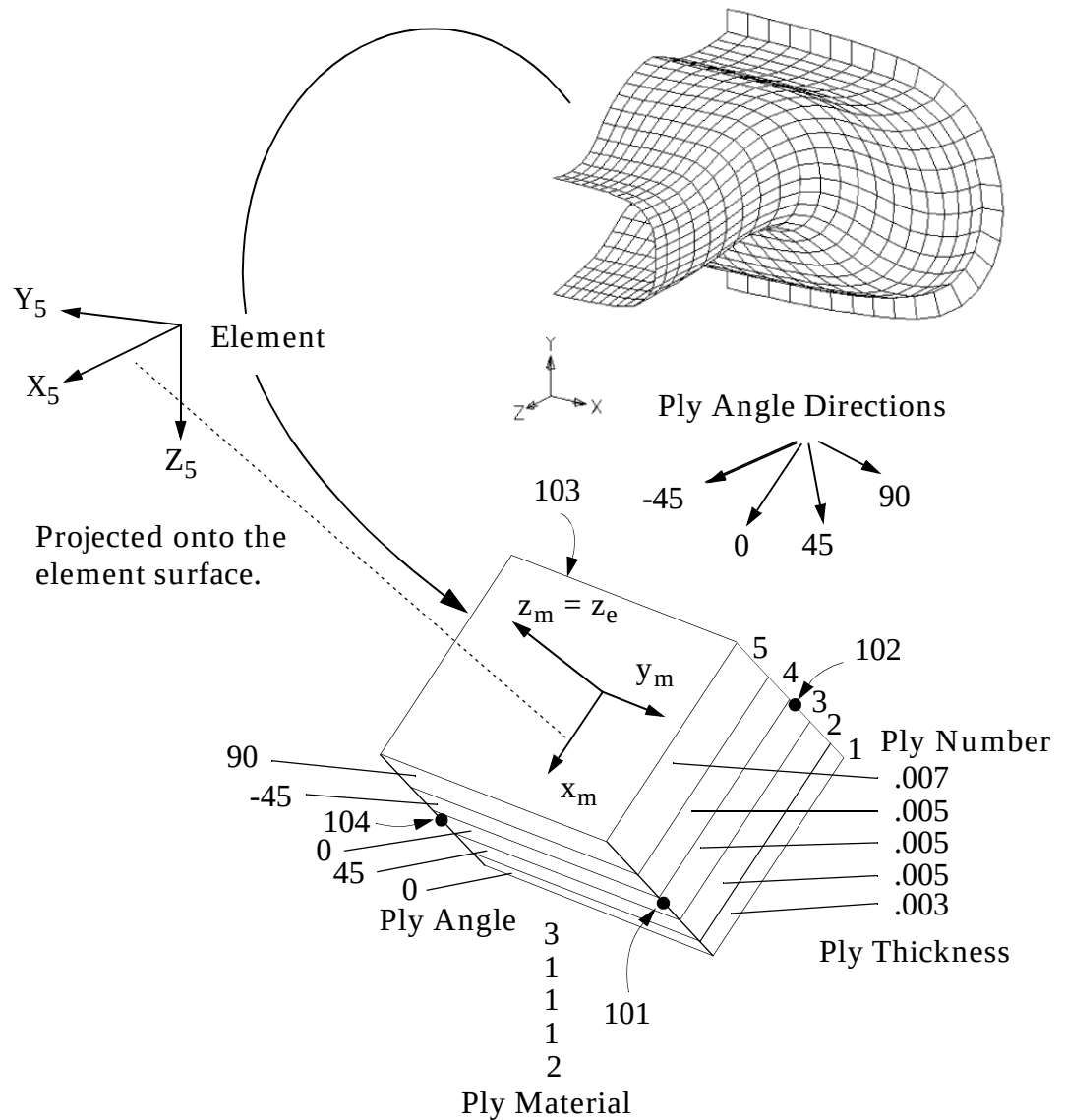
Figure 5-10 Composite Output of the Honeycomb Plate

The plate results indicate that the plates are bending downward due to the axial load, which is expected from a nonsymmetric cross section. The laminae stresses show the face sheets to be highly stressed and the core to have virtually no stress. The failure index table

shows the failure for each laminae and the bonding material to be less than 1, indicating that the structure will not fail. As a word of caution, since each lamina is assumed to be in a state of plane stress, the failure index does not take into account the crushing of the core.

As a final example showing the use of the PCOMP entry, consider the model shown in **Figure 5-11**.

Figure 5-11 shows an example of the data entries required to define a five layer composite material. A typical element in the model is magnified to illustrate the individual layers. The grid points 101, 102, 103, and 104 are used for illustrative purposes. Also included in the figure are the Bulk Data entries needed to model the composite material.



```

CQUAD4, 101, 1, 101, 102, 103, 104, 5
PCOMP, 1, , , 10000.0, STRN,
+, 2, .003, 0.0, YES, 1, .005, 45.0, YES,
+, 1, .005, 0.0, YES, 1, .005, -45.0, YES,
+, 3, .007, 90.0, YES
MAT8, 1, 2.0E+7, 2.0E+6, 0.35, 1.0E+6, 1.0E+6, 1.0E+6, 0.0
+, 0.0, 0.0, 0.0, .007, .006, .007, .006, 15000.,
+, , , 1.0
MAT8, 2, 1.0E+7, 1.0E+7, 0.05, 1.0E+6, 1.0E+6, 1.5E+6, 0.0
+, 0.0, 0.0, 0.0, .007, .006, .007, .006, 15000.,
+, , , 1.0
MAT8, 3, 0.8E+7, 0.8E+7, 0.05, 0.7E+6, 0.7E+6, 1.3E+6, 0.0
+, 0.0, 0.0, 0.0, .006, .005, .006, .005, 12000.,
+, , , 1.0
CORD2R, 5, , 0., 0., 0., 0., -1., 0.,
+, 0., 0., 1.

```

Figure 5-11 Typical Composite Layup

In the example above, the direction of the x-axis of the material coordinate system (x_m) is defined by the projection of the x-axis of coordinate system 5 onto the element. Using this technique, the direction of material coordinate system is independent of the shape of

the element. The material direction of all the elements in the model can be defined by referencing a single coordinate system. The MCID field on the CQUAD4 Bulk Data entry defines the ID of this coordinate system. The coordinate system is shown in the figure as the x_5 -, y_5 -, z_5 -axis and is defined on the CORD2R Bulk Data entry. The x-axis of the material coordinate system can also be defined with an angle. If the THETA or MCID field on the CQUAD4 Bulk Data entry is an integer, then the value is taken as a coordinate system. If it is real, then the value is taken as an angle. The presence of a decimal point makes the value real.

The z-axis (z_m) of the material coordinate system is in the direction of the z-axis of the element coordinate system (z_e). The direction of the y-axis (y_m) follows the right-hand rule.

The CQUAD4 entry references the PCOMP entry, which defines the following layup:

Ply	Material	Thickness	Angle
1	2	0.003	0.0
2	1	0.005	45.0
3	1	0.005	0.0
4	1	0.005	-45.0
5	3	0.007	90.0

The composite material is made of 5 plies. Ply number 1 is at the most negative z_m . Note that since z_m is in the direction of z_e , the element grid point connectivity order determines which is the top and bottom surface of the composite and thus, the stacking direction. Ply angle 0 is in the direction of x_m . The positive sense of the ply angle is in the direction from x_m to y_m . The PCOMP entry references the MAT8 data entries by the MID field.

The example uses three different materials. Note that strain allowables are specified on the MAT8 entry. In order to let MSC.Nastran know that these values are strain allowable, the STRN field must be set equal to 1.0. Material 1 is typical of a graphite epoxy tape. A tape has continuous fibers that all point in the same direction. Note that that E1 is ten times the value of E2. Material 2 is also a tape. Material 3 is a cloth since E1 equals E2. A cloth made of woven fibers has approximately the same number of fibers going in the 1 direction as in the 2 direction.

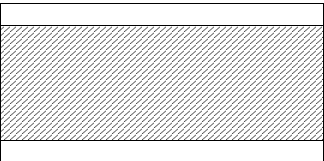
Directions 1 and 2 refer to the ply coordinate system. The ply material properties on the MAT8 entry are in the ply coordinate system. The 1-axis of the ply coordinate system is in the direction of the ply angle. The 3-axis is in the direction of z_m (or z_e). The 2-axis follows the right-hand rule. Ply results are output in the ply coordinate system.

5.9 Equivalent Composite Honeycomb Section

If you are only interested in the overall behavior of the honeycomb composite section, then an equivalent PSHELL / MAT2 or PSHELL / MAT1 combination can be used to model the PCOMP / MAT8 effect. This representation yields the correct equivalent stiffness and, consequently, the correct displacements. However, the stresses in general are meaningless since the information regarding the individual layers are not available to MSC.Nastran when using the PSHELL / MATi combination. If stresses are desired, use the PCOMP / MAT8 combination. This process is illustrated by the following example.

The goal of this example is to model the honeycomb section shown in [Figure 5-12](#) using PSHELL / MAT1. The assumptions are that the membrane and bending loads are carried by the facesheets and the transverse shear loads are carried by the honeycomb core. The cross section consists of two 0.01 in thick aluminum facesheets sandwiching a 0.75 in thick honeycomb core. The properties of the section are shown in [Figure 5-12](#).

Aluminum Facesheets	Honeycomb Core
E1=E2=1.E7	E1=E2=1000.
G12=G13=G23=3.85E6	G12=100., G13=1.E5, G23=5.E4



Aluminum Facesheet -- T=0.01"

Aluminum Honeycomb Core -- T=.75"

Aluminum Facesheet -- T=0.01"

Figure 5-12 Honeycomb Cantilever Plate Represented by PSHELL/MAT1

The calculation of the equivalent properties is shown below:

$$I = 2y^2A = 2(.38)^2(.01) = 2.888 \cdot 10^{-3}$$

$$T = 0.02$$

$$\frac{12I}{T^3} = \frac{12(2.88 \cdot 10^{-3})}{(0.02)^3} = 4332$$

$$T_s \approx .75 + .02 = .77$$

$$\frac{T_s}{T} = \frac{.77}{.02} = 38.5$$

$$G_{core} \approx \frac{1 \cdot 10^5 + 5 \cdot 10^4}{2} = 7.5 \cdot 10^4$$

The transverse shear moduli (G13 and G23) are different in the two transverse directions, which is typical of honeycomb core. Since the MAT1 entry only allows a single shear modulus, an average G_{core} is used. A more accurate representation of the G13 and G23 is to use the MAT2 entry instead of the MAT1 entry. **Listing 5-5** and **Listing 5-6** are the input files using the PCOMP / MAT8 and PSHELL / MAT1 entries, respectively. As shown in the corresponding abridged output files in **Figure 5-13** and **Figure 5-14**, the displacements calculated by using the PSHELL / MAT1 combination are comparable with the displacements obtained using the PCOMP / MAT8 combination.

Listing 5-5 Honeycomb Sandwich Using PCOMP

```

$
$  filename -  pcomp2.dat
$
ID      LINEAR,PCOMP2
SOL      101
TIME     5
CEND
TITLE = HONEYCOMB COMPOSITE CANTLEVER PLATE
SUBTITLE = SYMMETRIC - .01/.75/.01
$
SPC = 1
DISP = ALL
$
SUBCASE 1
LABEL = AXIAL LOAD
LOAD = 1
$
SUBCASE 2
LABEL = VERTICAL SHEAR LOAD
LOAD = 2
$
SUBCASE 3
LABEL = VERTICAL TWIST LOAD
LOAD = 3
$
BEGIN BULK
$
PARAM    POST    -1
$
GRDSET
GRID      1          0.0      -10.      0.0
GRID      2          10.      -10.      0.0
GRID      3          20.      -10.      0.0
GRID      4          30.      -10.      0.0
GRID      5          0.0       0.0       0.0
GRID      6          10.       0.0       0.0
GRID      7          20.       0.0       0.0
GRID      8          30.       0.0       0.0
GRID      9          0.0       10.       0.0
GRID     10          10.       10.       0.0
GRID     11          20.       10.       0.0
GRID     12          30.       10.       0.0
$
CQUAD4    1          100       1         2         6         5
CQUAD4    2          100       2         3         7         6
CQUAD4    3          100       3         4         8         7
CQUAD4    4          100       5         6        10         9
CQUAD4    5          100       6         7        11        10
CQUAD4    6          100       7         8        12        11
$
FORCE     1          12         0         1.        1000.0    0.0      0.0
FORCE     1           4         0         1.        1000.0    0.0      0.0
FORCE     1           8         0         1.        2000.0    0.0      0.0
$
FORCE     2          12         0         1.         0.         0.         1.
FORCE     2           4         0         1.         0.         0.         1.
FORCE     2           8         0         1.         0.         0.         2.
$

```


Listing 5-6 Honeycomb Sandwich Using PSHELL/MAT1

```
$
$  pshell2.dat
$
ID      LINEAR, PSHELL2
SOL      101
TIME      5
CEND
TITLE = EQUIVALENT HONEYCOMB COMPOSITE CANTLEVER PLATE
SUBTITLE = SYMMETRIC - .01/.75/.01
$
SPC = 1
DISP = ALL
$
SUBCASE 1
LABEL = AXIAL LOAD
LOAD = 1
$
SUBCASE 2
LABEL = VERTICAL SHEAR LOAD
LOAD = 2
$
SUBCASE 3
LABEL = TWIST LOAD
LOAD = 3
$
BEGIN BULK
PARAM    POST      -1
$
GRDSET
GRID      1          0.0      -10.      0.0
GRID      2          10.      -10.      0.0
GRID      3          20.      -10.      0.0
GRID      4          30.      -10.      0.0
GRID      5          0.0       0.0       0.0
GRID      6          10.       0.0       0.0
GRID      7          20.       0.0       0.0
GRID      8          30.       0.0       0.0
```

Listing 5-6 Honeycomb Sandwich Using PSHELL/MAT1 (continued)

```

GRID    9          0.0    10.    0.0
GRID   10          10.    10.    0.0
GRID   11          20.    10.    0.0
GRID   12          30.    10.    0.0
$
CQUAD4  1         100     1      2      6      5
CQUAD4  2         100     2      3      7      6
CQUAD4  3         100     3      4      8      7
CQUAD4  4         100     5      6     10      9
CQUAD4  5         100     6      7     11     10
CQUAD4  6         100     7      8     12     11
$
FORCE   1         12      0      1.    1000.0  0.0    0.0
FORCE   1          4      0      1.    1000.0  0.0    0.0
FORCE   1          8      0      1.    2000.0  0.0    0.0
FORCE   2         12      0      1.      0.      0.      1.
FORCE   2          4      0      1.      0.      0.      1.
FORCE   2          8      0      1.      0.      0.      2.
FORCE   3         12      0      1.      0.      0.      1.
FORCE   3          4      0      1.      0.      0.     -1.
$
SPC1     1         123456  1      5      9
PSHELL100  100    .02  100  4332. 200   38.5      +PSLL
+PSLL  -.385  .385
MAT1   100    10.+6      0.3
MAT1   200   1000.    .75+5
$
ENDDATA

```

1	HONEYCOMB COMPOSITE CANTILEVER PLATE				JANUARY 13, 1997	MSC/NASTRAN	12/10/96	PAGE	10
	SYMMETRIC - .01/.75/.01								
0	AXIAL LOAD							SUBCASE 1	
	D I S P L A C E M E N T V E C T O R								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
	1	G	.0	.0	.0	.0	.0	.0	
	2	G	9.938049E-03	3.192859E-03	.0	.0	.0	.0	
	3	G	1.951274E-02	3.057678E-03	.0	.0	.0	.0	
	4	G	2.948488E-02	2.936741E-03	.0	.0	.0	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	9.154656E-03	-1.381969E-17	.0	.0	.0	.0	
	7	G	1.958685E-02	-4.734998E-17	.0	.0	.0	.0	
	8	G	2.954503E-02	-8.476228E-17	.0	.0	.0	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	9.938049E-03	-3.192859E-03	.0	.0	.0	.0	
	11	G	1.951274E-02	-3.057678E-03	.0	.0	.0	.0	
	12	G	2.948488E-02	-2.936741E-03	.0	.0	.0	.0	
0	VERTICAL SHEAR LOAD							SUBCASE 2	
	D I S P L A C E M E N T V E C T O R								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
	1	G	.0	.0	.0	.0	.0	.0	
	2	G	.0	.0	8.081975E-03	2.360743E-04	-1.524400E-03	.0	
	3	G	.0	.0	2.971653E-02	2.389541E-04	-2.685955E-03	.0	
	4	G	.0	.0	5.863158E-02	1.268257E-04	-2.985823E-03	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	.0	.0	9.109288E-03	-7.489637E-18	-1.693645E-03	.0	
	7	G	.0	.0	3.090988E-02	-1.128154E-17	-2.562822E-03	.0	
	8	G	.0	.0	5.927901E-02	-1.922195E-17	-3.002027E-03	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	.0	.0	8.081975E-03	-2.360743E-04	-1.524400E-03	.0	
	11	G	.0	.0	2.971653E-02	-2.389541E-04	-2.685955E-03	.0	
	12	G	.0	.0	5.863158E-02	-1.268257E-04	-2.985823E-03	.0	
0	VERTICAL TWIST LOAD							SUBCASE 3	
	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	
	2	G	.0	.0	-1.255258E-03	1.145127E-04	2.226868E-04	.0	
	3	G	.0	.0	-3.594521E-03	3.597052E-04	2.259500E-04	.0	
	4	G	.0	.0	-5.950768E-03	5.769896E-04	2.259347E-04	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	.0	.0	-1.634409E-17	1.417446E-04	2.934695E-18	.0	
	7	G	.0	.0	-5.219971E-17	3.523428E-04	4.029600E-18	.0	
	8	G	.0	.0	-9.308500E-17	5.806674E-04	4.117684E-18	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	.0	.0	1.255258E-03	1.145127E-04	-2.226868E-04	.0	
	11	G	.0	.0	3.594521E-03	3.597052E-04	-2.259500E-04	.0	
	12	G	.0	.0	5.950768E-03	5.769896E-04	-2.259347E-04	.0	

Figure 5-13 Displacement Outputs for Honeycomb Sandwich Using PCOMP

1	EQUIVALENT HONEYCOMB COMPOSITE CANTLEVER PLATE				JANUARY 13, 1997	MSC/NASTRAN	12/10/96	PAGE	9
	SYMMETRIC - .01/.75/.01								
0	AXIAL LOAD							SUBCASE 1	
	D I S P L A C E M E N T V E C T O R								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
	1	G	.0	.0	.0	.0	.0	.0	
	2	G	9.977354E-03	3.215835E-03	.0	.0	.0	.0	
	3	G	1.958953E-02	3.081032E-03	.0	.0	.0	.0	
	4	G	2.960244E-02	2.958689E-03	.0	.0	.0	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	9.187398E-03	1.784324E-18	.0	.0	.0	.0	
	7	G	1.966428E-02	9.149946E-18	.0	.0	.0	.0	
	8	G	2.966328E-02	2.280977E-17	.0	.0	.0	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	9.977354E-03	-3.215835E-03	.0	.0	.0	.0	
	11	G	1.958953E-02	-3.081032E-03	.0	.0	.0	.0	
	12	G	2.960244E-02	-2.958689E-03	.0	.0	.0	.0	
0	VERTICAL SHEAR LOAD							SUBCASE 2	
	D I S P L A C E M E N T V E C T O R								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
	1	G	.0	.0	.0	.0	.0	.0	
	2	G	.0	.0	8.084896E-03	2.362314E-04	-1.527135E-03	.0	
	3	G	.0	.0	2.977032E-02	2.404278E-04	-2.690023E-03	.0	
	4	G	.0	.0	5.873456E-02	1.270834E-04	-2.989493E-03	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	.0	.0	9.143887E-03	-1.200619E-18	-1.694704E-03	.0	
	7	G	.0	.0	3.096434E-02	-3.004362E-18	-2.565396E-03	.0	
	8	G	.0	.0	5.937611E-02	-4.946220E-18	-3.006373E-03	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	.0	.0	8.084896E-03	-2.362314E-04	-1.527135E-03	.0	
	11	G	.0	.0	2.977032E-02	-2.404278E-04	-2.690023E-03	.0	
	12	G	.0	.0	5.873456E-02	-1.270834E-04	-2.989493E-03	.0	
0	TWIST LOAD							SUBCASE 3	
	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	
	2	G	.0	.0	-1.277012E-03	1.157813E-04	2.219504E-04	.0	
	3	G	.0	.0	-3.622530E-03	3.624341E-04	2.244598E-04	.0	
	4	G	.0	.0	-5.978552E-03	5.825486E-04	2.239124E-04	.0	
	5	G	.0	.0	.0	.0	.0	.0	
	6	G	.0	.0	-3.011317E-18	1.428981E-04	5.786049E-19	.0	
	7	G	.0	.0	-1.073759E-17	3.549739E-04	9.055212E-19	.0	
	8	G	.0	.0	-2.047949E-17	5.861556E-04	1.015648E-18	.0	
	9	G	.0	.0	.0	.0	.0	.0	
	10	G	.0	.0	1.277012E-03	1.157813E-04	-2.219504E-04	.0	
	11	G	.0	.0	3.622530E-03	3.624341E-04	-2.244598E-04	.0	
	12	G	.0	.0	5.978552E-03	5.825486E-04	-2.239124E-04	.0	

Figure 5-14 Displacement Outputs for Honeycomb Sandwich Using PSHELL/MAT1

Static Loads

- Introduction to Static Loads
- Loads on Grid Points and Scalar Points
- Distributed Loads on Line Elements
- Distributed Loads on Surfaces
- Gravity and Centrifugal Force (GRAV, RFORCE)
- Initial Strains (DEFORM)
- Load Combinations
- Thermal Loads

6.1 Introduction to Static Loads

This chapter describes the static loads available when performing a linear static analysis. Each of the load types discussed may be applied to your structure individually or in any combination.

- **FORCE** and **MOMENT** entries -- Concentrated forces and moments, which are applied directly to a grid point. The direction and magnitude is defined by entering the components of a vector.
- **FORCE1** and **MOMENT1** entries -- Concentrated forces and moments, which are applied directly to a grid point. The direction of the load is defined by a line joining two grid points.
- **FORCE2** and **MOMENT2** entries -- Concentrated forces and moments, which are applied directly to a grid point. The direction of the load is determined by a cross product of two vectors that are defined using four grid points.
- **PLOAD1** entry -- A distributed load applied to a line element (e.g., **CBAR** or **CBEAM**).
- **PLOAD** entry -- Defines a pressure load applied to a surface in terms of three or four grid points.
- **PLOAD2** entry -- Defines a pressure load applied to a two-dimensional element (e.g., **CQUAD4**, **CTRIA3**, or **CSHEAR**)
- **PLOAD4** entry -- General-purpose pressure and /or traction loads applied to a two-dimensional element or the surface of a solid element.
- **GRAV**, **RFORCE** entries -- Gravity and rotational static loads. The forces on the structure are based on the mass distribution in the model.
- **DEFORM** entry -- Applies a force that is needed to obtain a specified strain in an element.

A load is applied to the model only if it is specifically called out in the Case Control Section. Having a load entry in the Bulk Data Section does not mean that the load will necessarily be applied to the structure. For reasons discussed earlier, this provides you with greater versatility. If, however, you forget to request any load in the Case Control Section, the problem will be solved with zero loads applied. There are no error or warning messages indicating that there is no load being applied. Forgetting to specify a load request in the Case Control Section can be a common mistake many new users make. An indication of this problem is when all of the displacements and stresses come out to be zero.

6.2 Loads on Grid Points and Scalar Points

Concentrated forces can be applied directly to the grid points with the FORCE, FORCE1, and FORCE2 entries. The FORCE entry is the most commonly used entry and is the one used in most of the examples up to this point. The FORCE entry allows you to specify the magnitude and direction of a force vector in any coordinate system as shown below.

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

Field	Contents
SID	Load set identification number.
G	Grid point identification number.
CID	Coordinate system identification number.
F	Scale factor.
Ni	Components of a vector measured in coordinate system defined by CID.

The magnitude of the applied force is the scale factor, entered in field 5, times the magnitude of the vector defined in fields 6 through 8. For example, the force applied with the following two FORCE entries is the same.

1	2	3	4	5	6	7	8	9	10
FORCE	1	3	0	-100.	1.	0.	2.		

1	2	3	4	5	6	7	8	9	10
FORCE	1	3	0	2.	-50.	0.	-100.		

The FORCE1 entry allows you to define a force by specifying a magnitude and two grid points-not necessarily the loaded grid point-to determine the direction. The FORCE2 entry specifies a magnitude with the direction defined by the vector product of two other vectors. The format of the FORCE1 and FORCE2 entries is as follows:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Field	Contents
SID	Load set identification number.

G	Grid point identification number.
F	Magnitude of the force.
Gi	Grid point identification numbers.

While the FORCE1 and FORCE2 forms are not used extensively, they are useful for some modeling situations as shown in the following example.

Suppose you wish to model a curved structure with edge loads as shown in [Figure 6-1](#). The edge loads consist of 10 N per element (acting normal to the elements) and 8 N per element along the edge of each element (acting as a shear load). The FORCE1 and FORCE2 entries are the ideal choice to apply these loads. You could use the FORCE entry, but it would be quite a bit more difficult because you would need to compute the normals for each of the loaded CQUAD4 elements. Using FORCE1 and FORCE2 entries for the shear and normal loads, respectively, makes the task much easier.

The 10 N normal force for each element is divided up equally between the two grid points along the edge. The directions of the resulting 5 N forces are applied using the FORCE2 entry by taking the cross product of the vectors defined by the edges to which the grid points are attached. The result is a 5 N load acting normal to the plate. The 8 N shear force for each element is also divided up equally between the two grid points along the edge. The resulting 4 N loads are applied with the FORCE1 entry. The directions of these loads are determined by the line connecting the edge grid points.

The input file is shown in [Listing 6-1](#). Note that the normal forces and the shear forces are applied as one load condition. If you want the forces to act as separate loadings, you can use different IDs for the FORCE1 and FORCE2 entries and call them out using two separate subcases.

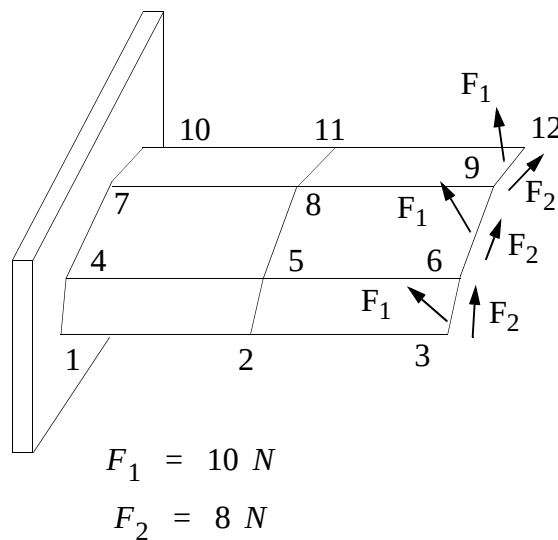


Figure 6-1 Edge Forces on a Plate Element

Listing 6-1 Edge Loads

```

$
$ FILENAME - EDGE.DAT
$
OLOAD=ALL
LOAD=3
.
.
.
BEGIN BULK
$
$ UNITS ARE CM AND N
$
FORCE1  3      3      4.      3      6
FORCE1  3      6      4.      3      6
FORCE1  3      6      4.      6      9
FORCE1  3      9      4.      6      9
FORCE1  3      9      4.      9     12
FORCE1  3     12      4.      9     12
FORCE2  3      3      5.      3      6      3      2
FORCE2  3      6      5.      3      6      3      2
FORCE2  3      6      5.      6      9      6      5
FORCE2  3      9      5.      6      9      6      5
FORCE2  3      9      5.      9     12      9      8
FORCE2  3     12      5.      9     12      9      8
$
GRID    1      0.0     -60.     -2.
GRID    2      60.     -60.     -2.
GRID    3     120.     -60.     -2.
GRID    4      0.0     -30.      0.0
GRID    5      60.     -30.      0.0
GRID    6     120.     -30.      0.0
GRID    7      0.0      30.      0.0
GRID    8      60.      30.      0.0
GRID    9     120.      30.      0.0
GRID   10      0.0      60.     -2.
GRID   11      60.      60.     -2.
GRID   12     120.      60.     -2.
$
PSHELL  1      4      .5      4
MAT1    4      7.1E6      .3
$
CQUAD4  1      1      1      2      5      4
CQUAD4  2      1      2      5      6      3
CQUAD4  3      1      4      5      8      7
CQUAD4  4      1      5      6      9      8
CQUAD4  5      1      7      8     11     10
CQUAD4  6      1      8      9     12     11
$
SPC1    1      123456  1      4      7      10
$
ENDDATA

```

You should note that the two loads are applied at the same grid points. This is perfectly acceptable-the total force acting at the grid points will be the sum of all the applied loads. The OLOAD output, requested through the Case Control Section, shows the applied force

resultant acting at a set of user specified grid points. The OLOAD output is useful in verifying that the loads are applied correctly. A partial listing of the output file showing the OLOAD output is given in [Figure 6-2](#).

L O A D V E C T O R								
POINT	ID.	TYPE	T1	T2	T3	R1	R2	R3
	3	G	0.0	3.658545E+00	5.255002E+00	0.0	0.0	0.0
	6	G	0.0	7.658545E+00	1.025500E+01	0.0	0.0	0.0
	9	G	0.0	8.323735E+00	9.722850E+00	0.0	0.0	0.0
	12	G	0.0	4.323736E+00	4.722850E+00	0.0	0.0	0.0

Figure 6-2 OLOAD Output

MOMENT, MOMENT1, and MOMENT2 are similar to the three FORCE entries. The only difference is that the force entries are used to apply concentrated force in the translational degrees of freedom (T1, T2, and T3). The moment entries apply concentrated moments in the rotational degrees of freedom (R1, R2, and R3).

The SLOAD entry is used to apply loads to scalar points only-it cannot be used with grid points.

The format of the SLOAD entry is as follows:

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	S1	F1	S2	F2	S3	F3		

Field	Contents
SID	Load set identification number.
Si	Scalar or grid point identification number.
Fi	Load magnitude.

6.3 Distributed Loads on Line Elements

To apply a distributed load to a CBAR, CBEAM, or CBEND element, you use the PLOAD1 entry. The PLOAD1 entry can be used for both concentrated and linearly distributed forces. For the CBAR and CBEAM elements, the linearly distributed force may be applied between any two locations on the element (or off the element if you wish.) For the CBEND element, the linearly distributed loads are restricted to linearly varying forces and moments between the end points.

The format of the PLOAD1 entry is as follows:

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

Field	Contents
SID	Load set identification number.
EID	CBAR, CBEAM, or CBEND element identification number.
TYPE	Load type. “FX”, “FY” or “FZ”: Force in the x-, y-, or z-direction of the basic coordinate system. “MX”, “MY” or “MZ”: Moment in the x-, y-, or z-direction of the basic coordinate system. “FXE”, “FYE” or “FZE”: Force in the x-, y-, or z-direction of the element’s coordinate system. “MXE”, “MYE” or “MZE”: Moment in the x-, y-, or z-direction of the element’s coordinate system.
SCALE	Determines scale factor for X1, X2. “LE” (length), the xi values are the actual distances along the element axis, and, if $X1 \neq X2$, then Pi values are the load intensities per unit length of the element. “FR” (fractional), the xi values are the ratios of the distance along the axis to the total length, and (if $X2 \neq X1$) Pi values are the load intensities per unit length of the element. “LEPR” (length projected), the xi values are the actual distances along the element axis, and (if $X2 \neq X1$) the distributed load is input in terms of the projected length of the element.
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A.
P1, P2	Load factors at position X1 and X2, respectively.

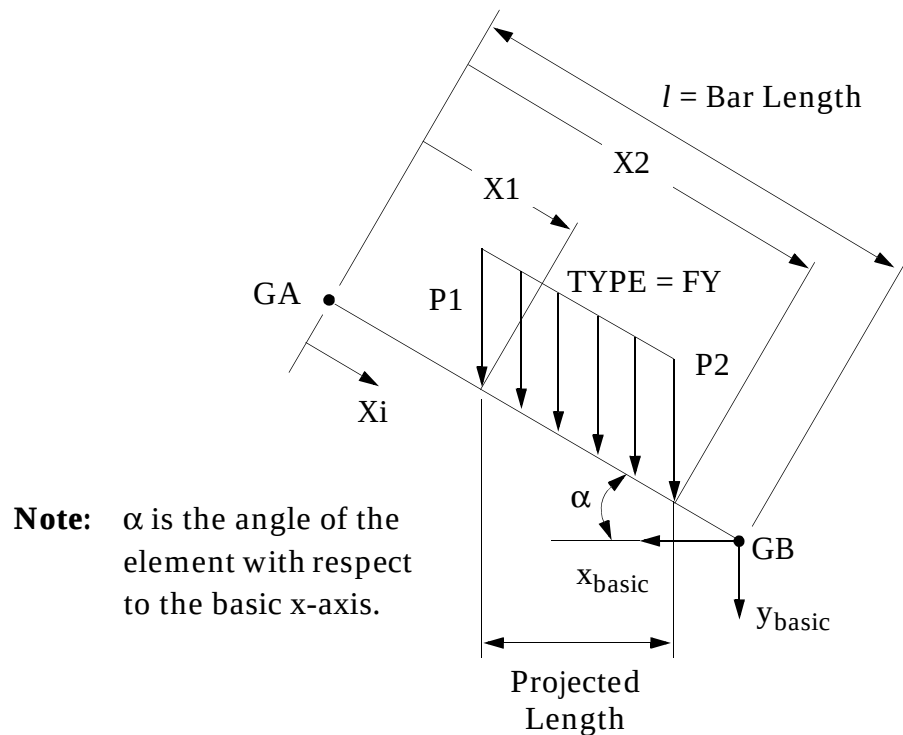


Figure 6-3 PLOAD1 Convention on Beam or Bar Elements

The type of load which is entered in field 4 may define either a concentrated force or moment in the basic or element coordinate system. If the applied load is to be a concentrated load, leave fields 8 and 9 blank and the concentrated load will be applied at the X1 location. If both X1 and X2 are input, the load will be taken as a linearly varying load between X1 and X2.

There are two ways to define the location of the load on the CBAR and CBEAM elements using field 5 of the PLOAD1 entry. Using the “LE” or “LEPR” methods, you specify the actual start and end positions of the load as measured from end A of the element. When using this method, the distances X1 and X2, as shown in [Figure 6-3](#), are in the same units as the dimensions used for the model.

The second way to define the location of the load is using “FR” or “FRPR”, in which case you specify the percent (using “X/XB”) along the element where the load starts and ends. A value of 0.0 defines end A, while a value of 1.0 defines end B. Again, the start of the load is measured from end A.

For both methods of describing the location of the loads, you also have the option of specifying whether the applied load is to be a direct load (scale “LE” and “FR”) or a projected load (scale “LEPR” or “FRPR”). For the projected loads, distributed loads are entered in terms of the projected length of the element as shown in [Figure 6-3](#). It is important to remember that if the loads are being input in terms of the basic coordinate system (“FX”, “FY”, “FZ”, “MX”, “MY”, or “MZ”), then the projected angle α is with respect to the basic coordinate system, not the element coordinate system ([Figure 6-3](#)). The effect of using projection is illustrated later with an example.

The first example demonstrating the use of the PLOAD1 entry consists of applying a direct linearly varying load to the three-bar structure shown in [Figure 6-4](#).

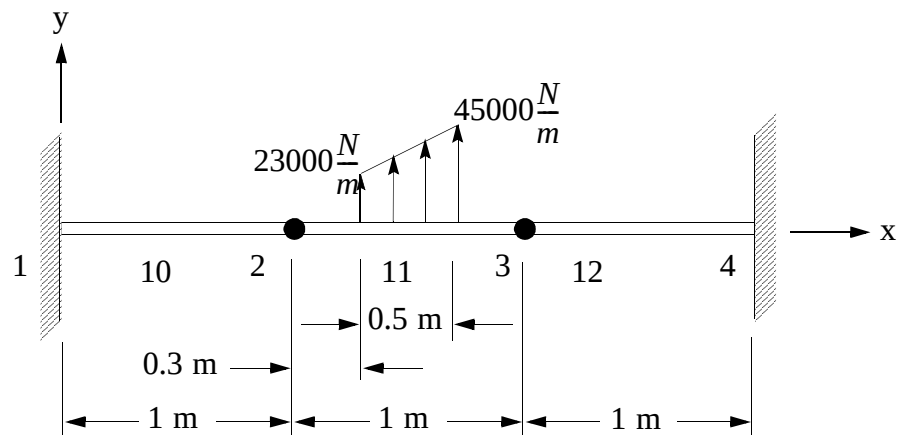


Figure 6-4 Distributed Load on CBAR Elements

The input file for this example is shown in [Listing 6-2](#).

Listing 6-2 PLOAD1 Loading

```
$
$ FILENAME - DISTRIB.DAT
$
ID LINEAR,DISTRIB
SOL 101
TIME 2
CEND
LOAD = 1
FORCE = ALL
BEGIN BULK
$
$   UNITS ARE MM AND N
$
GRID    1           0.      0.0    0.0           123456
GRID    2          1000.    0.0    0.0           345
GRID    3          2000.    0.0    0.0           345
GRID    4          3000.    0.0    0.0          123456
$
CBAR   10         1         1         2         1.0    0.0    1.0
CBAR   11         1         2         3         1.0    0.0    1.0
CBAR   12         1         3         4         1.0    0.0    1.0
CBARAO 11         FR        19        .05    .05
$
PBAR    1         1         100.0    100.0    100.0    100.0
$
MAT1    1         4.E4           .3
$
PLOAD1  1         11        FY        LE        300.    23.    800.    45.
$
ENDDATA
```

Note that the starting location for the linearly varying load starts and ends at 0.3 m (300 mm) and 0.8 m (800 mm), respectively. This is the distance measured from end A. One of the most common mistakes is to make these distances 1.3 m and 1.8 m, respectively, while thinking that it is the actual distance measured in the basic coordinate system. It is not; it is the distance measured from end A of the loaded element—in this case, grid point 2.

When you use the PLOAD1 entry to define a distributed load, you will find the CBARAO (CBAR Additional Output) entry very helpful. The CBARAO entry is used to request stress and force output at intermediate locations along the CBAR element. The CBARAO output for element 2 is shown in [Figure 6-5](#).

F O R C E D I S T R I B U T I O N I N B A R E L E M E N T S (C B A R)								
ELEMENT ID.	STATION (PCT)	BEND-MOMENT		SHEAR FORCE		AXIAL FORCE	TORQUE	
		PLANE 1	PLANE 2	PLANE 1	PLANE 2			
10	.000	.0	-5.982597E+06	.0	-7.851454E+03	.0	.0	
10	1.000	.0	1.868857E+06	.0	-7.851454E+03	.0	.0	
11	.000	.0	1.868857E+06	.0	-7.851454E+03	.0	.0	
11	.050	.0	2.261429E+06	.0	-7.851454E+03	.0	.0	
11	.100	.0	2.654002E+06	.0	-7.851454E+03	.0	.0	
11	.150	.0	3.046575E+06	.0	-7.851454E+03	.0	.0	
11	.200	.0	3.439148E+06	.0	-7.851454E+03	.0	.0	
11	.250	.0	3.831720E+06	.0	-7.851454E+03	.0	.0	
11	.300	.0	4.224293E+06	.0	-7.851454E+03	.0	.0	
11	.350	.0	4.587198E+06	.0	-6.646454E+03	.0	.0	
11	.400	.0	4.887105E+06	.0	-5.331454E+03	.0	.0	
11	.450	.0	5.118511E+06	.0	-3.906454E+03	.0	.0	
11	.500	.0	5.275917E+06	.0	-2.371454E+03	.0	.0	
11	.550	.0	5.353823E+06	.0	-7.264532E+02	.0	.0	
11	.600	.0	5.346729E+06	.0	1.028546E+03	.0	.0	
11	.650	.0	5.249135E+06	.0	2.893546E+03	.0	.0	
11	.700	.0	5.055540E+06	.0	4.868546E+03	.0	.0	
11	.750	.0	4.760446E+06	.0	6.953546E+03	.0	.0	
11	.800	.0	4.358353E+06	.0	9.148546E+03	.0	.0	
11	.850	.0	3.900926E+06	.0	9.148546E+03	.0	.0	
11	.900	.0	3.443498E+06	.0	9.148546E+03	.0	.0	
11	.950	.0	2.986070E+06	.0	9.148546E+03	.0	.0	
11	1.000	.0	2.528644E+06	.0	9.148546E+03	.0	.0	
12	.000	.0	2.528644E+06	.0	9.148546E+03	.0	.0	
12	1.000	.0	-6.619903E+06	.0	9.148546E+03	.0	.0	

Figure 6-5 CBARAO Output for the Center Element

The shear and moment diagrams are shown in [Figure 6-6](#). As you can see, the CBARAO entry is one of the best tools available for model checkout when using linearly varying loads on the CBAR elements. There is no easier way to generate a shear and moment diagram than by using the CBARAO output request.

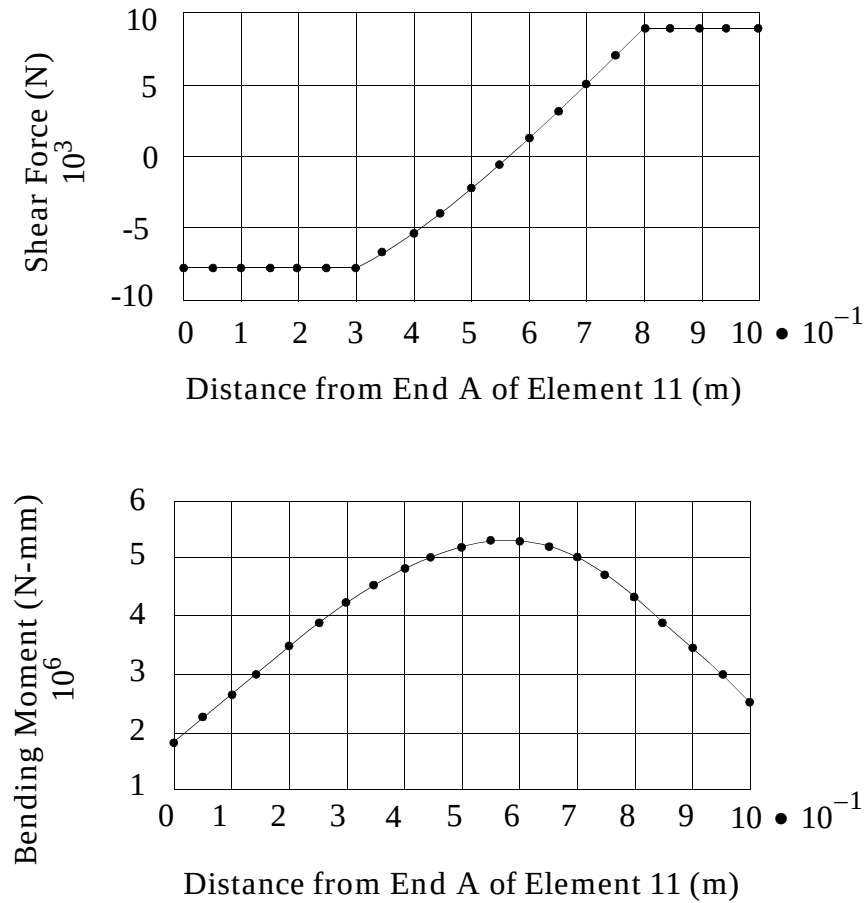


Figure 6-6 Shear and Moment Diagram Generated from CBARAO Output

For CBEAM elements, intermediate output along the element can be requested directly on the PBEAM entry.

Another feature of the PLOAD1 entry is the ability to apply projected loads. To apply a load as a projected load, you choose a scale (field 5) of “LEPR” for the actual length from end A or “FRPR” for the fractional distance from end A. A typical example of a projected load is that of a snow load on a truss structure as shown in [Figure 6-7](#).

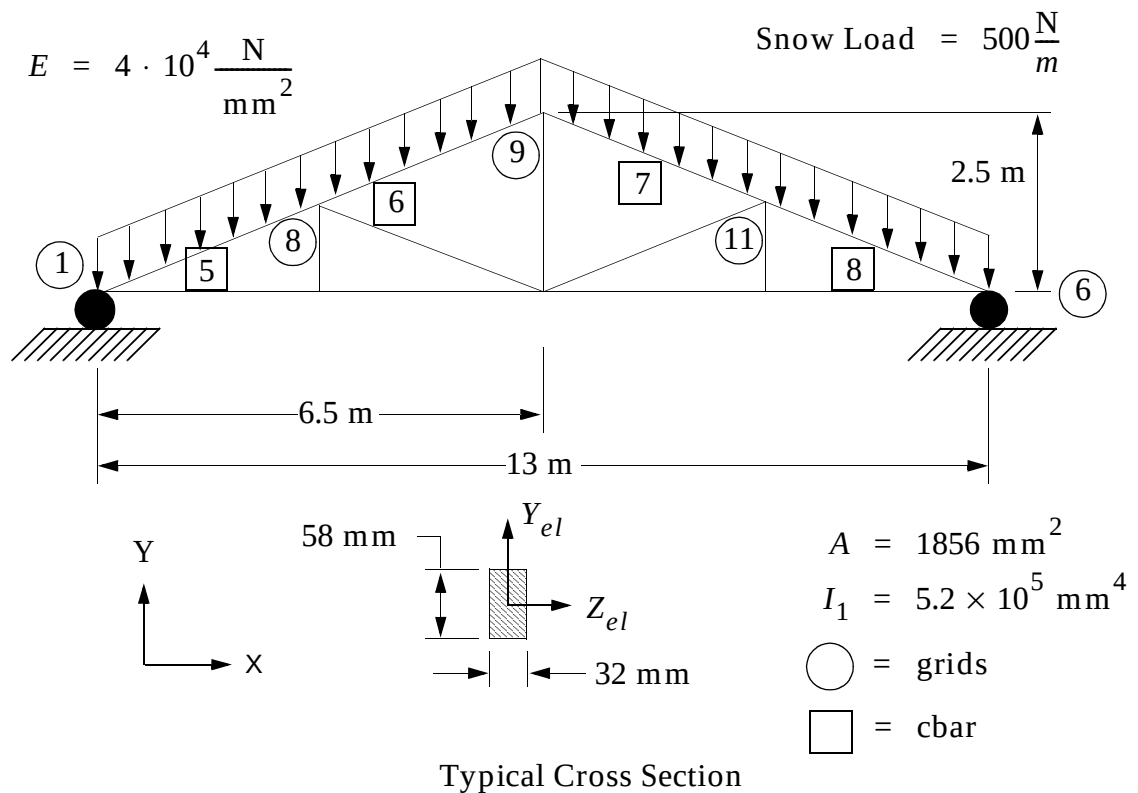


Figure 6-7 Planar Truss Structure with a Snow Loading

The snow load on the truss is 500 N/m acting in the basic Y-direction. The input file containing the grid points and PLOAD1 entries is shown in [Listing 6-3](#).

Listing 6-3 Truss with Snow Loading

```

$
$  FILENAME - PLOAD1.DAT
$
$  UNITS ARE MM AND N
$
GRID    1           0.0      0.0      0.0           345
GRID    6          13000.    0.0      0.0           345
GRID    8           3250.    1250.    0.0           345
GRID    9           6500.    2500.    0.0           345
GRID   11           9750.    1250.    0.0           345
$
CBAR    5           1         1         8         1.0      1.0      0.0
CBAR    6           1         8         9         1.0      1.0      0.0
CBAR    7           1         9        11         1.0      1.0      0.0
CBAR    8           1        11         6         1.0      1.0      0.0
$
PLOAD1  1           5         FY        FRPR      0.0      -0.5      1.0      -0.5
PLOAD1  1           6         FY        FRPR      0.0      -0.5      1.0      -0.5
PLOAD1  1           7         FY        FRPR      0.0      -0.5      1.0      -0.5
PLOAD1  1           8         FY        FRPR      0.0      -0.5      1.0      -0.5

```

To see the effect of using projected forces, the resulting SPC forces are shown in [Figure 6-8](#). The total reaction of the loads obtained by adding the SPC forces is 6500 N in the Y-direction. The total reaction load is equal to the projected length of 13 meters times the

distributed load of 500 N / m. If the projected option had not been used, the applied load would have been 6964.2 N, which is the total length of the top members times the distributed load.

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	-2.511155E-05	3.250000E+03	.0	.0	.0	.0
	6	G	.0	3.250000E+03	.0	.0	.0	.0

Figure 6-8 SPC Forces Due to the Snow Loading

As a final comment on the PLOAD1 entry, if you are using a concentrated load and do not need the additional features that the CBEAM element offers, you are better off using the CBAR element. This is due to the way the equivalent end loads are generated. For the CBAR element, the equivalent end loads are generated explicitly. For the CBEAM element, the end loads are generated by numerical integration along the length of the element, which may not be as accurate as the method used for the CBAR element. A linearly varying load applied with a PLOAD1 entry is accurate when used with either element.

6.4 Distributed Loads on Surfaces

There are three Bulk Data entries available to apply distributed loads to element surfaces. The PLOAD and PLOAD2 entries are used to apply a distributed load to the CQUAD4, CTRIA3, and CSHEAR elements only. The PLOAD4 entry can be used to apply distributed loads on any of the two-dimensional elements and on the surfaces of three-dimensional elements.

The PLOAD entry is different from the other PLOADi entries because it references three or four grid points rather than the element itself. When the PLOAD entry is used to load a CTRIA3 element, then the total force acting on the element is divided evenly among the three corner points. When the PLOAD entry is used to load a CQUAD4 element, the total force acting on the element is distributed to corner grid points using the geometric shape of the element. The resultant of the applied corner loads acts through the centroid of the element. The direction of the pressure load is determined using the right-hand rule by the numbering sequence of the grid points on the PLOAD entry.

The format of the PLOAD entry is as follows:

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

Field	Contents
SID	Load set identification number.
P	Pressure.
Gi	Grid point identification numbers.

The PLOAD2 entry is similar to the PLOAD entry except that the PLOAD2 references the element ID instead of the grid points where the element is attached. The PLOAD2 is usually preferred over the PLOAD entry because it is easier to use. The PLOAD2 entry, similar to the PLOAD entry, is limited to pressure acting normal to the element surface. The direction of the pressure load is determined by the numbering sequence of the grid points on the connectivity entry (CQUAD4, CTRIA3, etc.).

The PLOAD4 entry possesses the most general pressure definition. It can be used to apply pressures and / or tractions to any of the two-dimensional elements and the surfaces of the three-dimensional elements. The PLOAD4 entry can be used for everything that a PLOAD2 entry is used for. The direction of the pressure can be set by one of two methods. By default, the direction of the pressure is determined by the element normal. For two-dimensional elements, the direction of positive pressure is in the direction of the outward normal as is determined by applying the right-hand rule to the ordering sequence of the grid points on the connectivity entry. For surfaces of solid elements, the direction of positive pressure is inward toward the center of the element. The face of the solid to which you are applying the pressure is determined by specifying the appropriate corner grid points.

The format of the PLOAD4 entry is as follows:

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

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3					

Field	Contents
SID	Load set identification number.
EID	Element identification number.
EID1	
EID2	
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element.
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only.
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 the 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.
CID	Coordinate system identification number.
N1, N2, N3	Components of the vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity.

Alternatively, the direction of the pressure defined by the PLOAD4 entry can be input using an optional coordinate system and a vector entered on the continuation line. Using a local coordinate system, a pressure acting at any angle to the surface may be defined. Loads acting parallel to the surface, which are referred to as tractions, can also be applied.

Another feature unique to the PLOAD4 entry is the ability to apply a nonuniform pressure. The pressure may be entered at each of the corner grid points resulting in a linearly varying pressure load. If the pressure is entered for the first grid point only, the pressure is assumed to be constant over the element.

Later in the chapter, an example is shown that takes advantage of the optional coordinate system, but first it is useful to demonstrate the PLOAD4 entry as it is commonly used.

Consider the curved plate shown in **Figure 6-9**. PLOAD4 entries are to be used to apply a normal pressure to each of the six CQUAD4 elements.

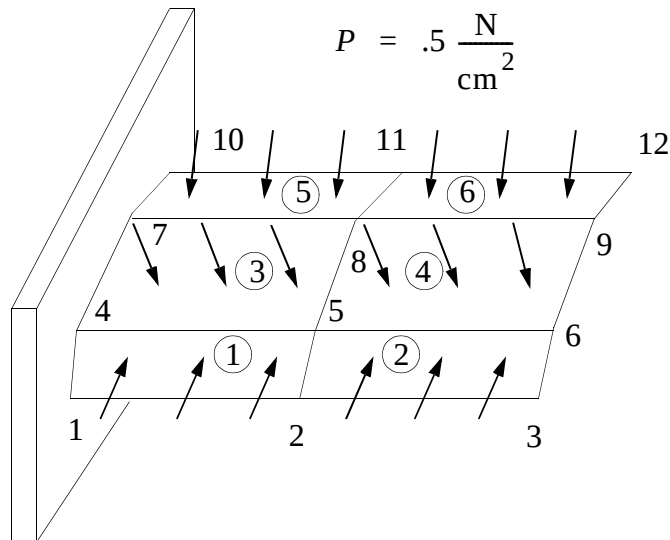


Figure 6-9 Pressure Loads on the CQUAD4s

This input file is given in **Listing 6-4**. When using the PLOAD4 entry, you may specify a PLOAD4 entry for every element or use the alternate method of specifying several elements that have the same pressure. The alternate method is used for this example.

Listing 6-4 CQUAD4 Elements with Pressure Loads

```

$
$  FILENAME - PRESS.DAT
$
$  UNITS ARE CM, N
$
PLOAD4    3      1      -.5                      THRU    6
$
GRID      1      0.0      -60.      -2.
GRID      2      60.      -60.      -2.
GRID      3      120.     -60.      -2.
GRID      4      0.0      -30.      0.0
GRID      5      60.      -30.      0.0
GRID      6      120.     -30.      0.0
GRID      7      0.0      30.       0.0
GRID      8      60.      30.       0.0
GRID      9      120.     30.       0.0
GRID     10      0.0      60.      -2.
GRID     11      60.      60.      -2.
GRID     12      120.     60.      -2.
$
CQUAD4    1      1      1      2      5      4
CQUAD4    2      1      2      5      6      3
CQUAD4    3      1      4      5      8      7
CQUAD4    4      1      5      6      9      8
CQUAD4    5      1      7      8      11     10
CQUAD4    6      1      8      9      12     11
$
SPC1      1      123456  1      4      7      10
$
PSHELL    1      4      .5      4
MAT1      4      7.1E6      .3

```

The resulting deflection is shown in [Figure 6-10](#). The deflected shape is not what you would expect. Element 2 appears to bend up. The problem occurs because the normal for element 2 is reversed (see the ordering of the grid point IDs on the CQUAD4s). The pressure on this element is acting upward. This type of mistake is quite common. Model checks to determine reversed normals and other potential modeling errors are discussed in “[Model Verification](#)” on page 369.

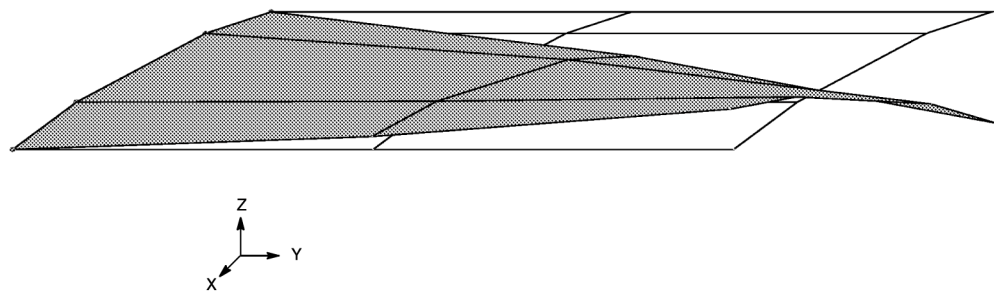


Figure 6-10 Displacements Due to Pressure Load

6.5 Gravity and Centrifugal Force (GRAV, RFORCE)

The GRAV entry is used to define the direction and magnitude of a gravity vector in any user-defined coordinate system. The components of the gravity vector are multiplied by the mass matrix to obtain the components of the gravity force at each grid point. Since the mass matrix is used to compute the forces, you must have mass in your model, typically defined by the density on a material entry. Note that the GRAV entry must have a unique SID-no other loading entry may use the same ID. The LOAD entry (discussed in the next section) can be used to combine gravity loading with other types of loading.

The format of the GRAV entry is as follows:

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

Field	Contents
SID	Set identification number.
CID	Coordinate system identification number.
G	Acceleration vector scale factor.
Ni	Acceleration vector components measured in coordinate system CID.

Gravity loading is one the best loadings to use when you are performing model checkout because the force is applied at every point in the structure that has mass. If you have modeling errors where elements or grids points are not attached to the structure, their displacement is unbounded and the job fails. If this occurs, a diagnostic message indicating the grid point and component, where the singularity is detected, is printed. An example using gravity loading is shown in “Gravity and Centrifugal Force (GRAV, RFORCE)” on page 234.

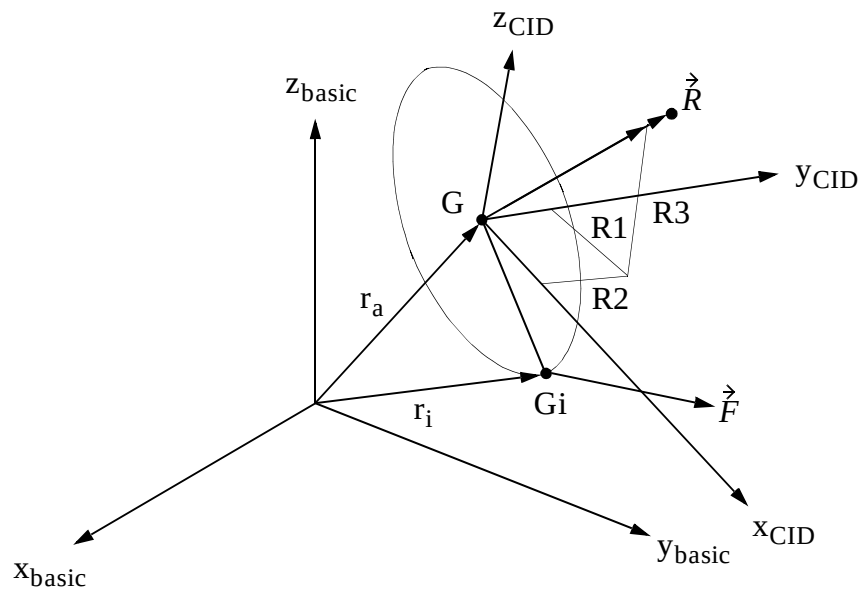
The RFORCE entry is used when you need to apply a force to your structure due to rotational velocity and /or acceleration. On the RFORCE entry, you input the components of a spin vector that are used internally to compute centrifugal forces. Each component of the spin vector is multiplied by the same scale factor.

The format of the RFORCE entry is as follows:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHO D	
	RACC								

Field	Contents
SID	Load set identification number.

G	Grid point identification number through which the rotation vector acts.
CID	Cartesian coordinate system defining the components of the rotation vector.
A	Scale factor of the angular velocity in revolutions per unit time.
R1, R2, R3	Rectangular components of rotation vector $\vec{R}$. The vector defined passes through point G.
METHOD	Method used to compute centrifugal forces due to angular velocity.
RACC	Scale factor of the angular acceleration in revolutions per unit time squared.



$$\{\vec{F}\}_i = [m]_i [\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)]$$

where:

$$\text{angular velocity} = \vec{\omega} = 2\pi A \cdot \vec{R} \text{ (radians/unit time)}$$

$$\text{angular acceleration} = \vec{\alpha} = 2\pi \text{RACC} \cdot \vec{R} \text{ (radians/unit time squared)}$$

$$[m]_i = 3 \times 3 \text{ translational mass matrix at grid point Gi}$$

Note: The equation for $\vec{F}_i$ has additional terms if the mass is offset and METHOD = 1 is selected.)

Figure 6-11 RFORCE Vector at Grid Point Gi

You must select one of two methods for the internal calculation of the loading vector (field 9 on the RFORCE entry).

Method=1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). Method=2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with method=2. All the possible combinations of mass matrices and offset and the correct method to be used are shown in [Table 6-1](#).

Table 6-1 Restrictions When Using the RFORCE Entry

	No Offset	Offset
Lumped	Method=1 or Method=2	Method=1
Coupled	Method=2	Neither

In addition, for problems with elements that have edge grid points (CQUAD8, CTRIA6, CTRLAX6, CHEXA, CPENTA, and CTETRA), correct centrifugal loads are produced only if the parameter PARAM,COUPMASS,x (where x is greater than 1), is included in the input file and Method 2 is used.

Defining Mass in Your Model

Common ways to define mass are the concentrated masses (CMASSi and CONMi), mass density on the material entries (MAT1, MAT2, etc.), and nonstructural mass defined on the property entries. The mass density defined on the material entries is given in terms of mass/unit volume. You must be sure the mass unit is consistent with the other units in the model. For example, in the English system (in, lb, sec), the mass density of steel is approximately $32 \cdot 10^{-4} \text{ lb-sec}^2/\text{in}$.

The nonstructural mass defined on the property entries is mass that is added to the structure in addition to the structural mass from the elements. For one-dimensional elements, the units are mass/unit length. For two-dimensional and three-dimensional elements, the units are mass/unit area and mass/unit volume, respectively.

It is often convenient to express the mass in terms of weight units instead of mass units. This can be accomplished with the use of the parameter WTMASS. The function of the WTMASS parameter is to multiply the assembled mass matrix by the scale factor entered with the WTMASS parameter. For the steel example, the mass density can be entered as a weight density of 0.283 lb/in^3 with a WTMASS parameter of 0.00259 (which is $1/386.4$). As a word of caution: if you enter any of the mass in terms of weight, you must enter all the mass in terms of weight. The WTMASS multiplies all of the mass in the model by the same scale factor. (The mass entered in Direct Matrix Input is an exception and is discussed in “[Model Verification](#)” on page 369 and “[Glossary of Terms](#)” on page 599.)

Consider the two rotating masses shown in [Figure 6-12](#). The masses are accelerated at constant angular acceleration of 20 rev/sec^2 from 0 to 200 rev/min . The goal is to determine the axial force in the CBAR as a function of angular velocity.

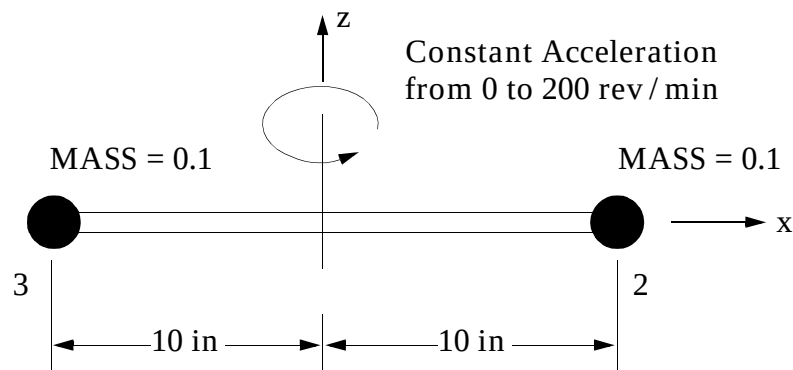


Figure 6-12 Rotating Concentrated Masses

The input file is shown in [Listing 6-5](#). Multiple load cases are used to determine axial forces at six rotational speeds. Using multiple load cases for this problem is much more efficient than submitting separate runs because MSC.Nastran does not have to regenerate the stiffness and mass matrices. The matrices are generated and decomposed (inverted) just once. Another interesting note is that the model really does not rotate; it is actually fixed at the center. If the constraints permitted the model to rotate, the run would fail.

Listing 6-5 Axial Force in the Rotating Arms

```
$
$ FILENAME ROTATE.DAT
$
ID LINEAR,ROTATE
SOL 101
TIME 2
CEND
$
  FORCE  = ALL
SUBCASE 1
  LABEL = 0 REV / min
  LOAD  = 1
SUBCASE 2
  LABEL = 40 REV / min
  LOAD  = 2
SUBCASE 3
  LABEL = 80 REV / min
  LOAD  = 3
SUBCASE 4
  LABEL = 120 REV / MIN
  LOAD  = 4
SUBCASE 5
  LABEL = 160 REV / MIN
  LOAD  = 5
SUBCASE 6
  LABEL = 200 REV / MIN
  LOAD  = 6
BEGIN BULK
PARAM  GRDPNT 0
```

Listing 6-5 Axial Force in the Rotating Arms (continued)

```

$
GRID      1      0.0      0.0      0.0      123456
GRID      2      10.      0.0      0.0
GRID      3     -10.      0.0      0.0
$
CBAR     10      1      1      2      1.0      0.0      1.0
CBAR     11      1      1      3      1.0      0.0      1.0
$
PBAR      1      1      1.0      1.0      1.0      1.0
$
MAT1      1      30.E6      .3
$
RFORCE    1      1      0.      0.0      0.0      1.0      1
20.
RFORCE    2      1      .666667 0.0      0.0      1.0      1
20.
RFORCE    3      1      1.333333 0.0      0.0      1.0      1
20.
RFORCE    4      1      2.0      0.0      0.0      1.0      1
20.
RFORCE    5      1      2.666667 0.0      0.0      1.0      1
20.
RFORCE    6      1      3.333333 0.0      0.0      1.0      1
20.
$
CONM2    101      2      .1
CONM2    102      3      .1
$
ENDDATA

```

A plot summarizing the axial force is shown in [Figure 6-13](#). As expected, the graph is not linear because centrifugal forces are not linear with angular velocity. It follows, therefore, that using a SUBCOM command to combine the subcases with RFORCE entries can lead to misleading results. In other words, the element axial forces for Subcase 3 cannot be obtained by merely doubling the element axial forces of Subcase 2 by using a Subcom / Subseq combination. This is due to the fact that the axial force is a function of ω^2 , and not two times ω .

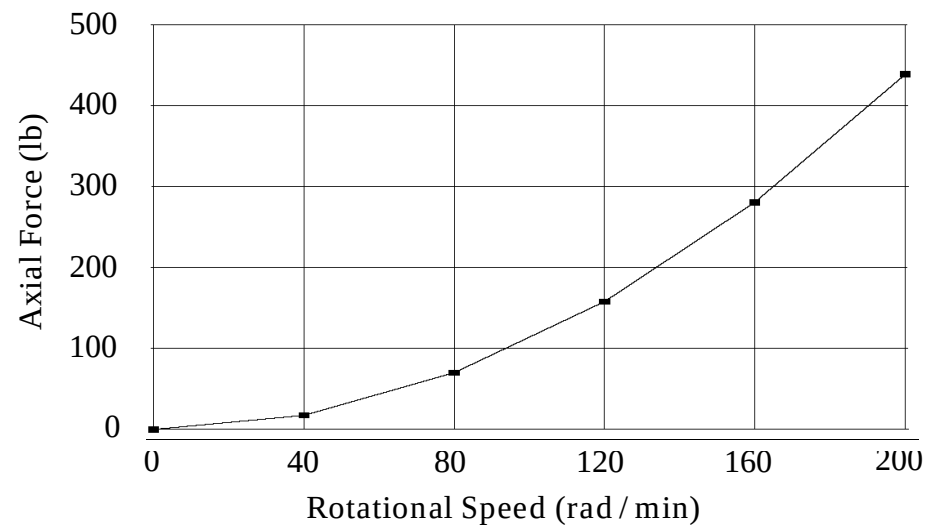


Figure 6-13 Axial Force in the Bar as a Function of Angular Velocity

6.6 Initial Strains (DEFORM)

MSC.Nastran includes a limited capability to apply initial strains. The DEFORM Bulk Data entry can be used to apply an enforced extension (units of length) to any line element except the CBEND element. The thermal expansion capability can also be used to provide initial strains besides the thermal strain. This feature is discussed in “[Thermal Loads](#)” on page 249.

The format of the DEFORM entry is as follows:

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

Field	Contents
SID	Deformation set identification number.
EIDi	Element number.
Di	Deformation.

When using the DEFORM entry, you must remember that in general you are not enforcing a strain or an actual extensional length to the element. What you are doing is applying a force to the element that produces the specified extension if the element is free to expand without internal forces being generated. This computed force is added to the other forces in the model. Since most elements in your model are not free to expand, the extension value you specify may not be achieved. This feature is best illustrated with an example.

Suppose you want to use the DEFORM entry to enforce a strain on the three-member structure shown in [Figure 6-14](#). The goal is to impose an initial strain equal to $100\text{ }\mu\text{m/cm}$ in the center member.

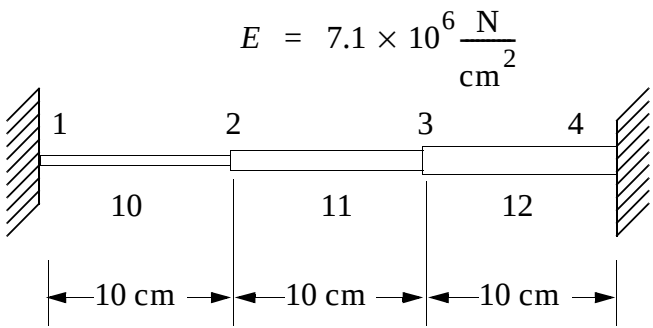


Figure 6-14 Three-Bar Structure with Initial Strain

Before solving this problem, we must consider the following items: (1) the DEFORM entry can only impose a force, (2) from static equilibrium, if no forces are imposed on grid points 2 and 3, the force in all three members must be the same. With this in mind, the following procedure is used.

1. Determine the force required in member 11 that produces the 100 mcm/cm strain.

$$\begin{aligned}
\sigma_{11} &= E\varepsilon \\
&= (7.1 \times 10^6)(100 \times 10^{-6}) = 710 \frac{\text{N}}{\text{cm}^2} \\
P &= A\sigma_{11} \\
&= (1.5)(710) = 1065 \text{ N}
\end{aligned}$$

2. Since there are no external forces acting on the elements, the force of 1065 N is same in all the members. Using the 1065 N force, the stresses computed in the other members.

$$\begin{aligned}
\sigma_{10} &= \frac{P}{A_{10}} = \frac{1065}{1} = 1065 \frac{\text{N}}{\text{cm}^2} \\
\sigma_{12} &= \frac{P}{A_{12}} = \frac{1065}{2} = 532.5 \frac{\text{N}}{\text{cm}^2}
\end{aligned}$$

3. From the stresses found in part 2, the element strains are computed for elements 10 and 12. Multiplying the strain within each element by its length yields the extensional change of the element. These extensional changes are the deform values required to produce a 100 $\mu\text{cm}/\text{cm}$ strain in element 11.

$$\begin{aligned}
\varepsilon_{10} &= \frac{\sigma_{10}}{E} = 150 \cdot 10^{-6} \\
\varepsilon_{12} &= \frac{\sigma_{12}}{E} = 75.0 \cdot 10^{-6}
\end{aligned}$$

Therefore,

$$\begin{aligned}
\Delta L_{10} &= L_{10}\varepsilon_{10} = (10)(150 \cdot 10^{-6}) = 1.5 \cdot 10^{-3} \text{ cm} \\
\Delta L_{11} &= L_{11}\varepsilon_{11} = (10)(150 \cdot 10^{-6}) = 1.0 \cdot 10^{-3} \text{ cm} \\
\Delta L_{12} &= L_{12}\varepsilon_{12} = (10)(75 \cdot 10^{-6}) = 0.75 \cdot 10^{-3} \text{ cm}
\end{aligned}$$

These values are entered on the DEFORM Bulk Data entry as shown in [Listing 6-6](#). To help understand the implementation of the DEFORM entry, two configurations are being analyzed. The first configuration (Subcase 1) specifies the boundary conditions as shown in [Figure 6-15\(a\)](#). For this case, the structure is free to expand without inducing any element forces. The results of this subcase should be the extension of the members as entered on the DEFORM entry. As mentioned previously, a force is applied to the element that produces the specified extension if the element is free to expand.

The second subcase, shown in [Figure 6-15\(b\)](#), is the constrained configuration that is of interest. If the calculations are correct, the strain in the center element should be the required 100 $\mu\text{cm}/\text{cm}$.

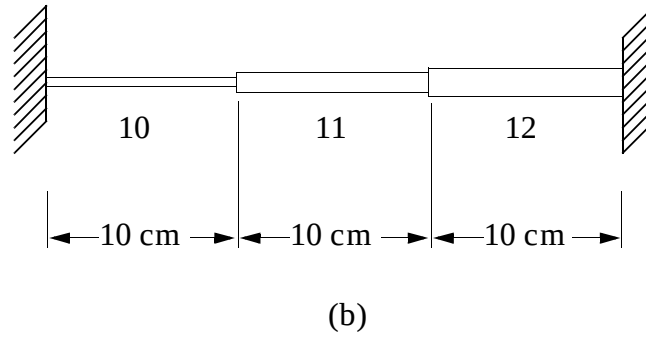
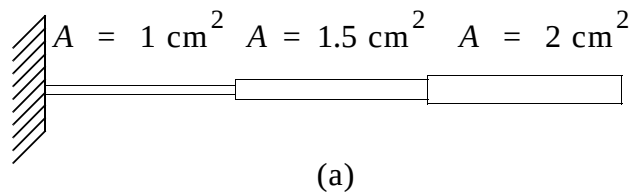


Figure 6-15 Three-Bar Structures with Initial Strain

Listing 6-6 Initial Strain Input File

```
$
$ FILENAME - STRAIN.DAT
$
ID LINEAR,STRAIN
SOL 101
TIME 2
CEND
$
DISP      = ALL
STRESS    = ALL
DEFORM    = 1
SUBCASE 1
SUBCASE 2
  SPC = 1
BEGIN BULK
$
SPC      1      4      1
$
GRID     1      0.0    0.0    0.0      123456
GRID     2      10.    0.0    0.0      23456
GRID     3      20.    0.0    0.0      23456
GRID     4      30.    0.0    0.0      23456
$
CBAR     10     1      1      2      1.0    0.0    1.0
CBAR     11     2      2      3      1.0    0.0    1.0
CBAR     12     3      3      4      1.0    0.0    1.0
$
PBAR     1      1      1.0
PBAR     2      1      1.5
PBAR     3      1      2.0
$
MAT1     1      7.1E6      .3
$
DEFORM   1      10      1.5E-3  11      1.E-3  12      .75E-3
$
ENDDATA
```

The stresses and displacements for the two subcases are shown in [Figure 6-16](#). As can be seen for the free end configuration, the extension of all of the members is consistent with the DEFORM entry.

Contrary to your expectations for the fixed end configuration, the extensions for the elements are not the same as the values entered on the DEFORM entry. The stress in member 11 is 710 N/cm^2 . Since $E = 7.1 \times 10^6$, the strain in element 11 is equal to $\epsilon = E/\sigma = 100 \text{ } \mu\text{cm/cm}$ strain. The stress output is requested for this example because strain output is not available for one-dimensional elements.

It is not expected that you will necessarily follow this procedure for an actual structure. The purpose of the example is to demonstrate how the DEFORM entry is implemented. Once you understand how it is implemented, it can be useful for applying initial strains to your model.

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	
2	G	1.500000E-03	.0	.0	.0	.0	.0	
3	G	2.500000E-03	.0	.0	.0	.0	.0	
4	G	3.250000E-03	.0	.0	.0	.0	.0	
SUBCASE 2								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	-1.791005E-11	.0	.0	.0	.0	.0	
3	G	8.955025E-12	.0	.0	.0	.0	.0	
4	G	.0	.0	.0	.0	.0	.0	
SUBCASE 1								
S T R E S S E S I N B A R E L E M E N T S (C B A R)								
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S. - T M.S. - C
10	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0	.0	.0	.0	
11	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0	.0	.0	.0	
12	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0	.0	.0	.0	
SUBCASE 2								
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S. - T M.S. - C
10	.0	.0	.0	.0	-1.065000E+03	-1.065000E+03	-1.065000E+03	
	.0	.0	.0	.0	-1.065000E+03	-1.065000E+03	-1.065000E+03	
11	.0	.0	.0	.0	-7.100000E+02	-7.100000E+02	-7.100000E+02	
	.0	.0	.0	.0	-7.100000E+02	-7.100000E+02	-7.100000E+02	
12	.0	.0	.0	.0	-5.325000E+02	-5.325000E+02	-5.325000E+02	
	.0	.0	.0	.0	-5.325000E+02	-5.325000E+02	-5.325000E+02	

Figure 6-16 Stress and Displacement Output of the Three Bar Structure

6.7 Load Combinations

Each applied load entry has a set identification number, SID, which may be the same or different from other applied load entries. The purpose of the SID is to allow different loading conditions within the same analysis without requiring that entries be physically added to or removed from the Bulk Data Section.

Subcases, which may or may not include different loading conditions, are requested in the Case Control Section. Three Case Control commands (LOAD, TEMP(LOAD), and DEFORM) are used in selecting loading conditions for subcases. Each of these commands identifies a unique set whose SID is the same as the SID of one or more Bulk Data entries.

Figure 6-17 shows the Bulk Data entries that can be selected by each of the three Case Control commands.

The LOAD Bulk Data entry is used to combine the loads specified on other Bulk Data entries to form new loading conditions. Also, as noted in **Figure 6-17**, the LOAD entry is required if you want to combine gravity loads with other applied loads, even if there is only one loading condition. The LOAD entry provides separate factors for each of the individual load sets and also an overall scale factor.

The format of the LOAD entry is as follows:

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

Field	Contents
SID	Load set identification number.
S	Overall scale factor.
Si	Scale factor on Li.
Li	Load set identification numbers defined on entry types listed above.

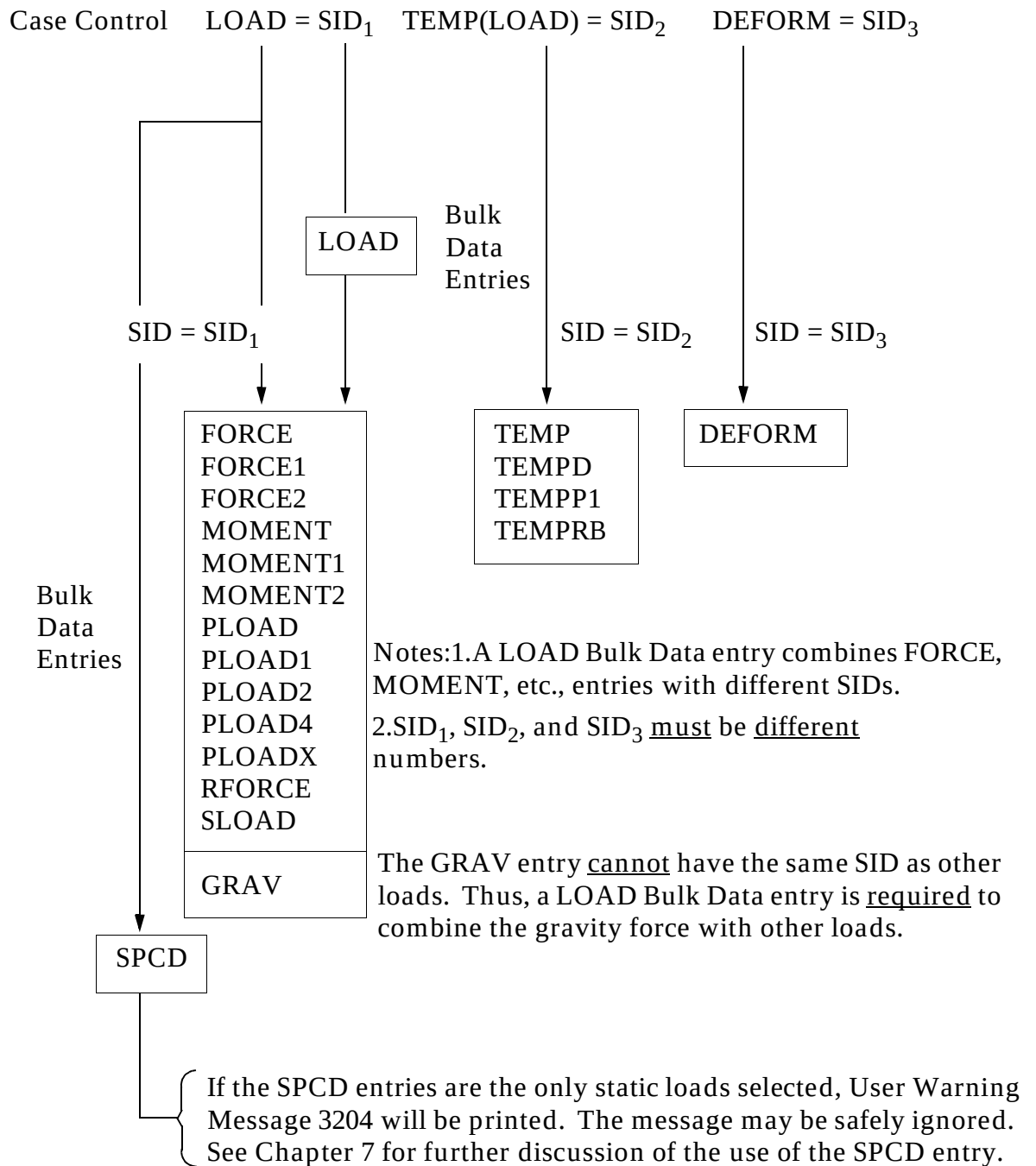


Figure 6-17 Load Combinations

As an example of the LOAD entry combining two loads, consider the umbrella tent shown in [Figure 6-18](#).

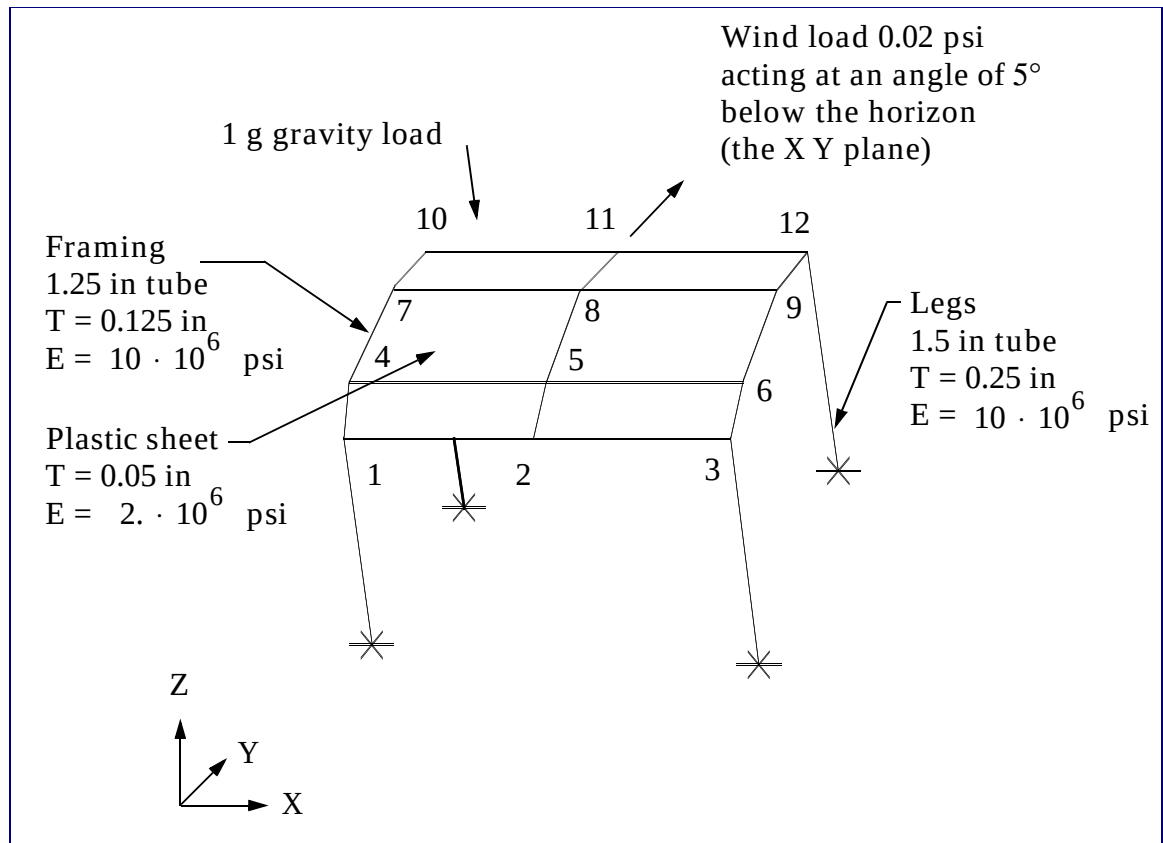


Figure 6-18 Umbrella Tent Subjected to Gravity and Wind Loads

The input file containing the combined loading is shown in [Listing 6-7](#). To apply the wind pressure in the correct direction, a local coordinate system is defined so that the local Y axis makes an angle of 5 degrees with the ground. This coordinate system is referenced on the PLOAD4 continuation entry. Since all of the elements in the model are exposed to the wind load, the simplest method is to use the thru option on the PLOAD4 entry (fields 8 and 9).

The gravity load of 386.4 entered on the GRAV entry corresponds to a 1 G load in the English system. Whenever you are working with an acceleration vector in terms of Gs, make sure that you convert the load such that its units are consistent with your structure.

Listing 6-7 Umbrella Tent with Wind and Gravity Loads

```

$
$   FILENAME - TENT.DAT
$
:
LOAD=5
:
BEGIN BULK
$
CORD2R  1      0      0.      0.      0.      0.      .08716  1.
        1.      .08716  1.
$
GRAV    1      0      386.4  0.0    0.0    -1.0
PLOAD4  2      1      .02          THRU    6
        1      0.      1.      0.
LOAD    5      1.      1.      1      1.      2
$
PARAM   POST    0
PARAM   AUTOSPC YES
PARAM   GRDPNT  0
$
PSHELL  1      4      .05
$
MAT1    1      10.E6      .3      2.59E-3
        25000.  25000.  15000.
$
MAT1    4      2.E6      .3      2.59E-5
        25000.  25000.  15000.
$
$ GRID, ELEMENT, AND CONSTRAINTS ARE NOT SHOWN
$
ENDDATA

```

The gravity load is defined by the GRAV entry with a loading set ID of 1. The wind loading is defined using the PLOAD4 entry with a loading set ID of 2. The LOAD entry (ID of 5) is used to combine the gravity and wind loadings. The LOAD entry is selected by the LOAD = 5 command in the Case Control Section. If the GRAV and the PLOAD4 entries were assigned the same ID, a fatal error would have resulted.

6.8 Thermal Loads

Thermal loads can be used on a structure to perform stress analysis or to determine thermal expansion. You must define a temperature distribution via TEMPij Bulk Data entries and thermal expansion coefficients. Thermal expansion coefficients are specified on the material Bulk Data entries. Temperatures can be specified at grid points (TEMP and TEMPD entries) and interpolated to grid points within elements. Alternatively, temperature data can be specified on an element-by-element basis as shown in **Table 6-2**.

Table 6-2 Bulk Data Entries Used for Temperature Definition on Elements

Elements	Temperature Data	Bulk Data Entry
CROD, CONROD, CTUBE	Average temperature at ends A and B.	TEMPRB
CBAR, CBEAM, CBEND	Average temperature and cross-sectional temperature gradients at ends A and B.	TEMPRB
CQUAD4, CTRIA3 CQUAD8, CTRIA6	Average temperature and gradient in the thickness direction.	TEMPP1

Average temperatures specified directly for an element take precedence over the temperatures interpolated from the element's connected grid points. Solid elements obtain their temperatures only by interpolation from connected grid points. Note that interpolated grid point temperatures provide temperature gradients over the neutral surface of shell elements, whereas the TEMPPi entries do not.

The temperature data and the thermal expansion coefficients are used internally to calculate equivalent forces and moments acting at the grid points.

The TEMP(INIT) and TEMP(LOAD) Case Control commands specify the initial temperature and applied temperature, respectively. The TEMP(INIT) command must appear either above the first subcase or inside the first subcase.

Using the SUBCOM with Thermal Loads

The SUBCOM/SUBSEQ combination is quite straightforward for mechanical loads, but it takes more effort when it is used for thermal loads. The mechanical load case need not be called out in the SUBCOM subcase; however, for thermal loads they do need to be called out. Element stresses and forces are calculated on an element-by-element basis from a knowledge of the displacement vector and the temperature field. Therefore, in a SUBCOM or a SYMCOM, you must supply a definition of the temperature field whenever element stresses and/or forces are requested. The TEMP(LOAD) for the subcom is calculated as follows:

$$T_{subcom} = T_0 + \sum_{i=1}^n a_i(T_i - T_0) \quad \text{Eq. 6-1}$$

where:

T_0 = reference temperature

T_i = temperature applied in Subcase i

a_i = subseq factor for Subcase i

The new TEMP(LOAD) must be supplied through the standard temperature Bulk Data entries such as TEMPD, TEMP, etc., and called out in the SUBCOM Case Control Section. This is best illustrated with the example problem shown in [Listing 6-8](#).

Listing 6-8 Using SUBCOMs with Temperature Loads

```
.  
.   
.   
CEND  
$  
TEMP(INIT) = 1  
$  
SUBCASE 1  
TEMP(LOAD) = 10  
$  
SUBCASE 2  
TEMP(LOAD) = 15  
$  
$ THE FOLLOWING SUBCOM WILL COMBINE  
$ 100% OF THE RESULTS FROM SUBCASE 1  
$ WITH 50% OF THE RESULTS FROM SUBCASE 2  
$  
SUBCOM 3  
SUBSEQ = 1.0,0.5  
TEMP(LOAD) = 20  
$  
BEGIN BULK  
$  
TEMPD,1, 100.  
TEMPD,10,150.  
TEMPD,15,250.  
$  
$ "TEMPD,20" IS CALCULATED AS FOLLOW:  
$ 100 + 1.0*(150-100) + 0.5*( 250 - 100 ) = 225.  
$  
TEMPD,20,225.  
$  
$ REST OF THE BULK DATA ENTRIES  
$  
ENDDATA
```

As a further example of thermal loads, again consider the umbrella tent shown in [Figure 6-19](#). The tent is subjected to thermal radiation from partial exposure to the sun. The ground is held at 40 degrees Fahrenheit and the air temperature is held at 50 degrees Fahrenheit. Can the tent withstand the thermal stresses imposed by solar radiation?

This problem involves conduction, convection, and radiation, which is beyond the scope of this user's guide, but the stress analysis of the resulting temperature profile is not. For further discussion on the subject of heat transfer, refer to the [MSC.Nastran Thermal Analysis User's Guide](#).

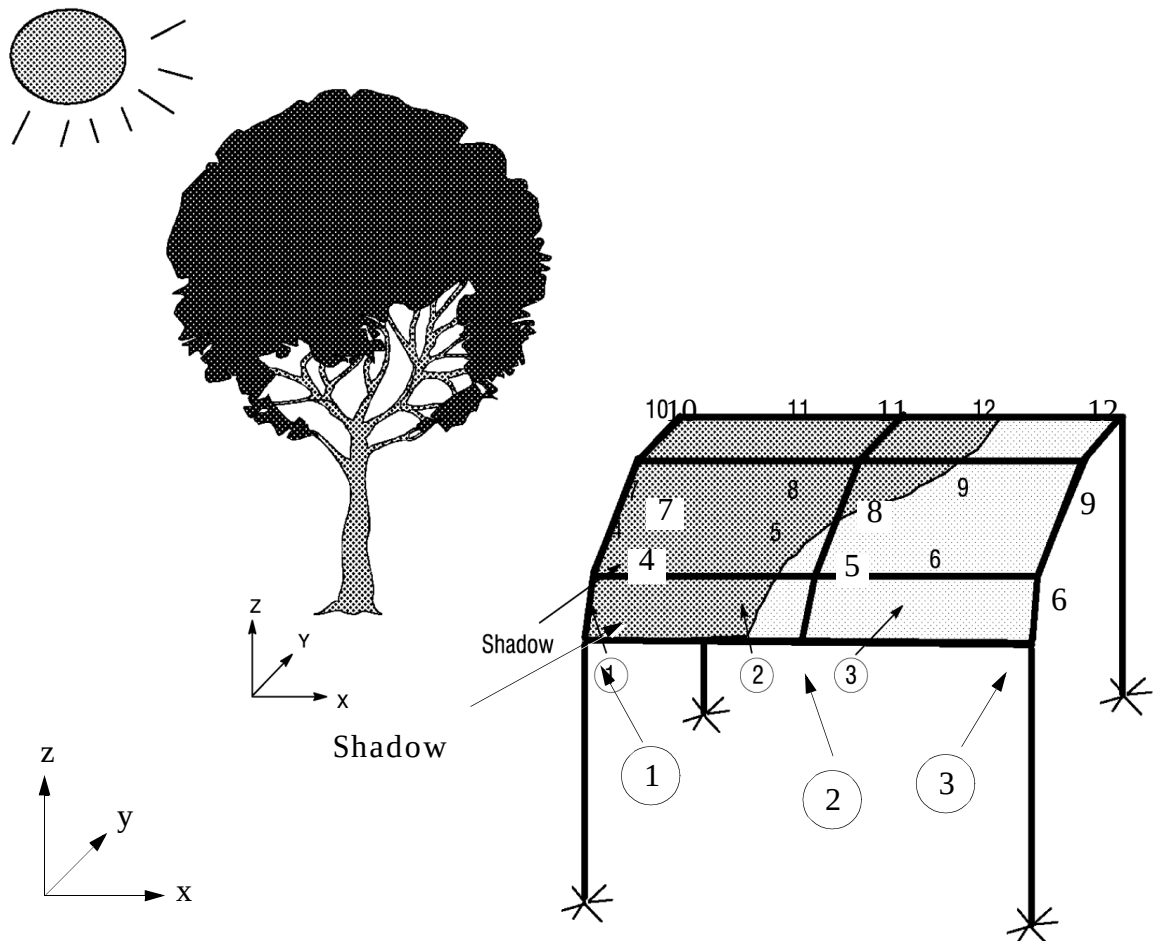


Figure 6-19 Umbrella Tent With Thermal Loads

To analyze this type of problem, you need to run the heat transfer Solution 153 to produce a temperature profile for all of the grid points in the model. A Solution 153 input file is generated for this model and is included on the delivery media with the name 'heat.dat'. One of the features of Solution 153 is the option of creating a temperature file that can be included directly into the solution 101 input file. The Solution 101 input file used for the stress analysis is shown in [Listing 6-9](#). The temperature file from the Solution 153 run is shown in [Listing 6-10](#). Note the use of the INCLUDE statement, which inserts the temperature input file at that location when the job is submitted. See "[Database Management](#)" on page 541 for a further discussion of the INCLUDE statement.

Listing 6-9 Umbrella Tent Thermal Analysis

```
$
$ FILENAME - THERMAL.DAT
$
ID LINEAR,THERMAL
SOL 101
TIME 10
CEND
TITLE = UMBRELLA TENT WITH THERMAL LOADING
$ DEFINE INITIAL TEMPERATURE
TEMP(INIT) = 2
$ DEFINE THERMAL LOADING
TEMP = 1
STRESS = ALL
SPC = 15
BEGIN BULK
$
INCLUDE 'heat.pch'
$
TEMPD  2    40.0
$
MAT1  1    10.E6    .3    2.59E-3 21.E-6
      25000. 25000. 15000.
MAT1  4     2.E6    .3    2.59E-5 21.E-6
      25000. 25000. 15000.
$
$  GRID, CBAR, CQUAD4, PBAR, PSHELL, AND SPC ENTRIES
$  REMOVED TO SPACE SPACE, THE FILE IS LOCATED ON THE
$  DELIVERY TAPE
$
ENDDATA
```

Listing 6-10 Temperature Distribution Generated with Solution 153

```

$
$FILENAME - HEAT.PCH
$
$TITLE  = TENT THERMAL ANALYSIS SUBJECTED TO RADIATION          1
$SUBTITLE=THE BASE TEMPERATURE AT 40 DEGREE F                  2
$LABEL  =                                                         3
$DISPLACEMENTS                                                    4
$REAL OUTPUT                                                       5
$SUBCASE ID =      1                                             6
TEMP*      1      1  4.011359E+01      7
TEMP*      1      2  5.999791E+01      8
TEMP*      1      3  1.194475E+02      9
TEMP*      1      4  4.001704E+01     10
TEMP*      1      5  4.705626E+01     11
TEMP*      1      6  6.816640E+01     12
TEMP*      1      7  4.000030E+01     13
TEMP*      1      8  4.004408E+01     14
TEMP*      1      9  6.814966E+01     15
TEMP*      1     10  4.005281E+01     16
TEMP*      1     11  4.012182E+01     17
TEMP*      1     12  1.193867E+02     18
TEMP*      1     99  4.000000E+01     19
TEMP*      1    101  7.500000E+01     20
TEMP*      1    103  7.500000E+01     21
TEMP*      1    110  7.500000E+01     22
TEMP*      1    112  7.500000E+01     23
TEMP*      1    999  0.000000E+00     24

```

The OLOAD and stress results for this model are shown in [Figure 6-20](#). The OLOAD output shows the applied loads generated by the imposed temperature. The thermal stresses for this example are quite small, so the structure should have no difficulty surviving the thermal loading. Of course, the model does not have enough detail to provide accurate results, but it does illustrate the procedure you can follow to perform a thermal stress analysis.

LOAD VECTOR

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	-6.966882E+03	-7.175819E+03	8.999034E+03	.0	.0	.0
2	G	-5.190949E+03	-1.430579E+04	-5.722314E+03	.0	.0	.0
3	G	1.215783E+04	-1.480121E+04	1.412900E+04	.0	.0	.0
4	G	-1.006932E+04	-2.448107E+02	2.870328E+03	.0	.0	.0
5	G	-3.668739E+03	1.792437E+03	5.722314E+03	.0	.0	.0
6	G	1.373806E+04	3.078795E+03	5.920484E+03	.0	.0	.0
7	G	-9.363078E+03	6.247873E+02	2.718337E+03	.0	.0	.0
8	G	-3.668739E+03	8.662622E+02	4.658834E+03	.0	.0	.0
9	G	1.303182E+04	-2.698817E+03	5.768493E+03	.0	.0	.0
10	G	-5.662744E+03	6.795843E+03	9.144758E+03	.0	.0	.0
11	G	-5.190948E+03	1.164709E+04	-4.658834E+03	.0	.0	.0
12	G	1.085369E+04	1.442123E+04	1.427472E+04	.0	.0	.0
101	G	.0	.0	-1.186936E+04	.0	.0	.0
103	G	.0	.0	-2.004948E+04	.0	.0	.0
110	G	.0	.0	-1.186309E+04	.0	.0	.0
112	G	.0	.0	-2.004321E+04	.0	.0	.0

STRESSES IN BAR ELEMENTS (CBAR)								
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
101	3.309835E+01	3.624385E+02	-3.624385E+02	-3.309835E+01	1.100334E+01	3.734419E+02	-3.514352E+02	6.6E+01
	-2.758225E+01	-3.055754E+02	3.055754E+02	2.758225E+01		3.165787E+02	-2.945720E+02	7.0E+01
102	-6.661619E+01	-3.129081E+02	3.129081E+02	6.661619E+01	8.939250E+01	4.023005E+02	-2.235155E+02	4.5E+01
	-1.640485E+02	4.599560E+02	-4.599560E+02	1.640485E+02		5.493485E+02	-3.705635E+02	6.6E+01
103	-2.039788E+02	1.703651E+02	-1.703651E+02	2.039788E+02	2.032520E+01	2.243040E+02	-1.836536E+02	4.5E+01
	3.399804E+02	-5.188448E+02	5.188448E+02	-3.399804E+02		5.391700E+02	-4.985197E+02	4.9E+01

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)								
ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
1	-2.500000E-02	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
	2.500000E-02	-2.478439E+00	-1.870431E+01	4.464139E+00	14.4109	-1.331339E+00	-1.985141E+01	1.922035E+01
2	-2.500000E-02	7.108396E+00	1.214202E+01	-4.387100E+00	-59.9211	1.468298E+01	4.567443E+00	1.301490E+01
	2.500000E-02	7.108396E+00	1.214202E+01	-4.387100E+00	-59.9211	1.468298E+01	4.567443E+00	1.301490E+01
3	-2.500000E-02	-3.498968E-01	-9.546574E+00	1.053041E-01	.6559	-3.486912E-01	-9.547779E+00	9.378296E+00
	2.500000E-02	-3.498968E-01	-9.546574E+00	1.053041E-01	.6559	-3.486912E-01	-9.547779E+00	9.378296E+00
4	-2.500000E-02	-1.165581E+01	2.807193E+00	-1.053041E-01	-89.5829	2.807959E+00	-1.165657E+01	1.328501E+01
	2.500000E-02	-1.165581E+01	2.807193E+00	-1.053041E-01	-89.5829	2.807959E+00	-1.165657E+01	1.328501E+01
5	-2.500000E-02	-7.158926E+00	-2.323350E+01	-4.590604E+00	-14.8667	-5.940316E+00	-2.445211E+01	2.208936E+01
	2.500000E-02	-7.158926E+00	-2.323350E+01	-4.590604E+00	-14.8667	-5.940316E+00	-2.445211E+01	2.208936E+01
6	-2.500000E-02	2.427620E+00	7.612546E+00	4.513566E+00	59.9359	1.022519E+01	-1.850235E-01	1.031895E+01
	2.500000E-02	2.427620E+00	7.612546E+00	4.513566E+00	59.9359	1.022519E+01	-1.850235E-01	1.031895E+01

Figure 6-20 Thermally Induced Loads and Stresses in the Tent Structure

Note: Only a partial listing of the CBAR element output is included.

Constraints

- Introduction to Constraints
- Single-Point Constraints
- Automatic Application of Single-Point Constraints (AUTOSPC)
- Enforced Displacements at Grid Points (SPCD, SPC)
- Multipoint Constraints
- Contact Problem Using Linear Static Analysis

7.1 Introduction to Constraints

As discussed in this chapter, performing a static analysis requires that all rigid body displacements be removed prior to solving the static equilibrium equations. This process involves specifying the appropriate boundary conditions for your model. Boundary conditions are imposed in the form of constraints on selected degrees of freedom on the model. Typically, several degrees of freedom (at least six) are constrained to ground using either SPC Bulk Data entries or the PS field of the GRID entry.

Besides single-point constraints, MSC.Nastran provides a method of creating linear constraint relationships between several degrees of freedom known as multiple-point constraints or MPCs. This chapter discusses both the single-point and multiple-point constraints.

The emphasis in this chapter is on applications, not on how the constraints are processed. If you want to understand how constraints are processed, you need to be familiar with the MSC.Nastran set notation and matrix operations (set notation is discussed in “[Matrix Operations](#)” on page 455. However, this level of understanding is not necessary for the majority of the applications.

7.2 Single-Point Constraints

A Single-Point Constraint (SPC) is a constraint that is applied to a single degree of freedom, which may be either a component of motion at a grid point or the displacement of a scalar point.

The primary applications for single-point constraints are as follows:

1. To tie a structure to ground.
2. To apply symmetric or antisymmetric boundary conditions by restraining the degrees of freedom that must have a zero value to satisfy symmetry or antisymmetry. Symmetry is discussed in “[Modeling Guidelines](#)” on page 311.
3. To remove degrees of freedom that are not used in the structural analysis (i.e., are not connected to any structural elements or otherwise joined to the structure).
4. To remove degrees of freedom that are very weakly coupled to the structure. This condition can occur, for example, to the rotations about the normal of a slightly curved shell. In this case, a judgment must be made whether to remove the degree of freedom using an SPC (in which case the structure may be over-constrained), or to leave it in the problem (in which case the stiffness matrix is nearly singular). A reasonable rule is to constrain the degree of freedom if its stiffness is less than 10^{-8} as large as the stiffness in another direction at the same grid point.

Under certain circumstances, a parameter called AUTOSPC causes an SPC to be applied automatically to components of motion with zero or very small stiffness. This feature is discussed in “[Automatic Application of Single-Point Constraints \(AUTOSPC\)](#)” on page 262.

When you apply a single-point constraint to remove a singularity, it is not required for the restrained component of motion to be aligned exactly with the singular direction of motion (however, it is highly recommended). Consider the pair of colinear pin-connected rods, shown in [Figure 7-1](#), that permit unrestrained motion at point G in any direction perpendicular to the axis of the rods.

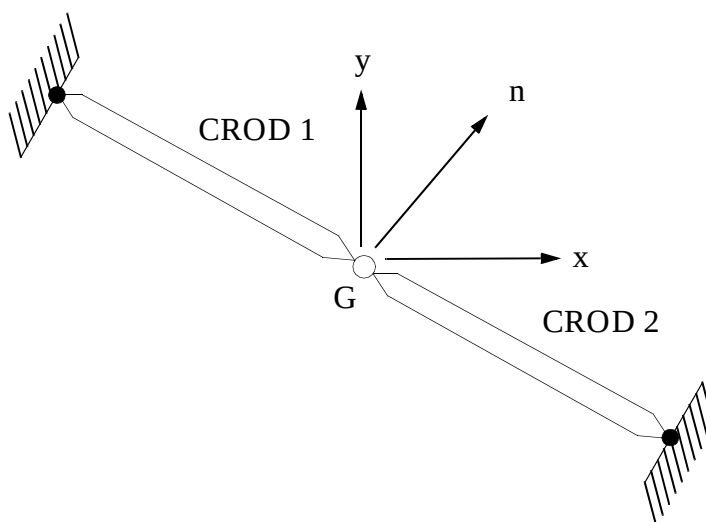


Figure 7-1 Pin-Connected Rods

After SPCs are applied to the translation in the z-direction and to all three components of rotation, unrestrained motion is still possible in the n-direction. An SPC in either the x- or the y-direction removes the remaining singularity without disturbing the axial forces in the rods (in this special case). Note, however, that the motion at point G has a zero component in the restrained direction, which may produce incorrect results for the displacements. For this reason, it is recommended that a local coordinate system be selected with the coordinate axes aligned with the singular components of motion. An example of this is shown in “**Enforced Displacements at Grid Points (SPCD, SPC)**” on page 266.

The SPC and SPC1 Bulk Data entries are recommended if you need to apply different sets of boundary conditions in different subcases. The constraints specified on the SPC and SPC1 entries belong to sets identified by set identification numbers (SIDs) that must be selected in the Case Control Section to be used. As a further convenience, the sets can be merged into combined sets by the SPCADD Bulk Data entry, which can be specified in the Case Control Section.

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
SID	Identification number of the single-point constraint set.
Gi	Grid or scalar point identification number.
Ci	Component number.
Di	Value of enforced displacement for all degrees of freedom designated by Gi and Ci.

The format of the SPC1 entry is as follows:

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

Field	Contents
SID	Identification number of the single-point constraint set.
C	Component numbers.
Gi	Grid or scalar point identification numbers.

The SPC entry can also be used to specify a nonzero displacement constraint. The values specified in fields 5 and 8 are enforced displacements for the components specified in fields 4 and 7, respectively. When using an enforced constraint within multiple subcases, it is more efficient to use the SPCD entry.

As an alternative, single-point constraints may be applied directly on the GRID and GRDSET Bulk Data entries. These SPCs are permanent, i.e., they are not selectable in the Case Control Section. Their primary use, in addition to applying permanent supports, is to remove unconnected or very weakly connected degrees of freedom. Such degrees of freedom can also be constrained automatically using the parameter AUTOSPC, which is described in “[Automatic Application of Single-Point Constraints \(AUTOSPC\)](#)” on page 262.

7.3 Automatic Application of Single-Point Constraints (AUTOSPC)

Degrees of freedom that are either unconnected or very weakly coupled to the finite element model can be removed automatically with the PARAM,AUTOSPC,YES in the Bulk Data or Case Control Section. In Solution 101, the Grid Point Singularity Processor (GPSP) module is invoked to identify potential singularities. When the AUTOSPC parameter is set to YES, GPSP constrains the degrees of freedom associated with the obvious singularities. GPSP identifies local singularities in the stiffness matrix after processing all of the SPC and MPC constraints in your model (this is different for Solution 24).

The identification of local grid point singularities begins with the examination of the $3 \cdot 3$ matrices representing the three translations and three rotations of each grid point. For each $3 \cdot 3$ matrix, an eigensolution is performed to obtain the principal values (eigenvalues) and the principal directions (eigenvectors). The principal values are normalized to obtain a measure R_i as shown in [Eq. 7-1](#):

$$\begin{aligned} R_i &= \frac{K_i}{|K_{MAX}|} && \text{if } |K_{MAX}| > 0 \\ &&& i = 1, 2, 3 \\ R_i &= 0.0 && \text{if } |K_{MAX}| = 0 \end{aligned} \tag{Eq. 7-1}$$

where:

K_i = the diagonal stiffness term corresponding to the i -th degree of freedom

K_{MAX} = the maximum principal stiffness

Any principal value for which $|R_i| < \text{EPZERO}$ represents a singularity. The principal direction corresponding to the singularity is examined, and the component with the largest ratio is identified as the failed direction for the grid point. Any degrees of freedom that you specify as belonging to the a -set are not constrained (a -sets are discussed in “[Matrix Operations](#)” on page 455).

Always review the Grid Point Singularity Table to ensure that the degrees of freedom that are automatically constrained are the degrees of freedom that you intend to constrain. There are two potential problems that can occur by not verifying the AUTOSPC constrained degrees of freedom in your model. One possible problem is when a grid point or components of a grid point are inadvertently left unattached in your model when you wanted them to be attached. The GPSP module identifies singular degrees of freedom and constrains them. It may hide a modeling error, and unless you reviewed the grid point singularity table, you may not know it. A common example of a modeling error that is hidden by the GPSP module is forgetting to specify bending stiffness (via MID2 on the PSHELL entry).

The second reason you need to review the Grid Point Singularity Table is to ensure that the constrained degrees of freedom make sense. It is possible that the GPSP module can constrain a degree of freedom that may result in wrong answers.

The output includes a grid point singularity table with the following format:

1. The TYPE specifies whether the failed degree of freedom is a component of a grid point, which is indicated by a “G”, or that of a scalar point, which is indicated by an “S”.
2. The component of motion that failed. Constraints are applied in the global coordinate direction closest to the weakest direction as defined above.
3. The threshold at which the DOFs are constrained may be changed using the parameter EPZERO.
4. The stiffness ratio is the ratio of the stiffness in the weakest direction to the stiffness in the strongest direction at the grid point (considering all possible directions rather than just the coordinate directions). Translational and rotational stiffness are considered separately. The default value for the failure criterion is 10^{-8} , which may be changed by PARAM,EPZERO.
5. If possible, the failed degrees of freedom are placed in the sb-set if AUTOSPC is set to YES (the default for Solution 101). If AUTOSPC is set to NO, the singularity is identified but not constrained and is not placed in the sb-set.
6. An asterisk (*) is printed at the end of the line and indicates that action was taken by MSC.Nastran.

To illustrate how failing to inspect the GPSP output can lead to a potential problem, consider the single CBAR element example shown in [Figure 7-2](#).

Note: Local coordinate system 1 is rotated -45° about the X_b axis.

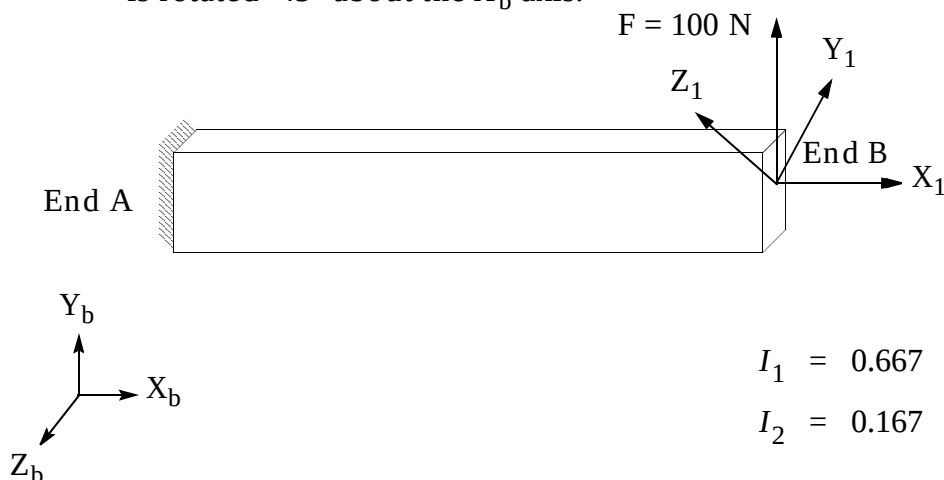


Figure 7-2 Single Element with Vertical Load

The CBAR element is fully constrained at end A and is loaded with a 100 N force in the Y_b - direction at end B. The input file for this model is shown in [Listing 7-1](#). Both grid points are defined in the basic coordinate system (the CP field for both grid points is blank). The

displacement coordinate system for grid point 1 is the basic system, but the displacement coordinate system for grid point 2 is in the local coordinate system 1. It is typical to define a local coordinate system when you need a displacement component in a specific direction.

Listing 7-1 Single Element with a Vertical Load

```
$
$ FILENAME - BAR3A.DAT
$
ID   LINEAR,BAR3A
SOL  101
TIME  5
CEND
TITLE = BAR WITH I1 AND I2
SPC = 1
LOAD = 1
FORCE = ALL
DISP = ALL
BEGIN BULK
PARAM POST  0
PARAM AUTOSPC YES
CORD2R 1    0    0.0    0.0    0.0    0.0    .7071068.7071068+
      1.    0.0    0.0
$
GRID  1      0.0    0.0    0.0
GRID  2      10.    0.0    0.0    1
$
CBAR  1    1    1    2    1.    1.    0.0
$
SPC   1    1    123456 0.0
$
PBAR  1    1    1.    .667    .167
FORCE 1    2      100.    0.    1.    0.
MAT1  1    1.+7      .3
ENDDATA
```

For this example, the displacement should be exclusively in the Y_b -direction. Since the displacement is computed and output in the CD coordinate system, you expect that the displacement components T2 (the Y_1 -direction) and T3 (the Z_1 -direction) for grid point 2 are the same for this model. The displacement and force results shown in [Figure 7-3](#) confirm this assumption.

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	NEW USET UNION	EXCLUSIVE	UNION	EXCLUSIVE	UNION
2	G	4	0.00E+00	B	F	SB	S		*
DISPLACEMENT VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	.0	.0		
2	G	.0	3.533767E-03	3.533767E-03	.0	-5.300650E-04	5.300650E-04		
FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT PLANE 1	END-A PLANE 2	BEND-MOMENT PLANE 1	END-B PLANE 2	- SHEAR - PLANE 1	PLANE 2	AXIAL FORCE	TORQUE	
1	1.000000E+03	1.761739E-14	4.263256E-13	-4.944563E-14	1.000000E+02	6.706302E-15	.0	.0	

Figure 7-3 Force and Displacement at Grid Point 2 with a Nonzero I2

However, suppose that when you construct the model, you know that the deflection is in plane 1 of the element only, so you leave the I2 field blank. The resulting grid point singularity table, element forces, and displacements are shown in [Figure 7-4](#)

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	NEW USET UNION	EXCLUSIVE	UNION	EXCLUSIVE	UNION
2	G	2	0.00E+00	B	F	SB	S		*
2	G	4	0.00E+00	B	F	SB	S		*
2	G	5	0.00E+00	B	F	SB	S		*
DISPLACEMENT VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	.0	.0		
2	G	.0	.0	7.067534E-03	.0	.0	1.060130E-03		
FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT PLANE 1	END-A PLANE 2	BEND-MOMENT PLANE 1	END-B PLANE 2	- SHEAR - PLANE 1	PLANE 2	AXIAL FORCE	TORQUE	
1	1.000000E+03	.0	4.263256E-13	.0	1.000000E+02	.0	.0	.0	

Figure 7-4 Force and Displacement at Grid Point 2 with a Zero I2

As can be seen, the GPSP module identified a singularity that is caused by the missing bending stiffness of the element. However, the constraint is placed on the Y_1 and θ_{y_1} the degrees of freedom, not the Y_b and θ_{y_b} degrees of freedom. The force in the CBAR element is correct, but the displacements are wrong. You made a common assumption that for a symmetric structure with in-plane loads, the out-of-plane properties are not needed. However, the choice of the output coordinate system and parameter AUTOSPC resulted in a modeling error. If you chose the output coordinate system for grid point 2 as the basic system, the displacements would be correct. Also, for this model, if parameter AUTOSPC is set to NO, the job will fail.

If this had been a production size model, the error would not have been so obvious and may well have gone undetected. Always look at the Grid Point Singularity Table results to make sure that the constrained degrees of freedom are acceptable.

7.4 Enforced Displacements at Grid Points (SPCD, SPC)

There are two methods available to you for specifying an enforced displacement at a component. The first method is to enter the value of the enforced displacement directly on an SPC entry.

The alternate method to enforce a displacement at a component is to use the SPCD Bulk Data entry. The SPCD entry is actually a force, not a constraint, but it is used in conjunction with an SPC entry to enforce the displacement. When you use an SPCD entry, internal forces are computed that are applied to the structure to produce the desired enforced displacements.

The format of the SPCD entry is as follows:

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

Field	Contents
SID	Identification number of a static load set.
Gi	Grid or scalar point identification number.
Ci	Component numbers.
Di	Value of enforced displacement for at Gi and Ci.

Grid points with an enforced displacement using the SPCD entry must also appear on an SPC or an SPC1 Bulk Data entry.

The SPCD method of enforcing a nonzero constraint is more efficient than using an SPC entry alone when you are using multiple subcases that specify different constraint conditions. Note also that when you use an SPCD entry, the displacement values entered on the SPC entry are ignored-only the SPCD values are used.

As an example of enforced displacement, consider the flat plate shown in [Figure 7-5](#). The plate is modeled with 2500 CQUAD4 elements.

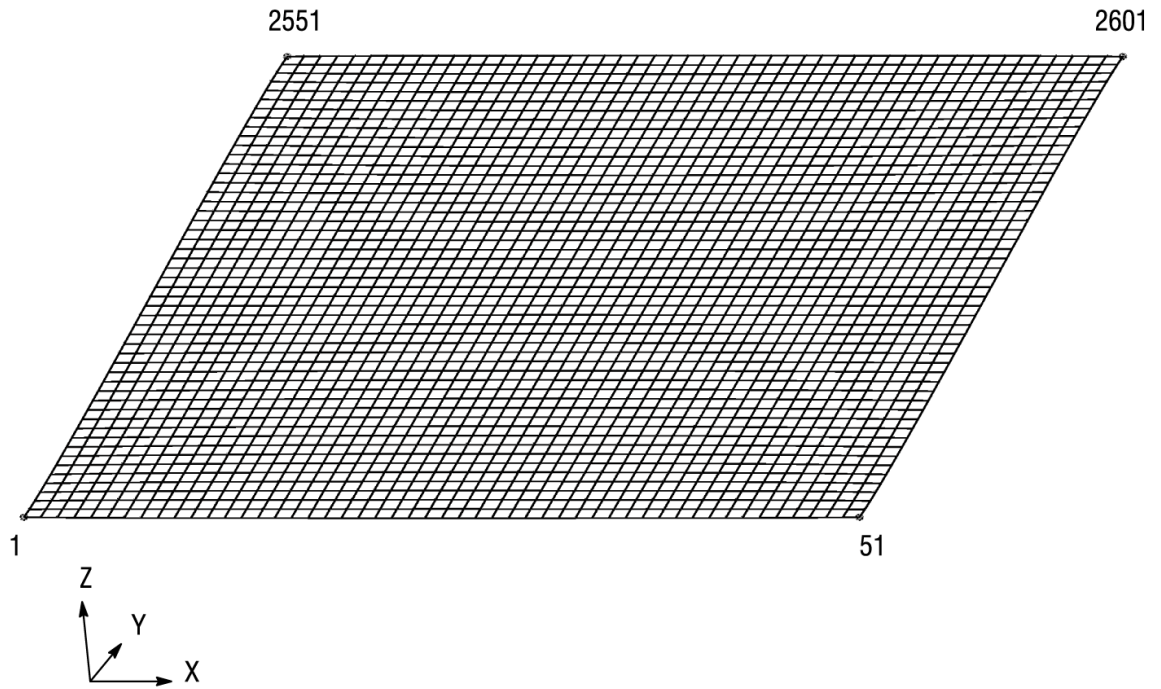


Figure 7-5 Flat Plate with Various Corner Constraints

The purpose of this example is to compare the efficiency of using the SPC /SPCD combination to that of using the SPC entry only. For the plate model, four runs are made, each having two subcases with different constraint conditions. For the input files “const1.dat” and “const2.dat”, the locations of the constraints are the same for both subcases, but the magnitudes of the constraints change. The file “const1.dat” uses a different SPC for both subcases. The file “const2.dat” specifies the same SPC for both subcases, but it has different SPCD entries.

The input files “const3.dat” and “const4.dat” are similar to the first two; however, the locations of the constraints in the two subcases are different. **Table 7-1** summarizes the constraints and CPU time for all four runs.

Table 7-1 Comparison of the SPC Versus the SPC/SPCD Combination

	Location of Enforced Displacement							
	const1.dat		const2.dat		const3.dat		const4.dat	
	SPC Only		SPC / SPCD		SPC Only		SPC / SPCD	
Corner Disp (in)	Subcase 1	Subcase 2	Subcase 1	Subcase 2	Subcase 1	Subcase 2	Subcase 1	Subcase 2
Grid 1	0	0	0	0	-	-	-	-
Grid 51	0	5	0	5	-	0	-	0
Grid 2551	5	5	5	5	0	5	0	5
Grid 2601	5	0	5	0	5	-	5	-
CPU Time*	7.3 (sec)		5.1 (sec)		7.3 (sec)		7.3 (sec)	
- free to move * CPU time on Windows 2000								

As can be seen, the SPC / SPCD combination is more efficient for both modeling situations. It is, however, most advantageous to use when the magnitudes but not the locations of the constraints change. The larger improvement seen by the magnitude only change is reasonable because only one SPC needs to be processed; the multiple boundary conditions are represented as multiple loads. In general, whenever you have multiple subcases with different constraint conditions, the SPC / SPCD combination is the preferred method for efficiency.

As mentioned previously, the SPCD is a force entry, not a constraint. Therefore, the SPCD is called out in the Case Control Section with the LOAD command. The input file for “const2.dat” is shown in [Listing 7-2](#).

Listing 7-2 The SPC/SPCD Combination

```
$
$ FILENAME - CONST2.DAT
$
ID   LINEAR,CONST2
SOL   101
TIME  200
CEND
TITLE = ENFORCED DISPLACEMENTS USING SPC1 ONLY
DISP(PLOT) = ALL
SPC = 1
SUBCASE 1
    LOAD = 11
SUBCASE 2
    LOAD = 12
BEGIN BULK
PARAM POST 0
$
SPC1   1      123456 1      51      2551      2601
$
SPCD   11      1      3      0.0
SPCD   11      51      3      0.0
SPCD   11      2551     3      5.0
SPCD   11      2601     3      5.0
$
SPCD   12      1      3      0.0
SPCD   12      51      3      5.0
SPCD   12      2551     3      5.0
SPCD   12      2601     3      0.0
$
INCLUDE 'const.bulk'
$
ENDDATA
```

As a final comment, if the SPCD entry is the only loading in the model, a warning message is issued alerting you to the fact that no loading is specified for your model. If you intended for the SPCD entry to be the only loading condition, you may safely ignore this warning message.

7.5 Multipoint Constraints

In MSC.Nastran, the term Multiple-Point Constraint (MPC) is used to describe an imposed linear relationship between two or more degrees of freedom that are expressed in the form

$$\sum_j R_j u_j = 0$$

Eq. 7-2

where:

u_j = any degree of freedom defined by a grid point or an S point

R_j = user-defined scale factor

- Multipoint constraints have many important practical applications and can be used to
- Define the relative motion between two grid points as a degree of freedom.
 - Define the average of several motions as a degree of freedom.
 - Supply hinges or sliding joints between parts of a structure.
 - Join dissimilar elements, for example, to join elements with rotational degrees of freedom to elements that have only translational degrees of freedom (e.g., to join shell elements to solid elements).
 - Obtain the resultant of forces on a structure or on part of a structure.
 - Distribute a force to several points in a structure. This is particularly useful if the force is an unknown force, for example, the force required to compress a fluid.
 - Join elements with noncoincident grid points, for example, to change mesh size within a structure.
 - Replace extremely stiff structural members with rigid connections. This modeling technique should only be used when necessary to improve the numerical conditioning of the stiffness matrix. MPCs can be used for this application, but they are not recommended. The preferred method is to use an R-type element because this element is simpler, and therefore less prone to error. R-type elements are discussed in “**R-Type Elements**” on page 285.
 - Define a component of motion at a grid point that is not aligned with the axes of the global (i.e., local) coordinate system, for example, to enable the user to supply a constraint in such a direction with an additional SPC entry.

The format of the MPC entry is as follows:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

Field	Contents
SID	Set identification number.
Gj	Identification number of grid or scalar point.
Cj	Component number.
Aj	Coefficient.

As an illustration of the use of the MPC entry, consider the following example.

Consider the model shown in **Figure 7-6**. The goal is to determine the relative motion between component 1 of grid point 1 and component 1 of grid point 2 using an MPC entry and a scalar point.

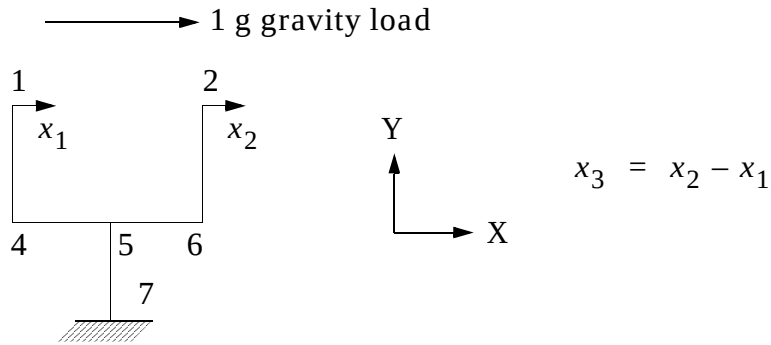


Figure 7-6 Using an MPC to Compute Relative Displacement

Let the components of motion at the two grid points whose difference is desired be designated as x_1 and x_2 as given by **Eq. 7-3**. The difference is represented by scalar point 3 defined on an SPOINT entry.

$$x_3 = x_2 - x_1 \tag{Eq. 7-3}$$

The equation of constraint can be in any of the forms shown in **Eq. 7-4**.

$$\begin{aligned} x_3 - x_2 + x_1 &= 0 & (a) \\ x_2 - x_1 - x_3 &= 0 & (b) \\ x_1 - x_2 + x_3 &= 0 & (c) \end{aligned} \tag{Eq. 7-4}$$

Listing 7-3 shows the implementation of **Eq. 7-4(a)** to obtain the relative displacement. The first degree of freedom on the MPC entry is the dependent degree of freedom—which in this case is scalar point 3. It can just as well be made a component of motion at a grid point defined on a GRID entry. Grid points 1 and 2 are the independent degrees of freedom. Note that the MPC set ID (SID) is selected in the Case Control Section.

The first listed degree of freedom on the MPC entry (i.e., x_3 in this example) is designated as dependent and is placed in the *m*-set. The degrees of freedom in the *m*-set are eliminated at an early stage in the solution process. If you want the relative motion to appear as an independent degree of freedom, express the multipoint constraint in either the second or third forms of **Eq. 7-4**.

Listing 7-3 Relative Displacement Using an MPC Entry

```
$
$ FILENAME - RELDIFF.DAT
$
ID  LINEAR,RELDIFF
SOL  101
TIME  5
CEND
TITLE = RELATIVE DISPLACEMENT
SET 1 = 1,2,3
DISP = 1
MPC = 1
LOAD = 1
SPC = 1
BEGIN BULK
$
$ ONLY THE GRIDS, SPOINT AND MPC ENTRIES ARE SHOWN
$
SPOINT 3
MPC  1    3    0    1.    2    1    -1.
      1    1    1.
$
$
GRID  1      -10.  20.  0.0
GRID  2       10.  20.  0.0
GRID  4      -10.  10.  0.0
GRID  5       0.0  10.  0.0
GRID  6       10.  10.  0.0
GRID  7       0.0  0.0  0.0
$
ENDDATA
```

There are three important rules that must be observed when using MPCs (or R-type elements). These rules are as follows:

1. A member of the *m*-set cannot be a member of any exclusive subset of the *g*-set, for example, it cannot be a degree of freedom that is removed by single-point constraints. MSC.Nastran displacement sets are discussed in “[Matrix Operations](#)” on page 455.
2. A given degree of freedom cannot be designated as a member of the *m*-set (dependent) more than once. Thus, it cannot appear as the first listed component on more than one MPC entry.

3. You must avoid redundant constraints. This redundancy occurs, for example, if two of the forms in [Eq. 7-4](#) are supplied simultaneously on MPC entries. The technical difficulty that occurs is that the matrix of constraint coefficients for the m -set is singular so that the dependent degrees of freedom cannot be evaluated in terms of the independent ones. This problem also occurs when R-type elements are interconnected in a closed loop.

An important aspect of multipoint constraints is that they simultaneously supply relationships between components of motion as well as relationships between the forces of constraint. This can be demonstrated by using the principle that the forces acting on a rigid body can do no work. Consider, for example, the equation of constraint among three degrees of freedom shown in [Eq. 7-5](#).

$$R_1 u_1 + R_2 u_2 + R_3 u_3 = 0 \quad \text{Eq. 7-5}$$

The work done by the corresponding forces of constraint is given by [Eq. 7-6](#).

$$W = 1/2(F_1 u_1 + F_2 u_2 + F_3 u_3) = 0 \quad \text{Eq. 7-6}$$

In order for both [Eq. 7-5](#) and [Eq. 7-6](#) to be valid, it is necessary that the following equation also be valid.

$$\frac{F_1}{R_1} = \frac{F_2}{R_2} = \frac{F_3}{R_3} \quad \text{Eq. 7-7}$$

Thus, the forces of constraint are proportional to the coefficients in the equations of constraint. As an example, consider the use of a constraint to evaluate the average of four motions, i.e.,

$$u_5 = 1/4(u_1 + u_2 + u_3 + u_4) \quad \text{Eq. 7-8}$$

From [Eq. 7-8](#), the forces of constraint are given by:

$$F_1 = F_2 = F_3 = F_4 = 1/4 F_5 \quad \text{Eq. 7-9}$$

It is clear, from [Eq. 7-9](#), that a force applied to u_5 is distributed equally to u_1 , u_2 , u_3 , and u_4 .

In general, any constraint can be viewed either as a means to constrain a component of displacement or as a means to distribute the net force acting on the constrained component of displacement (i.e., the resultant of applied force and internal forces).

7.6 Contact Problem Using Linear Static Analysis

Introduction

GAP elements have traditionally been used to model contact-type problems. However, the GAP element requires the use of a nonlinear solution sequences--specifically, SOL 106 or 129--and estimation of the gap stiffness. Certain types of contact problems, however, may be modeled in linear static analysis using SOL 101. In contrast to the gap element available for nonlinear analysis (SOL 106 and SOL 129), the “linear” gap capabilities in SOL 101 do not require estimation of gap stiffness. It uses an iterative constraint type approach. The constraints are applied to grid points or spoints. The constraint ensures that:

1. The displacement (UR) cannot be negative. This is to ensure that there is no penetration. Therefore, the chosen degree-of-freedom must be perpendicular to the contact surface and positive in the opening direction.
2. The force of constraint (QR) cannot be negative. This is to ensure that there is no tension.

The constraints are satisfied by an iterative technique that is built into SOL 101. The iterative process starts with a random vector. This random vector assumes certain grids to be in contact and other grids to be in an open state. A solution is obtained when all the gap constraints are satisfied, i.e., there's no penetration and no tension forces. If a limit cycle (return to a previous state) is detected during the iterations, a new random start vector is tried.

This approach provides an alternate method to the use of GAP elements in SOL 106. Some experiments have shown that the cost of analysis will be about the same. The advantage is that you do not have to learn how to calculate the GAP stiffness nor how to control SOL 106. Multiple load conditions are allowed, and each will be solved separately.

If the constraint is between a finite element model and a fixed boundary, then arrange to have one of the degrees-of-freedom at the boundary grid points represent motion perpendicular to the boundary. A positive displacement represents motion away from the boundary. If, on the other hand, the constraint represents relative motion between two bodies, MPC equations are needed to define a relative motion degree-of-freedom, which is then constrained to have a non-negative displacement.

Input

A SUPORT entry along with PARAM,CDITER are required. These two items plus some optional parameters are described below.

SUPPORT entry	Selects constrained degrees-of-freedom. These points must be in the a-set of the residual structure. This means they must not be dependent in an MPC equation, constrained by SPC, partitioned by OMIT, or in an upstream superelement.
PARAM,CDITER,x	CDITER Constraints will be applied if CDITER is greater than zero. The value x is the maximum number of iterations allowed. (Default = 0).
PARAM,CDPRT	Controls the printing of constraint violations during the iterations. The sparse matrix printer prints UR (negative displacements) and QR (negative forces of constraint) for constrained degrees-of-freedom. (Default = 'YES')
DMIG,CDSHUT	Optional DMIG input of the vector defining the state of the constrained degrees-of-freedom. A one indicates closed and a zero indicates open. If DMIG,CDSHUT is not provided, all SUPPORT degrees of freedom will assumed to be closed initially.
PARAM,CDPCH	Controls the PUNCH output of DMIG,CDSHUT records for the final state of the constrained degrees-of-freedom. (Default = 'NO'). This output will be created in a separate .pch file.

Output

The output is similar to standard SOL 101 output. Forces for closed degrees-of-freedom will appear in the SPCFORCE output. In addition, optional diagnostic information for the iterations can be requested (PARAM,CDPRT). A final state vector may also be sent to a .pch file (PARAM,CDPCH).

The .f06 file should be examined to ensure that the iterations have converged, since the results of the last iteration will be output. The last iteration should have zero changes. A typical converged output is shown below. More detailed explanation are included in the actual examples.

1	DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1	CD_1	SEPTEMBER	6, 2001	MSC.NASTRAN	4/ 9/01	PAGE	10
0								
0	POINT	QRI	VALUE	POINT	VALUE	POINT	VALUE	POINT
	COLUMN	1						
	5 T2	-2.37141E-02						
***	END ITERATION	3, CHANGES	1					
***	END ITERATION	4, CHANGES	0					
***	USER INFORMATION MESSAGE 9097 (CSTRDISP)							
***	CONSTRAINED DISPLACEMENT ITERATIONS CONVERGED	LOAD CASE =	1					
1	DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1	CD_1	SEPTEMBER	6, 2001	MSC.NASTRAN	4/ 9/01	PAGE	11

Limitations

The following is a list of limitations for the linear gap elements.

- The only nonlinearity allowed is the constrained displacements.

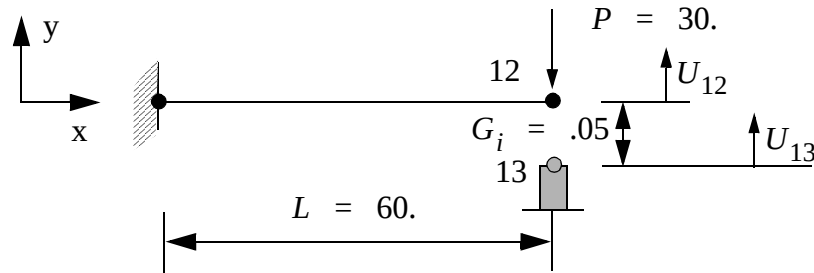
- There is no sliding friction.
- Free bodies--using the SUPORT entry to define rigid body modes--cannot be used in the same model with linear gap elements. The parameters INREL and CDITER are mutually exclusive. A fatal message is issued if both parameters are present.
- No constraint changes are allowed between subcases.
- There is no guarantee that the solution will converge or that all systems will follow the same path.

Examples

Two examples are shown below to illustrate the use of the linear GAP element.

EXAMPLE 1(cd_0.dat)

This model consists of a cantilever beam with a stopper and a vertical load of 30 lbs applied at the free-end. There is a clearance of 0.05 inch between the free-end and the stopper. The purpose of this problem is to find out if 30 lbs is sufficient to cause the free-end to hit the stopper.



The first step in solving this problem is to define a variable (S_{51}) to monitor the opening between the free-end (grid point 12) and the stopper (grid point 13). The following equation

$$S_{51} = U_{12} + G_i - U_{13} \quad \text{Eq. 7-10}$$

where:

U_{12} = the displacement at grid point 12

U_{13} = the displacement at grid point 13

G_i = the initial gap opening

S_{51} = the relative distance between grids points 12 and 13 including the effect of the initial opening

accomplishes this goal. Grid point 13 can be removed from the equation since the stopper does not move. [Eq. 7-10](#) can be simplified and recast as

$$-U_{12} + S_{51} - G_{101} = 0 \quad \text{Eq. 7-11}$$

so that an MPC equation can be written. Two scalar points are introduced in this case: Scalar points 101 for G_{101} (G_i) and 51 for S_{51} . Note that neither S_{51} nor G_{101} can be defined as dependent since they already belong to the r-set and s-set, respectively. A copy of the input file is shown in [Listing 7-4](#).

Listing 7-4 Input file for Cantilever Beam with Stopper.

```
$
$ cd_0.dat
$
sol 101
cend
Title = Gap elements, Cantilever beam
mpc = 77
spc = 88
load = 300
disp = all
spcf = all
mpcf = all
$
begin bulk
$
grid 11      0.    0.    0.
grid 12      60.   0.    0.
cbar 15      1     11    12    0.    1.    0.
pbar 1       2     1.    1.    1.    1.
mat1 2       3.e7    0.3
force 300    12     -30.   0.    1.    0.
spoint      51    101
support     51     0
spc 88      11    123456
spc 88      101    0     .05
mpc 77      12     2     -1.0  51     0     1.
           101    0     -1.0
param cditer 10
param cdpert yes
$
enddata
```

[Eq. 7-11](#) is represented by MPC,77. Scalar point 51 must also be placed on the SUPORT entry. In addition, an enforced displacement of 0.05 must also be imposed on scalar point 101. PARAM,CDITER,10 requests that a maximum of 10 iterations be performed for this analysis. If more than 10 iterations are required for this problem, the results at the end of the 10th iteration will be printed. PARAM,CDPRT,YES is optional in this case since it is the default.

An abridged output is shown in [Eq. 7-7](#). The large non-zero values for epsilon and strain energy for the SUPORT degree of freedom can be safely ignored as we are not performing inertia relief analysis. The solution converges in 1 iteration as indicated by the following output.

The gap is in contact as indicated by the following scalar point 51 output.

0										
0										
	POINT	SHUT VALUE	POINT	VALUE	POINT	VALUE	POINT	VALUE	POINT	VALUE
COLUMN		1								
	51	S		1.000000E+00						

The displacement T2 at grid point 12 is -0.05, which implies that the cantilever beam hits the stopper. The load required to close the gap is:

$$P_g = \frac{3 \cdot EI \cdot U_{12}}{L^3} = \frac{3(3E7)(1)(0.05)}{60 \cdot 60 \cdot 60} = 20.833 \quad \text{Eq. 7-12}$$

which is what the spcforce output at grid point 11 shows. The rest of the applied load 9.167 (30-20.833) is reacted by the stopper as indicated by spcforce at grid point 51 and, similarly, the mpcforce at grid point 12.

```

0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
    DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
    FOR DATA BLOCK KLR
    SUPPORT PT.NO.      EPSILON      STRAIN      ENERGY      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH
    ASTERISKS
        1      2.5000000E-01      2.0833333E+02      *****
*** 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      0.0000000E+00      1.5625000E+00
^^^ USER INFORMATION MESSAGE 9077 (CSTRDISP)
^^^ CONSTRAINED DISPLACEMENT ITERATIONS
^^^ CONSTRAINED DISPLACEMENT  DIAGNOSTICS, UR= NEGATIVE DISPLACEMENT, QR= NEGATIVE FORCE
^^^ END ITERATION      1, CHANGES      0
^^^ USER INFORMATION MESSAGE 9097 (CSTRDISP)
^^^ CONSTRAINED DISPLACEMENT ITERATIONS CONVERGED      LOAD CASE =      1
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
7

0
0
    SHUT
    POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT
    VALUE

COLUMN      1
    51 S 1.000000E+00
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
8
0
.
.

DISPLACEMENT VECTOR

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    11      G      0.0      0.0      0.0      0.0      0.0      0.0
    12      G      0.0      -5.000000E-02      0.0      0.0      0.0      -1.250000E-03
    51      S      0.0
   101      S      5.000000E-02
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
11
0

FORCES OF SINGLE - POINT CONSTRAINT

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    11      G      0.0      2.083333E+01      0.0      0.0      0.0      1.250000E+03
    51      S      9.166666E+00
   101      S      -9.166666E+00
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
13
0

FORCES OF MULTIPOINT CONSTRAINT

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    12      G      0.0      9.166666E+00      0.0      0.0      0.0      0.0
    51      S      -9.166666E+00
   101      S      9.166666E+00

```

Figure 7-7 Abridged Output of Cantilever Beam with Stopper.

EXAMPLE 2(cd_1.dat)

The second example is a thick pad supported on a rigid base with a vertical point load applied at the center. Due to symmetry, a half model is used by applying the appropriate boundary condition at the center. The problem is to determine where the lift off occurs along the base (grid points 1 through 7.) An abridged input file for this example is shown in [Listing 7-5](#). A planar problem ([Figure 7-8a](#)) is considered in this case by constraining

3456 on the GRDSET entry. SPC1,200 applies the symmetric boundary condition at the center line of the structure. The potential contact points are specified on the seven SUPORT entries--one for each grid point at the bottom. In this case, since the contact is between a structure and a rigid surface, and there is no initial gap, the use of the MPC is not required. A maximum of up to 20 iterations (PARAM,CDITER,20) are allowed for this case. Furthermore, an initial guess of where the contact may occur is provided (DMIG,SHUT.) In this case, grid points 1 and 7 are assumed to be shut and grid points 2 through 6 are assumed to be opened initially. PARAM,CDPCH,YES requests that the final open / shut configuration be created in a separate punch file. PARAM,CDPRT,YES requests that the printing of the constraint violations during the iterations, which is the default.

Listing 7-5 Input File for Pad on Rigid Foundation.

```
$ cd_1.dat
$
SOL 101
CEND
TITLE = DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1 CD_1
SPC = 200
LOAD = 300
DISPL = ALL
OLOAD = ALL
SPCFO = ALL
BEGIN BULK
$
$ SYMMETRIC BOUNDARY AT X=0, DOWNWARD LOAD AT TOP CENTER
$
SPC1 200 1 1 11 21 31 41
GRDSET 3456
FORCE 300 41 1. -1.
$
$ CONSTRAIN DISPLACEMENTS ON BOTTOM EDGE, OPTIONAL CDSHUT INPUT
$
$2345678 2345678
SUPORT 1 2
SUPORT 2 2
SUPORT 3 2
SUPORT 4 2
SUPORT 5 2
SUPORT 6 2
SUPORT 7 2
$
PARAM CDITER 20
PARAM CDPRT YES
PARAM CDPCH YES
$
DMIG CDSHUT 0 9 1 0 1
DMIG CDSHUT 1 0 1 2 1.
7 2 1.
$
.
```

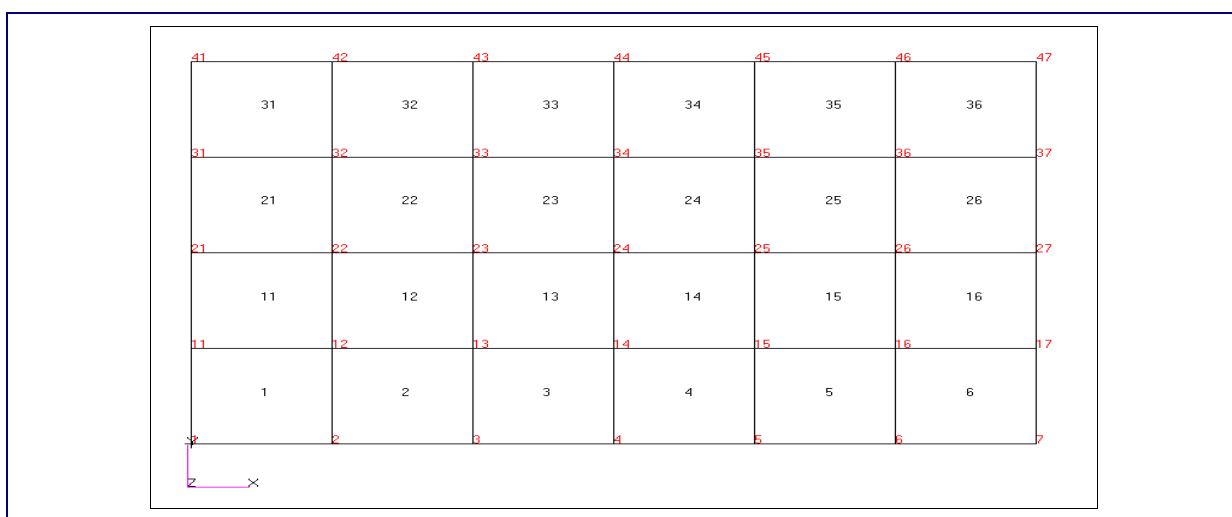


Figure 7-8 Undeformed Plot of the Thick Pad.

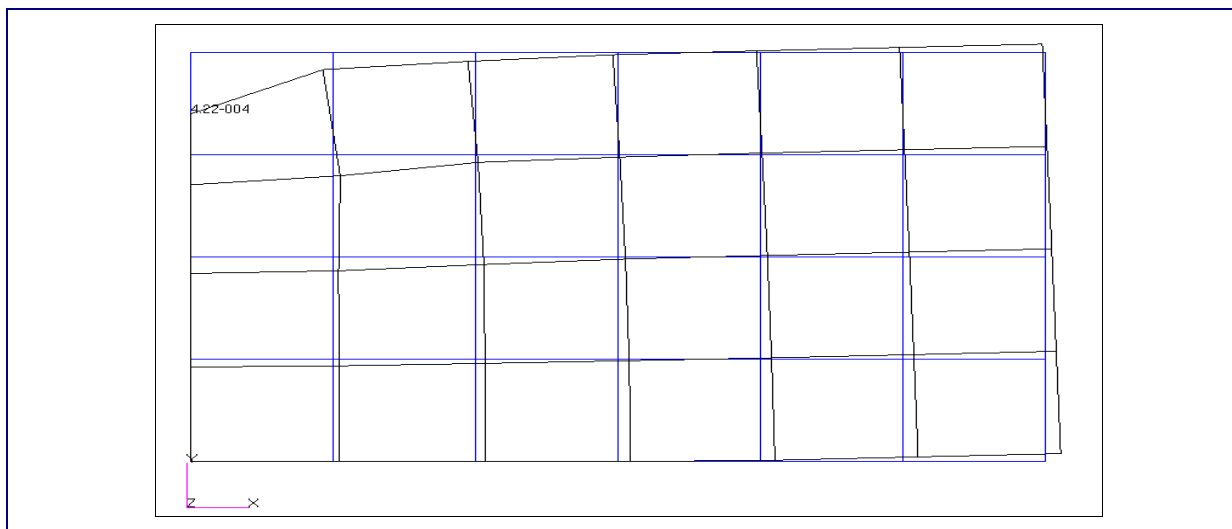


Figure 7-8(cont.) Deformed Plot of the Thick Pad.

From **Figure 7-8b**, it is obvious that grid points 6, 7, and possibly 5 have lifted off the foundation due to the downward vertical load. Looking at the output (**Figure 7-9**), one can see that grid points 1, 2, 3, and 4 are in contact as indicated under the “SHUT” output. Furthermore, the displacement output illustrates that grid points 1, 2, 3, and 4 have zero displacement in the T2 direction--indicating that these four points are in contact. The T2 displacements for grid points 5, 6, and 7 are positive--indicating that these three points have lifted off the foundation. The applied vertical load is reacted at the four grid points (1-4) as illustrated by the spcforce output.


```

0
0      SHUT
POINT  VALUE  POINT  VALUE  POINT  VALUE  POINT  VALUE  POINT
VALUE

COLUMN      1
      1 T2  1.00000E+00      2 T2  1.00000E+00      3 T2  1.00000E+00      4 T2  1.00000E+00
1 DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1  SEPTEMBER 10, 2001  MSC.NASTRAN  4/9/01
PAGE 12

0

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

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
      1      G      0.0      0.0      0.0      0.0      0.0      0.0
      2      G      2.815052E-05  0.0      0.0      0.0      0.0      0.0
      3      G      4.985067E-05  0.0      0.0      0.0      0.0      0.0
      4      G      6.308859E-05  0.0      0.0      0.0      0.0      0.0
      5      G      7.122347E-05  8.332880E-06  0.0      0.0      0.0      0.0
      6      G      7.456562E-05  3.158361E-05  0.0      0.0      0.0      0.0
      7      G      7.511267E-05  5.419435E-05  0.0      0.0      0.0      0.0
      .
      .

0

      L O A D   V E C T O R

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
      41      G      0.0      -1.000000E+00  0.0      0.0      0.0      0.0
1 DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1  SEPTEMBER 10, 2001  MSC.NASTRAN  4/9/01  PAGE
15

0

      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      -8.072071E-02  2.248780E-01  0.0      0.0      0.0      0.0
      2      G      0.0      3.992608E-01  0.0      0.0      0.0      0.0
      3      G      0.0      2.689573E-01  0.0      0.0      0.0      0.0
      4      G      0.0      1.069040E-01  0.0      0.0      0.0      0.0
     11      G      -1.541045E-01  0.0      0.0      0.0      0.0      0.0
     21      G      -1.733866E-01  0.0      0.0      0.0      0.0      0.0
     31      G      -1.259178E-01  0.0      0.0      0.0      0.0      0.0
     41      G      5.341296E-01  0.0      0.0      0.0      0.0      0.0

```

Figure 7-9 Abridged Output of Thick Pad.

R-Type Elements

- Introduction to R-Type Elements
- Description of the R-Type Elements
- The RBAR Element
- The RBE2 Element
- The RBE3 Element

8.1 Introduction to R-Type Elements

An R-type element is an element that imposes fixed constraints between components of motion at the grid points or scalar points to which they are connected. Thus, an R-type element is mathematically equivalent to one or more multipoint constraint equations (discussed in “[Constraints](#)” on page 257). Each constraint equation expresses one dependent degree of freedom as a linear function of the independent degrees of freedom.

In the past, and in some of the current MSC.Nastran documentation, all the R-type elements were referred to as rigid elements; however, the name “rigid” is misleading. The R-type elements that are rigid consist of the RROD, RBAR, RBE1, RBE2, and RTRPLT. The RBE3 and RSPLINE are interpolation elements and are not rigid. A brief description of all of the R-type elements is presented in the next section followed by a detailed description of the RBAR, RBE2, and RBE3 elements. These three elements are the most commonly used R-type elements in the MSC.Nastran element library.

In addition to the above R-type elements, the RSSCON element can be used to transition between plate and solid elements (see [page 338](#) for further details).

8.2 Description of the R-Type Elements

Each of the R-type elements generates internal MPC equations in MSC.Nastran. These equations are generated automatically; you do not need to specify an MPC request in the Case Control Section. R-type elements are included in your model if they are included in the Bulk Data Section. (This is similar to any of the other elements in MSC.Nastran.)

Unlike the MPC entries, an R-type element cannot be changed between subcases.

When using an R-type element, it is your responsibility to define which degrees of freedom are dependent and which are independent. The simplest way to describe this is to say that the motion of a dependent degree of freedom is expressed as a linear combination of one or more of the independent degrees of freedom. All dependent degrees of freedom are placed in what is referred to as the *m*-set. The independent degrees of freedom are temporarily placed in the *n*-set, which is the set that is not made dependent by MPCs or R-type elements. A constraint equation (an internal MPC equation) is generated for each dependent degree of freedom. A complete description of MSC.Nastran sets is given in “[Matrix Operations](#)” on page 455.

Seven R-type elements are described in, together with the number of constraint equations generated for each of the elements. This number is the same as the number of dependent degrees of freedom that you specify for the element.

Table 8-1 R-Type Element Available in MSC.Nastran

Name	Description	m = No. of Equations of Constraint Generated
RROD	Pin-ended rod which is rigid in extension.	$m = 1$
RBAR	Rigid bar with six degrees of freedom at each end.	$1 \leq m \leq 6$
RTRPLT	Rigid triangular plate with six degrees of freedom at each vertex.	$1 \leq m \leq 12$
RBE1	A rigid body connected to an arbitrary number of grid points. The independent and dependent degrees of freedom can be arbitrarily selected.	$m \geq 1$
RBE2	A rigid body connected to an arbitrary number of grid points. The independent degrees of freedom are the six components of motion at a single grid point. The dependent degrees of freedom at the other grid points all have the same user-selected component numbers.	$m \geq 1$

Table 8-1 R-Type Element Available in MSC.Nastran

RBE3	Defines a constraint relation in which the motion at a “reference” grid point is the least square weighted average of the motions at other grid points. The element is useful for “beaming” loads and masses from a “reference” grid point to a set of grid points.	$1 \leq m \leq 6$
RSPLINE	Defines a constraint relation whose coefficients are derived from the deflections and slopes of a flexible tubular beam connected to the referenced grid points. This element is useful in changing mesh size in finite element models.	$m \geq 1$
RSSCON	Transitions between plate and solid elements.	$M > 1$

As an introduction to the R-type elements, consider the RROD rigid element. For the RROD element, you specify a single component of translation at one of its two end points as a dependent degree of freedom. The equivalent component at the other end is the independent degree of freedom. Consider the example shown in [Figure 8-1](#).

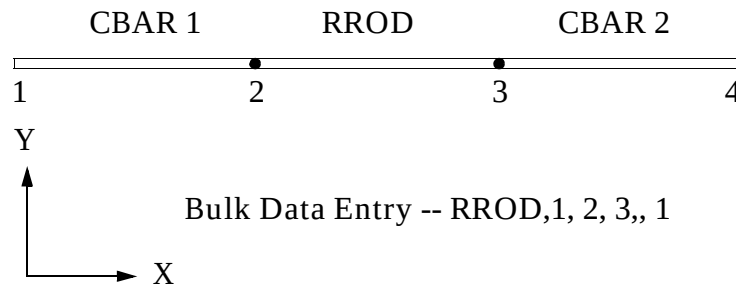


Figure 8-1 An RROD Connection

For this example, a rigid connection is made between the X component of grid point 3 to the X component of grid point 2. When you specify the RROD entry as shown, you are placing component 1 of grid point 3 (in the global system) into the m -set. The remaining 5 components at grid point 3 and all the components at grid point 2 are placed in the n -set and hence, are independent. CBAR 1 is not connected to components Y, Z, R_x , R_y , and R_z of grid point 3 in any manner; therefore, the ends of the bars are free to move in any of these directions. However, the ends of the two CBAR elements are rigidly attached in the X-direction. Having the connection only in the X-direction is consistent with , which shows that one degree of freedom is placed in the m -set for each RROD element.

For the other six R-type elements, several degrees of freedom may be specified as members of the m -set or the n -set. (Again, the latter designation may be temporary; they may be removed by additional constraints in your model.) For the RTRPLT, RBE1, RROD, and

RBE3 elements, any unlisted degrees of freedom at the grid points to which the element is joined are not connected to the element. This lack of connection can be regarded as either a sliding release or a rotating joint release, or both.

Four of the R-type elements (RBAR, RTRPLT, RBE1, and RBE2) must have exactly six components of motion in the n -set (i.e., independent degrees of freedom). These six degrees of freedom must be able to represent all of the rigid body motions of the element. Another important requirement is that the three rules described in “**Constraints**” in Chapter 7 for MPC entries must also be observed by the R-type elements. Typical applications that use R-type entries are shown in **Table 8-2**.

Table 8-2 Typical Application for Rigid Element

Application	R-Type Entries
Triangular Bell Crank	RTRPLT
Rigid Engine Blocks	RBE1
Tripod with Hinged Rigid Legs	RROD
Rigid Bulkhead	RBE2
Evaluation of Resultant Loads	RBE2
Connection of a Bar Element to a Shell	RBE2 or RBE3
Hinge Between Two Plates	RBAR
Recording Motion in a Nonglobal Direction	RBAR
Relative Motion	MPC
Incompressible Fluid in an Elastic Container	MPC
“Beaming” Loads and Masses	RBE3
Change in Mesh Size	RSPLINE
Transitions Between Plate and Solid Elements	RSSCON

One last word of caution is that the connection of two or more rigid elements to the same grid point should be done carefully to avoid specifying a degree of freedom as a member of the m -set more than once; otherwise, a fatal message is issued.

The next section is devoted to the RBAR element, one of the most commonly used R-type elements in the MSC.Nastran element library.

8.3 The RBAR Element

The RBAR element rigidly connects from one to six dependent degrees of freedom (the m -set) to exactly six independent degrees of freedom. The six independent degrees of freedom must be capable of describing the rigid body properties of the element.

The format for the RBAR entry is as follows:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB		

Field	Contents
EID	Element identification number.
GA, GB	Grid point identification number of connection points.
CNA, CNB	Component numbers of independent degrees of freedom in the global coordinate system for the element at grid points GA and GB.
CMA, CMB	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid points GA and GB.

The most common approach when using the RBAR element is to define one end of the RBAR with all six independent degrees of freedom with dependent degrees of freedom at the other end. (However, placing all of the independent degrees of freedom at one end is not a requirement.) To determine if the choice you make for the independent degrees of freedom meets the rigid body requirements, ensure that the element passes the following simple test: If you constrain all of the degrees of freedom defined as independent on the RBAR element, is the element prevented from any possible rigid body motion?

As an example, consider the RBAR configurations shown in [Figure 8-2](#). For configurations (a) and (c), if the six independent degrees of freedom are held fixed, the element cannot move as a rigid body in any direction. However, for (b), if all six of the independent degrees of freedom are held fixed, the element can still rotate about the Y-axis. Configuration (b) does not pass the rigid body test and does not work as an RBAR element.

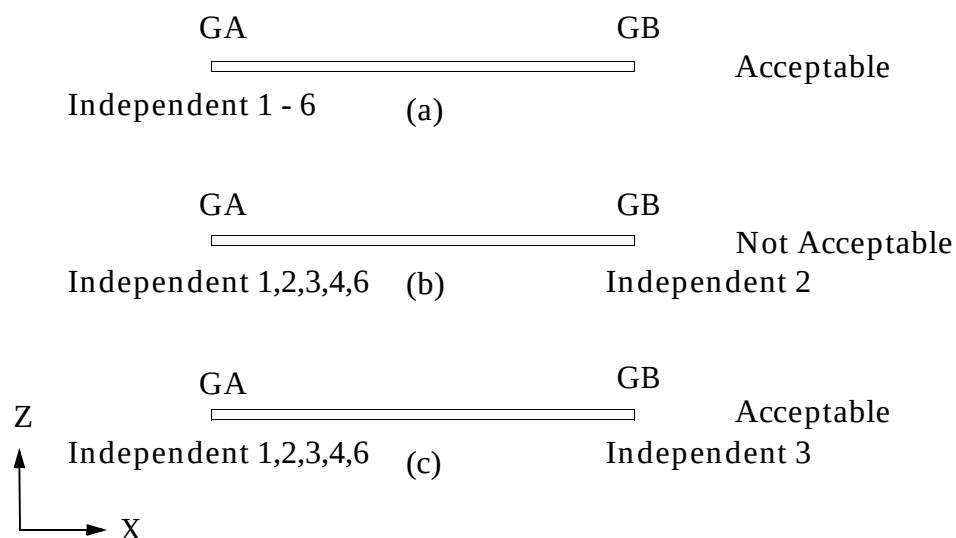


Figure 8-2 Defining Independent DOFs on the RBAR

MSC.Nastran generates internal MPC equations for the R-type elements. As an example of this, consider the model of a thick plate with bars attached as shown in **Figure 8-3**. The interface between the bars and the plate is modeled two ways, first using MPC entries and second using RBAR elements.

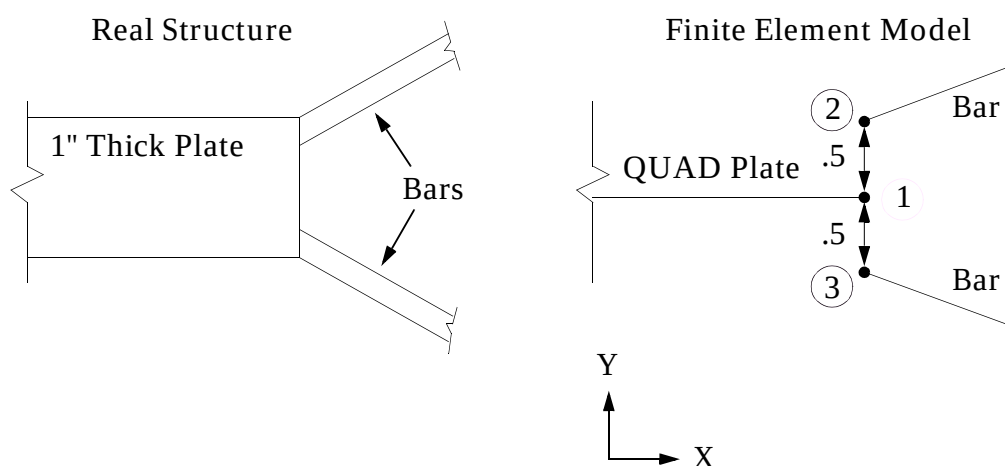


Figure 8-3 Model of a Thick Plate with Bars Attached

Option 1 -- Model the Transition with MPC Equations

Plate theory states that plane sections remain planar. If this is the case, then grid points 2 and 3 are slaves to grid point 1. Therefore, you need to write the equations for the in-plane motion of grid points 2 and 3 as a function of grid point 1. Each RBAR element creates up to six constraint equations.

Looking only at the motion in the x-y plane,

$$\begin{aligned}
 u_{1_2} &= u_{1_1} - .5 \cdot u_{6_1} & (u_{1_2} = \text{displacement in the 1-direction at grid point 2}) \\
 u_{1_3} &= u_{1_1} + .5 \cdot u_{6_1} \\
 u_{6_3} &= u_{6_1} \\
 u_{6_2} &= u_{6_1} \\
 u_{2_2} &= u_{2_1} \\
 u_{2_3} &= u_{2_1}
 \end{aligned}$$

The MPC entries for this model are as follows:

	1	2	3	4	5	6	7	8	9	10
MPC		1	2	1	1.	1	1	-1.		
			1	6	.5					
MPC		1	3	1	1.	1	1	-1.		
			1	6	-.5					
MPC		1	3	6	1.	1	6	-1.		
MPC		1	2	6	1.	1	6	-1.		
MPC		1	2	2	1.	1	2	-1.		
MPC		1	3	2	1.	1	2	-1.		

Note: MPC = 1 must appear in the Case Control Section to use these entries.

Option 2 - Model the Transition with RBAR Entries

The R-type elements are easier to use than the equivalent MPC entries.

	1	2	3	4	5	6	7	8	9	10
\$RBAR	EID	GA	GB	CNA	CNB	CMA	CMB			
RBAR	99	1	2	123456						
RBAR	100	1	3	123456						

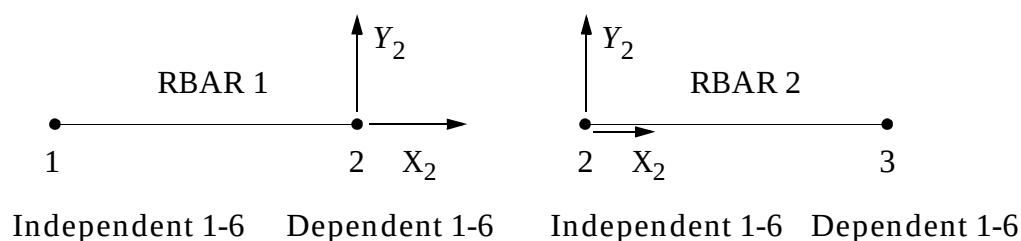
RBAR 99 generates MPC equations for the motion of grid point 2 as a function of grid point 1. Likewise, RBAR 100 generates MPC equations for the motion of grid point 3 as a function of grid point 1. These RBARs generate the MPC equations for all six DOFs at grid points 2 and 3. If it is desired to have the equations generated only for the in-plane motion, the field labeled as CMB in the RBAR entries has the values 126 entered.

A particular area of confusion for the new user is when you need to connect R-type elements together. The important thing to remember is that you can place a degree of freedom into the *m*-set only once. Consider the two RBAR elements shown in [Figure 8-4](#) that are acting as a single rigid member.

If you choose grid point 1 for RBAR 1 to be independent (1-6), then grid point 2 for RBAR 1 must be dependent (1-6). Since grid point 2 is dependent for RBAR1, it must be made independent for RBAR 2. If you made grid point 2 dependent for RBAR 2 as well as RBAR 1, a fatal error would result. Since grid point 2 is independent (1-6) for RBAR 2, grid point 3 will be dependent (1-6).

If you chose grid point 1 of RBAR 1 to be dependent, then grid point 2 for RBAR 1 would be independent. Grid point 2 of RBAR 2 would be dependent, and grid point 3 of RBAR 2 would be independent.

The RBAR element is often used to rigidly connect two grid points in your model.



Note: RBARs 1 and 2 are connected. They are shown separated for clarity.

Figure 8-4 Connecting Two RBAR Elements

Both options for connecting the RBAR elements are shown in [Listing 8-1](#). For clarity, CBAR3, which is connected to grid point 3, is not shown in [Figure 8-4](#).

Listing 8-1 Connecting RBAR Elements

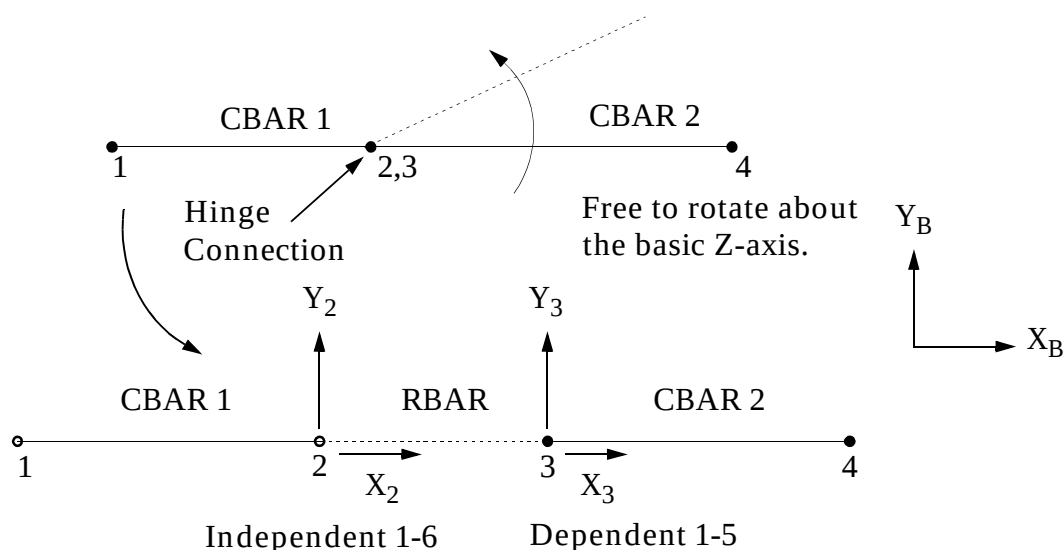
```

$
$ FILENAME RBAR1.DAT
$
ID LINEAR,RBAR1
SOL 101
TIME 2
CEND
TITLE = CONNECTING 2 RBARS
DISPLACEMENT = ALL
SUBCASE 1
    LOAD = 1
SUBCASE 2
    LOAD = 2
BEGIN BULK
$
GRID      1          0.      0.      0.
GRID      2          10.      0.      0.

GRID      3          20.      0.      0.
GRID      4          30.      0.      0.          123456
$
$ OPTION 1
$
RBAR      1          1          2          123456
RBAR      2          2          3          123456
$
$ OPTION 2
$
$RBAR     1          1          2          123456
$RBAR     2          2          3          123456
$
CBAR      3          1          3          4          0.      1.      0.
PBAR      1          1          1.          1.          1.      1.
MAT1      1          20.4          .3
$
$ POINT LOAD
$
FORCE     1          1          1.          1.          0.      0.
FORCE     2          1          1.          0.          1.      0.
$
ENDDATA

```

As a final example of the RBAR element, consider the hinge model shown in [Figure 8-5](#). A similar model was used in Chapter 4 to demonstrate pin flags.



Note: Grid points 2 and 3 are coincident.
They are shown separated for clarity.

Figure 8-5 Modeling a Hinge Using an RBAR

The simplest way to model the hinge connection with an RBAR is to use coincident grid points at the center of rotation (grid points 2 and 3 in this example) and define an RBAR between the two grid points. This RBAR has zero length, which is acceptable for the RBAR. Make all six of the components associated with one grid point independent. Make only a select number of components of the other grid point dependent, leaving independent the components representing the hinge. A partial listing of the input file for this model is shown in [Listing 8-2](#). Note that the components 1 through 6 of grid point 2 are independent and components 1 through 5 of grid point 3 are dependent. Component 6 of the grid point 3 is left independent, permitting CBAR 2 to rotate about the Z axis with respect to CBAR 1.

Listing 8-2 Hinge Joint Using an RBAR

```

$
$ FILENAME - RBAR2.DAT
$
ID LINEAR,RBAR2
SOL 101
TIME 2
CEND
TITLE = CONNECTING 2 BARS WITH AN RBAR HINGE
DISPLACEMENT = ALL
LOAD = 1
FORCE = ALL
BEGIN BULK
$
GRID      1          0.      0.      0.          123456
GRID      2          10.      0.      0.
GRID      3          10.      0.      0.
GRID      4          20.      0.      0.          123456
$
RBAR      99      2      3      123456          12345
$
CBAR      1      1      1      2      0.      1.      0.
CBAR      2      1      3      4      0.      1.      0.
PBAR      1      1      .1      .01      .01      .02
MAT1      1      20.+4      .3
$
$ POINT LOAD
$
FORCE      1      2          100.      0.      1.      0.
$
ENDDATA

```

A word of caution: if you intentionally use coincident grid points (as in this example), you may run the risk of removing them accidentally if you later use the “equivalence” option available in most preprocessors. Equivalencing grid points causes all duplicate grid points in your model to be deleted; however, this may not be your intention as illustrated in this example.

8.4 The RBE2 Element

The RBE2 (Rigid Body Element, type 2) provides a very convenient tool for rigidly connecting the same components of several grid points together. You should note that multiple RBARs or an RBE1 can be used wherever an RBE2 is used; however, they may not be as convenient.

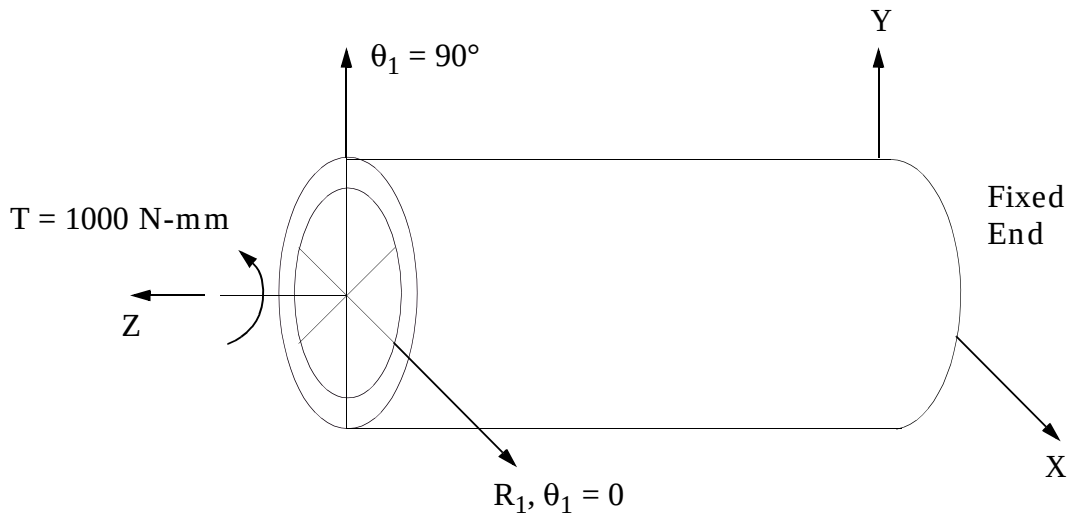
The format for the RBE2 entry is as follows:

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

Field	Contents
EID	Element identification number.
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned.
CM	Component numbers of the dependent degrees of freedom in the global coordinate system at grid points GMi.
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned.

When using an RBE2, you need to specify a single independent grid point (the GN field) in which all six components are assigned as independent. In the CM field, you can specify the dependent degrees of freedom at grid points GMi in the global coordinate system. The GMi grid points are the grid points at which the dependent degrees of freedom are assigned. The dependent components are the same for all the listed grid points (if this is unacceptable, use the RBAR elements, multiple RBE2s elements, or the RBE1 element).

As an example showing the use of an RBE2 element, consider the tube shown in **Figure 8-6**. The goal is to maintain a circular cross section at the end of the tube while applying a torque about the axis of the tube. Furthermore, you allow the tube to expand in the R direction, but the center of the end of the tube should not move from its original position.



Tube Dimensions:

Mean Diameter=30 mm

T=3 mm

Length=90 mm

$E=25 \times 10^4 \text{ N/mm}^2$

$\nu=0.3$

Figure 8-6 Tube with an End Torque Loading

The input file for this model is given in [Listing 8-3](#). The goal is to have the end of the tube rotate uniformly while allowing the tube to expand in the R-direction. The simplest way to accomplish this is to define a local cylindrical coordinate system with the origin located at the center of the free end as shown in [Figure 8-6](#). This cylindrical coordinate system is then used as the displacement coordinate system (field 7) for all of the grid points located at the free end. Now when an RBE2 element is connected to these grid points, the dependent degrees of freedom are in the local coordinate system. By leaving the R-direction independent, the tube is free to expand in the radial direction.

Grid point 999 is defined at the center of the free end to serve as the independent point. The θ , Z , R_r , R_θ , and R_z components of the grid points on the end of the tube are dependent degrees of freedom. To ensure that the axis of the tube remains in the same position, an SPC is applied to all components of grid point 999 except in the R_z direction, which is the component about which the torque is applied. It is interesting to note that the CD field of grid point 999, the independent point, is different than that of the CD field of the dependent point, and this is an acceptable modeling technique. Furthermore, you should always use a rectangular coordinate system in the CD field for any grid point that lies on the polar axis. In this example, grid point 999 lies on the Z-axis (see [Figure 8-6](#)); therefore, it should not use coordinate system 1 for its CD field.

Listing 8-3 Applying a Torque to a Tube Using an RBE2

```

$
$ FILENAME - TORQUE.DAT
$
ID      LINEAR,TORQUE
SOL      101
TIME     5
CEND
TITLE = TUBE WITH END TORQUE
SET 1 =110, 111, 112, 119, 120, 127, 128,132
DISP = 1
SPC = 1
LOAD = 1
BEGIN BULK
PARAM    AUTOSPC YES
$
CORD2C   1          0          0.0          0.0          90.          0.0          0.0          91.0
          1.0          0.0          91.0
$
$
GRID     101          15.          0.0          0.0
GRID     102          10.6066 10.6066 0.0
GRID     103          7.105-1515.          0.0
GRID     104          15.          0.0          30.
GRID     105          10.6066 10.6066 30.
GRID     106          1.066-1415.          30.
GRID     107          15.          0.0          60.
GRID     108          10.6066 10.6066 60.
GRID     109          1.066-1415.          60.
GRID     110          15.          0.0          90.          1
GRID     111          10.6066 10.6066 90.          1
GRID     112          1.421-1415.          90.          1
GRID     113          -10.606610.6066 0.0
GRID     114          -15.          7.105-150.0
GRID     115          -10.606610.6066 30.
GRID     116          -15.          1.066-1430.
GRID     117          -10.606610.6066 60.
GRID     118          -15.          1.066-1460.
GRID     119          -10.606610.6066 90.          1
GRID     120          -15.          1.421-1490.          1
GRID     121          -10.6066-10.60660.0
GRID     122          0.0          -15.          0.0
GRID     123          -10.6066-10.606630.
GRID     124          -1.78-14-15.          30.
GRID     125          -10.6066-10.606660.
GRID     126          -3.2-14 -15.          60.
GRID     127          -10.6066-10.606690.          1
GRID     128          -4.97-14-15.          90.          1
GRID     129          10.6066 -10.60660.0
GRID     130          10.6066 -10.606630.
GRID     131          10.6066 -10.606660.
GRID     132          10.6066 -10.606690.          1
GRID     999          0.0          0.0          90.
$
RBE2     200          999          23456          110          111          112          119          120          +
+         127          128          132
$
$QUAD4S REMOVED, SEE THE FILE ON THE DELIVERY MEDIA

```

Listing 8-3 Applying a Torque to a Tube Using an RBE2 (continued)

```

$
$ THIS SECTION CONTAINS THE LOADS, CONSTRAINTS, AND CONTROL BULK DATA ENTRIES
$
$
MOMENT  1      999      0      1000.    0.0    0.0    1.
$
SPC1    1      123456  101      102      103      113
SPC1    1      123456  114      121      122      129
SPC1    1      12345   999
$
$
$ THIS SECTION CONTAINS THE PROPERTY AND MATERIAL BULK DATA ENTRIES
$
$
PSHELL  1      1      3.      1
$
MAT1    1      250000.      .3
ENDDATA

```

The displacement vector of the end grid points is shown in [Figure 8-7](#). The q direction T2 is the same for each of the end grid points as desired. The R-direction T1 is small but not exactly zero, indicating that the tube is permitted to expand in the radial direction.

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
110	G	1.487426E-20	2.526745E-04	.0	.0	.0	1.684497E-05
111	G	8.858868E-20	2.526745E-04	.0	.0	.0	1.684497E-05
112	G	-4.178414E-21	2.526745E-04	.0	.0	.0	1.684497E-05
119	G	-1.985002E-20	2.526745E-04	.0	.0	.0	1.684497E-05
120	G	-4.204334E-20	2.526745E-04	.0	.0	.0	1.684497E-05
127	G	4.297897E-20	2.526745E-04	.0	.0	.0	1.684497E-05
128	G	8.858550E-21	2.526745E-04	.0	.0	.0	1.684497E-05
132	G	-9.068584E-20	2.526745E-04	.0	.0	.0	1.684497E-05

Figure 8-7 Selected Output for the RBE2 Example

8.5 The RBE3 Element

The RBE3 element is a powerful tool for distributing applied loads and mass in a model. Unlike the RBAR and RBE2 discussed in the previous sections, the RBE3 does not add additional stiffness to your structure. Forces and moments applied to reference points are distributed to a set of independent degrees of freedom based on the RBE3 geometry and local weight factors.

The format of the RBE3 entry is as follows:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			

Field	Contents
EID	Element identification number. Unique with respect to other rigid elements.
REFGRID	Reference grid point identification number.
REFC	Component numbers at the reference grid point.
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j.
Ci	Component numbers with weighting factor WTi at grid points Gi,j.
Gi,j	Grid points whose components Ci have weighting factor WTi in the averaging equations.
“UM”	Indicates the start of the degrees of freedom belonging to the <i>m</i> -set. The default action is to assign only the components in REFC to the <i>m</i> -set.
GMi	Identification numbers of grid points with degrees of freedom in the <i>m</i> -set.
CMi	Component numbers of GMi to be assigned to the <i>m</i> -set.

The manner in which the forces are distributed is analogous to the the classical bolt pattern analysis. Consider the bolt pattern shown in [Figure 8-8](#) with a force and moment *M* acting at reference point A. The force and moment can be transferred directly to the weighted center of gravity location along with the moment produced by the force offset.

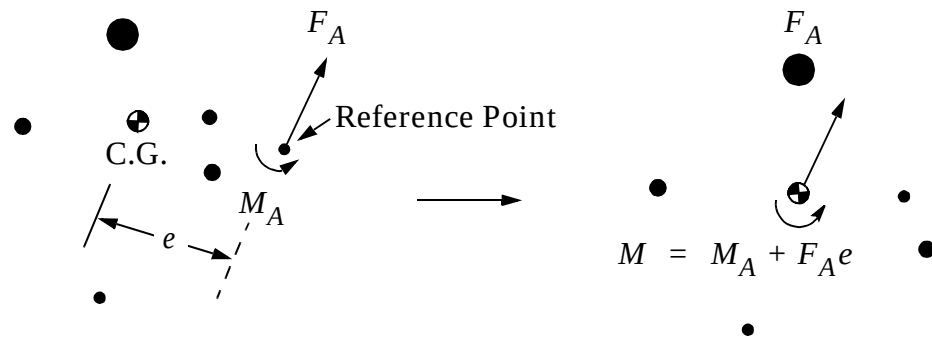
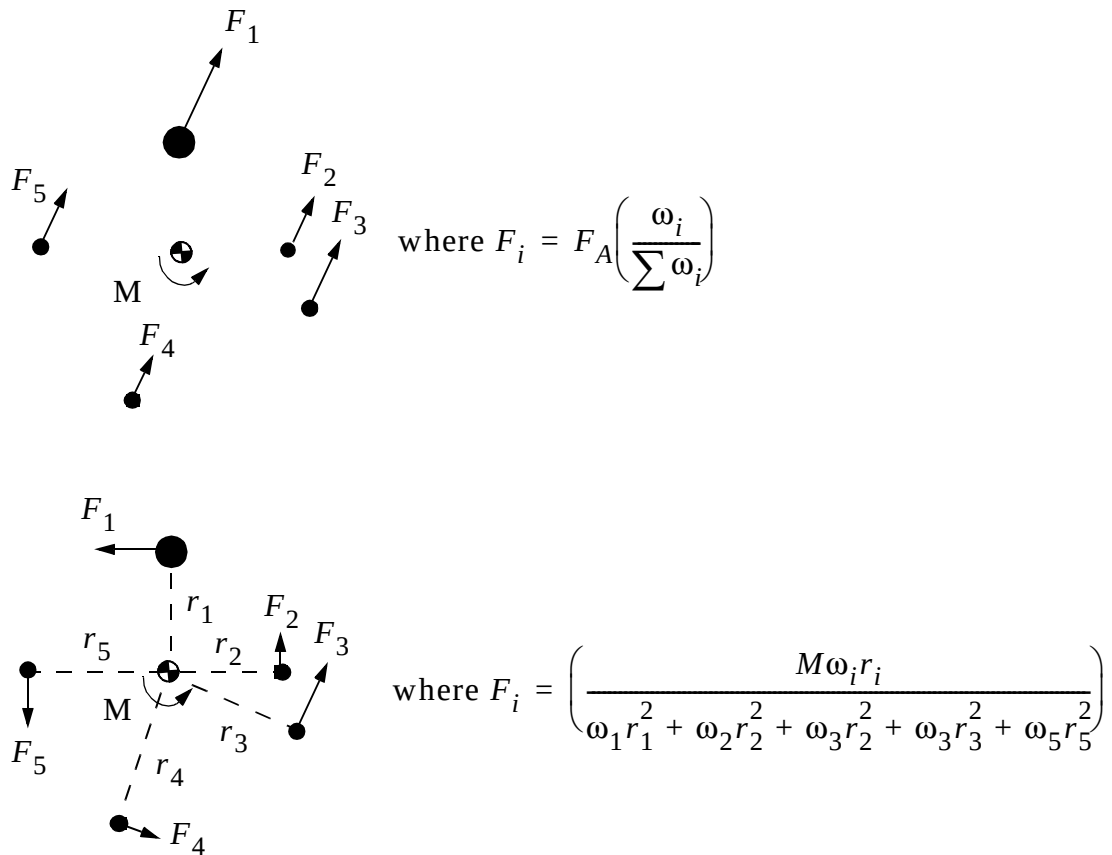


Figure 8-8 RBE3 Equivalent Force and Moment at the Reference Point

The force is distributed to the bolts proportional to the weighting factors. The moment is distributed as forces, which are proportional to their distance from the center of gravity times their weighting factors, as shown in [Figure 8-10](#). The total force acting on the bolts is equal to the sum of the two forces. These results apply to both in-plane and out-of-plane loadings.



where:

F_i = force at DOF i

ω_i = weighting factor for DOF i

r_i = radius from the weighted center of gravity to point i

Figure 8-9 RBE3 Force Distribution

As an example, consider the cantilever plate modeled with a single CQUAD4 element shown in **Figure 8-10**. The plate is subjected to nonuniform pressure represented by a resultant force acting at a distance of 10 mm from the center of gravity location. The simplest way to apply the pressure is to use an RBE3 element to distribute the resultant load to each of the four corner points.

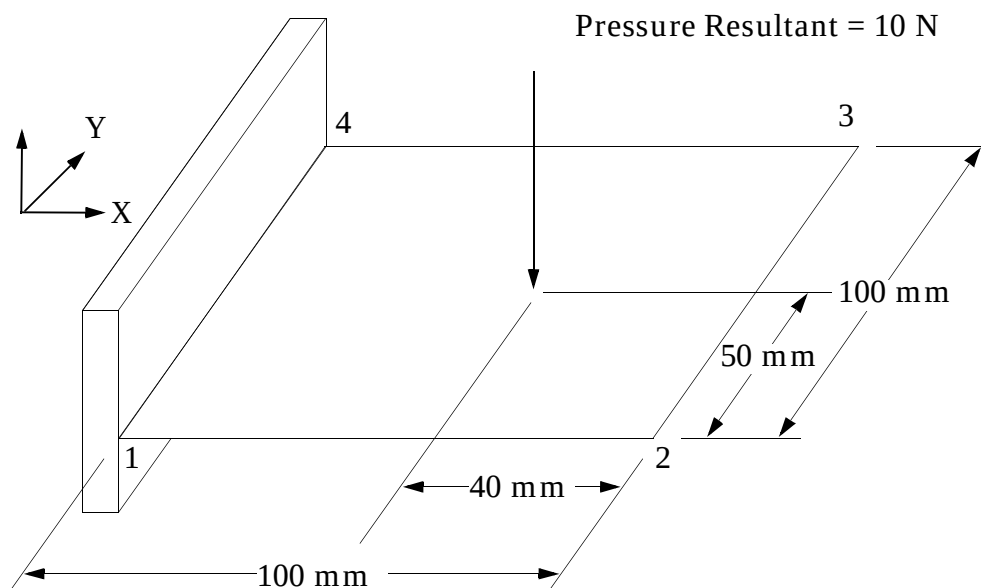


Figure 8-10 Using an RBE3 to Represent a Nonuniform Pressure Load

The input file representing this example is shown in **Listing 8-4**. Grid point 99 is called the REFGRID and is the location where the force is applied. This point is connected only to those degrees of freedom listed on the REFC field (the T3 component in this example). The default action of this element is to place the REFC degrees of freedom in the *m*-set. The element has provisions to place other DOFs in the *m*-set instead. However, this is an advanced feature and is beyond the scope of this user's guide. The groups of connected grid points begin in field 5. For this example, the connected grid points are the corner points.

Listing 8-4 Distributing Force with an RBE3

```

$
$ FILENAME - RBE3.DAT
$
ID      LINEAR,RBE3
SOL     101
TIME    5
CEND
TITLE = SINGLE ELEMENT WITH RBE3
SPC = 1
LOAD = 1
OLOAD   = ALL
GPFORCE = ALL
SPCFORCES = ALL
BEGIN BULK
$
RBE3    10          99      3      1.0      123      1      2
        3          4
FORCE   1          99          100.    0.      0.      1.
$
PARAM   POST      0
$
GRID    1          0.      0.      0.
GRID    2          100.    0.      0.
GRID    3          100.    100.    0.
GRID    4          0.      100.    0.
GRID    99         60.      50.    0.
$
PSHELL  1          4          10.      4
$
MAT1    4          4.E6          0.
$
CQUAD4  1          1          1          2          3          4
$
SPC1    1          123456  1          4
ENDDATA

```

The start of a group is indicated by a real number WT_i, which is used as a weighting factor for the grid points in the group. In this example, a simple distribution based only on the geometry of the RBE3 is desired so that a uniform weight is applied to all points. The weighting factors are not required to add up to any specific value. For this example, if the WT1 field is 4.0 instead of 1.0, the results will be the same.

The independent degrees of freedom for the group are listed in the C_i field. Note that all three translational DOFs are listed even though the REFC field does not include the T1- and T2-direction. All three translational DOFs in the C_i field are included because the DOFs listed for all points must be adequate to define the rigid body motion of the RBE3 element even when the element is not intended to carry loads in certain directions. If any translational degrees of freedom are not included in C1 in this example, a fatal message is issued.

The element described by this RBE3 entry does not transmit forces in the T1- or T2- direction. The two reasons for this are that the reference grid point is not connected in this direction and all of the connected points are in the same plane. Note that the rotations are not used for the independent DOFs. In general, it is recommended that only the translational components be used for the independent degrees of freedom.

A selected portion of the output file produced by this example is shown in [Figure 8-11](#).

D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	6.000000E-04	1.567011E-20	-9.000000E-06	.0	
3	G	.0	.0	6.000000E-04	3.282253E-21	-9.000000E-06	.0	
4	G	.0	.0	.0	.0	.0	.0	
99	G	.0	.0	3.600000E-04	.0	.0	.0	

L O A D V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
99	G	.0	.0	1.000000E+02	.0	.0	.0	

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	-5.000000E+01	7.275958E-12	3.000000E+03	.0	
4	G	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0	

G R I D P O I N T F O R C E B A L A N C E								
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0
1	1	QUAD4	.0	.0	3.000000E+01	-3.681428E-12	-3.000000E+03	.0
1		*TOTALS*	.0	.0	-2.000000E+01	3.594530E-12	4.092726E-12	.0
2	1	QUAD4	.0	.0	-3.000000E+01	8.115214E-12	1.315632E-11	.0
2		*TOTALS*	.0	.0	-3.000000E+01	8.115214E-12	1.315632E-11	.0
3	1	QUAD4	.0	.0	-3.000000E+01	7.376250E-12	-6.994877E-12	.0
3		*TOTALS*	.0	.0	-3.000000E+01	7.376250E-12	-6.994877E-12	.0
4		F-OF-SPC	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0
4	1	QUAD4	.0	.0	3.000000E+01	-4.079332E-12	-3.000000E+03	.0
4		*TOTALS*	.0	.0	-2.000000E+01	3.196626E-12	4.547474E-13	.0
99		APP-LOAD	.0	.0	1.000000E+02	.0	.0	.0
99		*TOTALS*	.0	.0	1.000000E+02	.0	.0	.0

Figure 8-11 Select Output for the RBE3 Example

The displacement of grid points 1 and 4 is zero due to the SPC applied to these points. The sum of the SPC forces at these two grid points is equal to the load applied to the reference grid point. The load transmitted to the corner points can be seen by inspecting the GPFORCE output. The force applied to the points due to the R-type elements and MPC entries is not listed specifically in the GPFORCE output. These forces show up as unbalanced totals (which should typically be equal to numeric zero). The forces applied to the corner grid points 1 through 4 are -20, -30, -30, and -20 N, respectively.

The most common usage of the RBE3 element is to transfer motion in such a way that all six DOFs of the reference point are connected. In this case, all six components are placed in the REFC field, and only components 123 are placed in the Ci field.

The load distributing capability of the RBE3 element makes it an ideal element to use to apply loads from a coarse model (or hand calculation) onto a detailed model of a component. For example, the shear distribution on a cross section is a function of the properties of that section. This shear loading may be applied to a cross section by performing a calculation of the shear distribution based on unit loading and using an RBE3 element with appropriate weighting factors for each grid point. In this manner, only one shear distribution need be calculated by hand. Since there are usually multiple loading conditions to be considered in an analysis, they may be applied by defining different loads to the dependent point on the RBE3 element.

For example, consider the tube attached to a back plate as shown in [Figure 8-12](#). Suppose that you are not particularly interested in the stress in the tube or the attachment, but you are concerned about the stresses in the back plate. For this reason, you choose not to include the tube in the model; however, you want the load transferred from the tube into the back plate attachment to be approximately correct. The question is: How should the loads be applied to the attachment to simulate the behavior of the tube?

Engineering principles dictate that the Z forces (forces acting normal to the back plate) acting on the attachment vary linearly as a function of the distance from the neutral axis. The simplest method of distributing the Z forces to the attachment grid is with an RBE2 or an RBE3 element. If an RBE2 is used, the attachment ring is rigid in the Z-direction. If an RBE3 element is used, no additional stiffness is added to the attachment ring. It is an engineering decision regarding which element to use since both are approximations. For this example, use the RBE3 element. Since the weight factors for the grid points in the Z-direction are equal, the forces are distributed to the grid points based on the geometry of the grid pattern only.

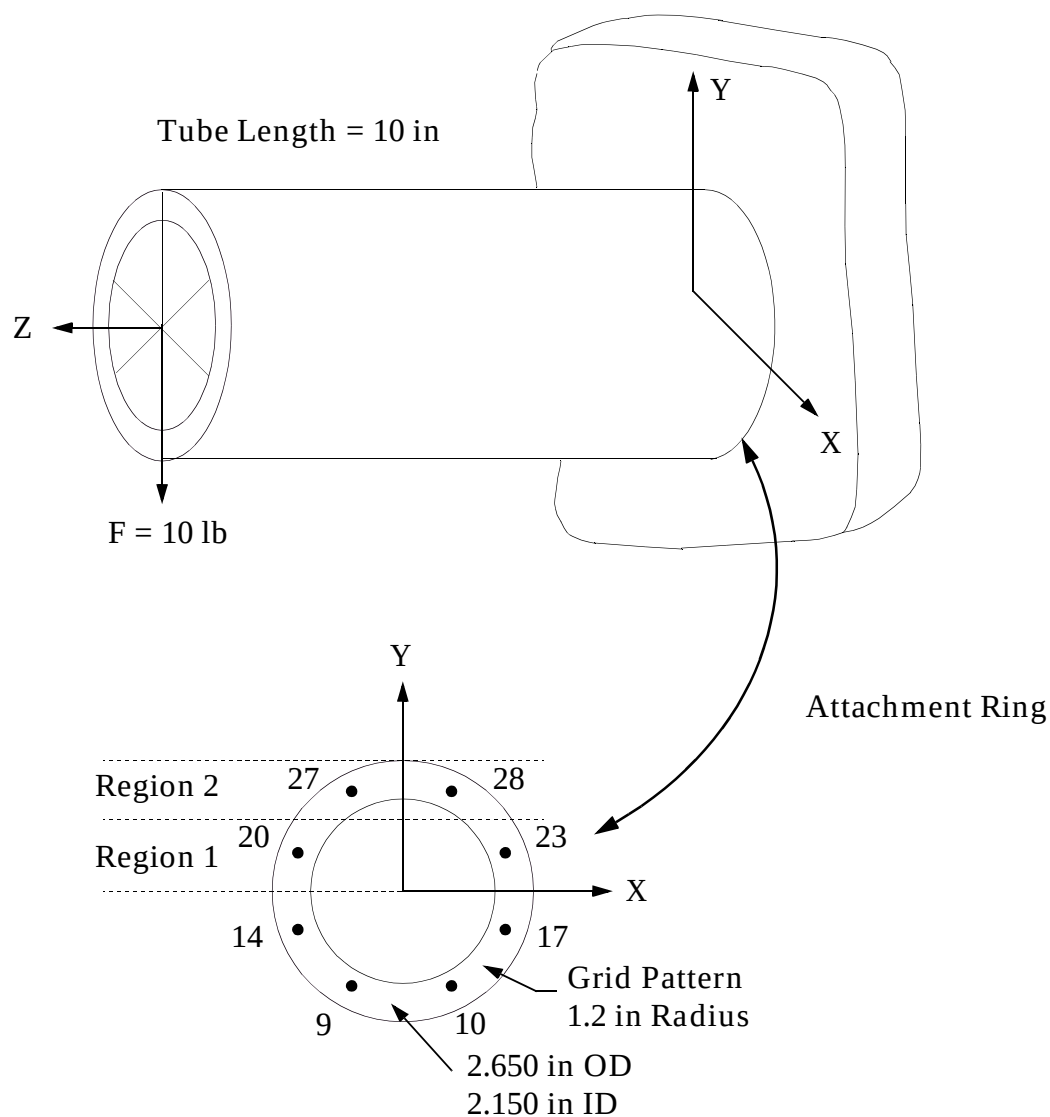


Figure 8-12 Attachment Ring

The shear force acting on the attachment does not act linearly; it is maximum at the neutral plane and tapers to zero at the top and bottom fibers (if the tube is solid, the shear distribution is a quadratic function, but in our example, it is a thick walled tube). The first step is to calculate the shear forces acting on the attachment ring as a function of the distance from the neutral plane using the classical strength of materials calculations. The result of this calculation is shown in [Figure 8-13](#). The shear force curve is divided into two regions, each region representing a grid point region as shown in [Figure 8-12](#). The area under the curve for each region represents the portion of the shear force transmitted to the grid points within the region. Using these area values as the coefficients for an RBE3 entry, the RBE3 distributes the shear force in a manner similar to the shear force curve.

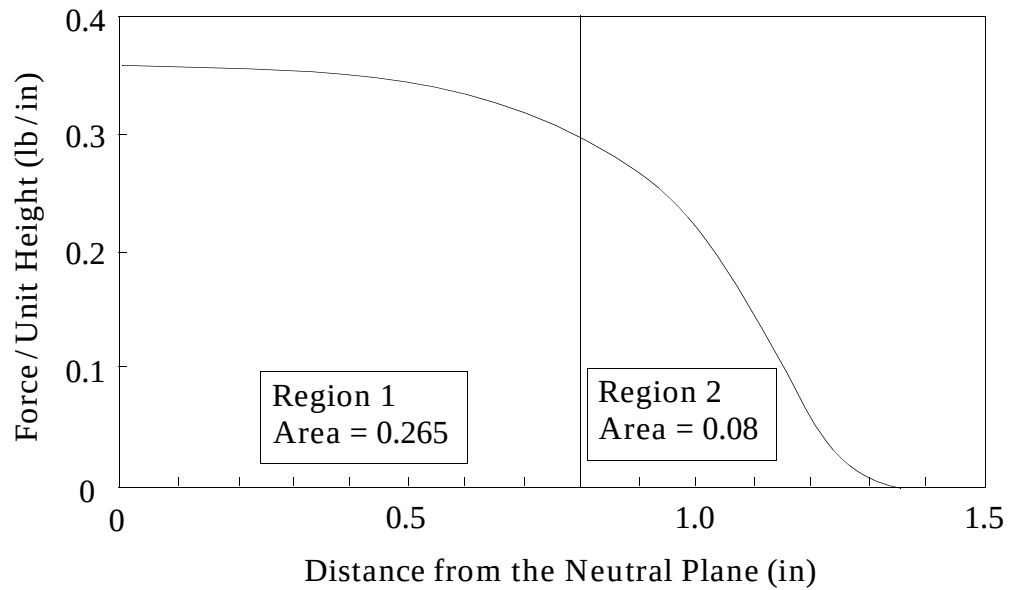


Figure 8-13 Shear Force on the Attachment Ring

The input file for this example is shown in [Listing 8-5](#). The reaction force and moment due to the applied load of 10 lb acting at the end of the tube are 10 lb and 100 in-lb, respectively.

Listing 8-5 Distributing the Attachment Forces with RBE3

```

$
$FILENAME - SHEAR1.DAT
$
ID      LINEAR,SHEAR1
SOL     101
TIME    5
CEND
TITLE = SHEAR TEST CASE USING AN RBE3
SET 1 = 9,10,14,17,20,23,27,28
GPFORCE = 1
SPC = 1
LOAD = 1
BEGIN BULK
PARAM   POST      0
PARAM   AUTOSPC   YES
$
$ RIGID CONNECTION USING TWO RBE3
$
RBE3    100        99      3456    1.0    123     9      10
        14        17      20      23     27     28
RBE3    101        99      12      0.08   123     9      10
        27        28      0.265   12     14     17     20     23
$
GRID    99         2.0     2.0     0.0
$
FORCE   1          99      1.       0.0    1.0     0.0
MOMENT  1          99      1.       1.0    0.0     0.0
$
$
$ ONLY THE END GRIDS ARE SHOWN
$

```

Listing 8-5 Distributing the Attachment Forces with RBE3 (continued)

```

GRID    9          1.6    .8    0.0
GRID   10          2.4    .8    0.0
GRID   14           .8    1.6    0.0
GRID   17          3.2    1.6    0.0
GRID   20           .8    2.4    0.0
GRID   23          3.2    2.4    0.0
GRID   27          1.6    3.2    0.0
GRID   28          2.4    3.2    0.0
$
$ QUAD4S, PSHELL, MAT1, AND SPC NOT SHOWN
$
ENDDATA

```

A full model including the model of the tube was also generated for comparison. The grid point forces for the attachment points for both the simple model described above and the full model are summarized in [Table 8-3](#). As can be seen, the force distribution from the RBE3 is close to that of the full model. This approximation is acceptable for many applications. The disadvantage of using this type of simplified model is that the stiffness of the tube is neglected. The full model is located on the delivery media with the name “shear1a.dat”.

Table 8-3 Comparison of RBE3 Attachment Forces to the Full Model

Grid Point	F_y		F_z	
	RBE3 Model	Full Model	RBE3 Model	Full Model
8,9	-0.058	-0.077	0.188	0.180
14,17	-0.192	-0.172	0.063	0.074
20,23	-0.192	-0.172	-0.063	-0.074
27,28	-0.058	-0.077	-0.188	-0.180

The most common user error in RBE3 element specification results from placing 4, 5, or 6 in the C_i (independent DOF) field in addition to the translation components. The rotations of the dependent point are fully defined by the translational motion of the independent points. The ability to input 4, 5, or 6 in the C_i field is only for special applications, such as when all of the connected points are colinear.

Small checkout models are recommended whenever you are specifying elements with nonuniform weight factors, asymmetric geometry or connected degrees of freedom, or irregular geometry. Using small checkout models is especially necessary when the reference point is not near the center of the connected points.

In summary, the intended use of the RBE3 element is to transmit forces and moments from a reference point to several non-colinear points. The rotation components 4, 5, and 6 should be placed in the C_i field only for special cases, such as when the independent points are colinear. For further details, see [MSC.Nastran Common Questions and Answers](#).

Modeling Guidelines

- Introduction to Modeling Guidelines
- Choosing the Right Element
- Mesh Density
- Mesh Transitions
- Grid Point Stresses
- Consistent Loading
- Symmetry

9.1 Introduction to Modeling Guidelines

Finite element modeling in many ways is more like an art than a science since the quality of the results is dependent upon the quality of your model. One of the more common errors that a beginning finite element analyst makes in modeling is to simply simulate the geometry rather than to simulate both the geometry and the physical behavior of the real structure. The following modeling guidelines are provided to put a little more science back into the art of finite element modeling:

- Choosing the right element.
- Mesh density.
- Mesh transitions.
- Grid point stresses.
- Consistent loading.
- Symmetry.

The above guidelines are by no means complete; however, they do serve as a good starting point. There is no better substitute for good modeling than experience. It is also good modeling practice to simulate and validate a new capability or a feature that you have not used before with a small prototype model before applying this feature to your production model. Model verification techniques are covered in “[Model Verification](#)” on page 369.

9.2 Choosing the Right Element

MSC.Nastran contains a large library of structural elements. In many situations several elements are capable of modeling the same structural effects. The criteria for the selection of an element may include its capabilities (for example, whether it supports anisotropic material properties), its cost (in general, the more DOF an element has, the more expensive it is), and / or its accuracy.

In many cases the choice of the best element for a particular application may not be obvious. For example, in the model of a space frame, you may choose to use CROD elements if end moments are unimportant or to use CBAR elements if end moments are important. You may choose to use CBEAM elements with warping if the members have open cross sections and torsional stresses are estimated to be significant. You may even choose to represent the members with built-up assemblies of plate or solid elements. The choice of which type and number of elements to use depends primarily on your assessment of the effects that are important to represent in your model and on the cost and accuracy you are willing to accept.

In this context, it is critical that you have a fairly good idea of how the structure will behave prior to generating your finite element model. The best source of such insight is usually experience with similar structures. In other words, understanding the load path is crucial in the selection of the appropriate element. In addition, a few hand calculations can usually provide a rough estimate of stress intensities. Such calculations are always recommended. If you do not have a fairly good idea of how the structure will behave, you may be misled by incorrect results due to errors or incorrect assumptions in your input data preparation.

The following guidelines are provided to help you in selecting the “right” element for your task.

General Guidelines

Always experiment with a small test model when using elements that you are not familiar with. This practice is cheaper than experimenting with a large production model, and it gives you a better understanding of an element’s capabilities and limitations prior to applying it to a large production model.

Avoid using obsolete elements. Elements are classified as obsolete only if their utility is completely superseded by newer elements and after the newer elements are in service long enough to be reliable. In most cases, they are retained in MSC.Nastran only to satisfy the requirements for upward compatibility. Elements that are considered obsolete are no longer included in any updated documentation.

Zero-Dimensional Elements

When you use CELASi elements to represent concentrated springs between two components of translation, the directions of the two components must be coaxial. Even small deviations in direction can induce a significant moment to your model that does not exist in your physical structure. It is recommended that when a CELASi element is used,

the locations of the two end points be coincident in order to avoid this type of problem. If the two end points are not coincident, you should consider using a CROD or CBUSH element instead.

One-Dimensional Elements

If only an axial and / or torsion load is to be transmitted in an element, then the CROD is the easiest element to use.

A CBAR is easier to use than a CBEAM element. The I1 and / or I2 values can be set to zero.

Use the CBEAM element instead of the CBAR element if any of the following features is important:

- The cross-sectional properties are tapered.
- The neutral axis and shear center do not coincide.
- The effect of cross-sectional warping on torsional stiffness is critical.
- The difference in the mass center of gravity and the shear center is significant.

The formulation for the CBEAM element is based on a flexibility approach; the element stiffness matrix is generated by inverting the flexibility matrix. For this reason, I1 and I2 must not be zero for the CBEAM element.

Two-Dimensional Elements

In general, quadrilateral elements (CQUAD4 and CQUAD8) are preferred over the triangular elements (CTRIA3 and CTRIA6). The CTRIA3 element is a constant strain element. It is excessively stiff, and when used alone, it is generally less accurate than the CQUAD4 element, particularly for membrane strain. Whenever feasible, the CQUAD4 element should be used instead of the CTRIA3 element. CTRIA3 should only be used when necessary for geometric or topological reasons, for example, mesh transition between regions of quadrilateral elements with different meshes or near the polar axis of a spherical shell.

Avoid using CTRIA3 in locations where the membrane stresses are changing rapidly, for example, in the web of an I-beam. Since CTRIA3 has constant membrane stresses, a large number of them may be needed to obtain acceptable accuracy. It is better to use quadrilateral elements or CTRIA6 elements, if possible.

Do not use plate or shell elements (CQUAD*i*, CTRIA*i*) in stiffened shell structures with very thin panels that can buckle. Shear panels (CSHEAR) should be used in this case or in any situation where direct stresses cannot be supported, such as in a very thin curved panel.

Avoid highly skewed elements (see [Figure 9-1](#)). The angle α should be as close to 90 degrees as possible.

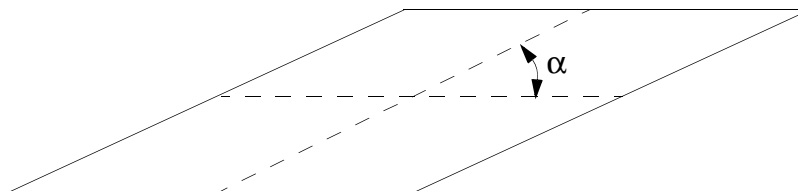


Figure 9-1 Highly Skewed Element

For the CTRIA3 element, the skew test is based upon the three vertex angles.

Aspect ratio is defined as l/ω (length / width). Very high aspect ratio (see [Figure 9-2](#)) should also be avoided, although it is no longer true that accuracy degrades rapidly with aspect ratios as it once did with some of the obsolete elements.

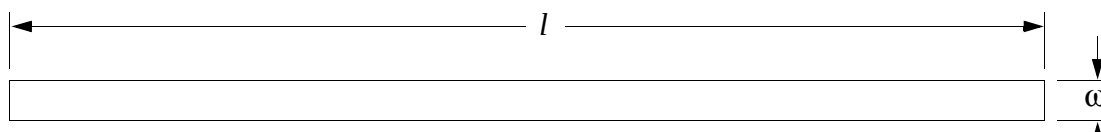


Figure 9-2 Element with High Aspect Ratio

Warping is a measure of the amount the element deviates from being planar (see [Figure 9-3](#)). Element warping should be minimized.

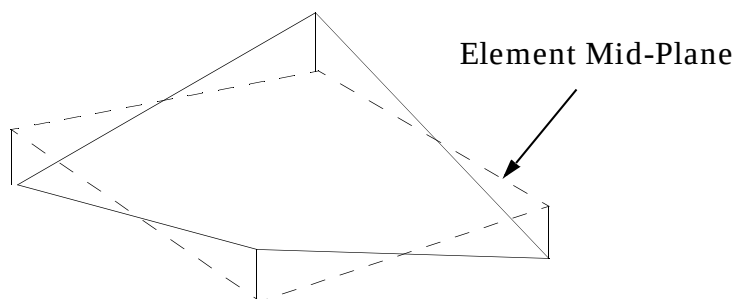


Figure 9-3 Highly Warped Element

For the CQUAD8 elements, if midside nodes are present, they should be located within the middle third of the edge. If a midside node is located at one-fourth the distance of the edge as measured from either corner node on the edge, the internal strain field becomes singular at the corners of the element. For best results, it is recommended that the midside node be located as close to the center of the edge as possible. If midside nodes are desired, the general recommendation is to include all of them. A CQUAD8 element with midside nodes deleted is excessively stiff and therefore is inferior to a CQUAD4 element.

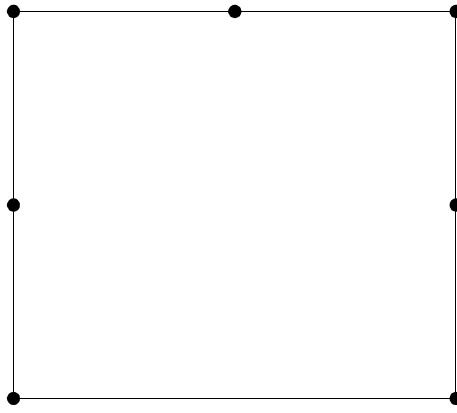


Figure 9-4 CQUAD8 with Missing Midside Node

For single curved structures (e.g., a cylinder), the CQUAD8, in general, yields better results than the CQUAD4 element. For doubly curved structures (e.g., a spherical dome), the CQUAD4 element, in general, performs better than the CQUAD8 (see Reference [15](#)).

The shell normal (param,snorm,x) should be turned on when using the CQUAD4, CTRIA3, CQUADR, or CTRIAR element. See the next section on Shell Normals for further details.

The membrane properties for the CQUADR and CTRIAR elements are less sensitive to the element shape than the CQUAD4 and CTRIA3. The same thickness should be used for all four corners ($T1 = T2 = T3 = T4$, or use the T Field on the PSHELL entry). The CQUADR and CTRIAR elements can be used in conjunction with each other, but not with other elements. They do not allow for membrane-bending coupling, the z-offset feature, nor the heat transfer capability. CQUADR and CTRIAR should not be used for buckling or nonlinear analysis.

Shell Normals

By default, the direction of the normal rotation vector for flat plate elements is assumed to be perpendicular to the plane of each element. If the model is curved, the shell bending and twist moments must change direction at the element intersection. If transverse shear flexibility is present, the deformations may be too large. (Because elements using low-order formulations ignore the edge effect, this rarely causes any problems—a large value of the parameter K6ROT partially cures the problem.) With the unique normal (SNORM) option, the rotational degrees of freedom at each corner of an element are measured relative to the specified normal vector direction. Thus, all elements connected to a grid point will use a consistent direction for defining shell bending and twisting moments.

In CQUAD4 and CTRIA3 elements, the stiffness matrices of the elements are modified to eliminate the undesirable small stiffness in the rotational motions about the shell normal vector. In effect, the transformation replaces the normal moments with in-plane forces. No changes were made to the basic element stiffness matrix, and therefore flat plate models will not be affected. The objective of the new transformation was to remove a potential weakness in curved shell models and allow the automatic constraint process to remove the true singularity in the assembled stiffness matrix.

The CQUADR and CTRIAR elements are also improved for some types of shell problems. Because of their extra degrees of freedom, these elements are more sensitive to the coupling between in-plane and out-of-plane motion due to curvature. The new formulation provides more consistency between adjacent elements in a curved shell.

Shell normals are available for CQUAD4, CQUADR, CTRIA3, and CTRIAR elements. Normals are activated if the actual angle between the local element normal and the unique grid point normal is less than 20° , the default value for β (see [Figure 9-5](#).) The default for β can be changed by setting `PARAM,SNORM, $\beta$` to the desired real value up to 89 degrees. The unique grid point normal is the average of all local shell element normals at a specific grid point. Generated grid point normals may be overwritten by user-defined normals.

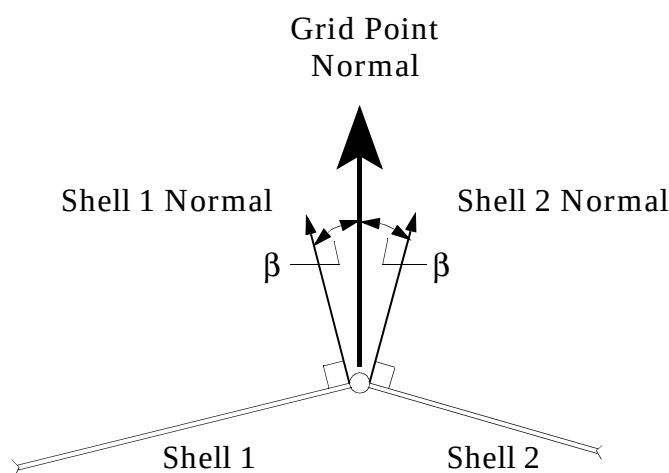


Figure 9-5 Unique Grid Point Normal

A shell normal defines a unique direction for the rotational degrees of freedom of all adjacent elements. A shell normal vector is created by averaging the normal vectors of the attached elements. If the actual angle is greater than the value defined on `PARAM,SNORM`, (default = 20.) the edge is assumed to be a corner, and the old method is used. Shell normals improve the accuracy of the results in curved shells where in-plane shear and twisting moments act together.

If the actual shell normal vector is known, such as in a cylinder or sphere, the automatically calculated values may be superseded by the actual vectors using the Bulk Data entry

`SNORM,GID,CID,V1,V2,V3`

where:

GID is a unique grid point,

CID is the coordinate system for defining the shell normal vector,

V1, V2, and V3 are unscaled components of the vector in the coordinate system.

A second parameter, `SNORMPRT`, controls the print or punch of values of the internal shell normals. The output format is the same as the `SNORM` Bulk Data so that the individual values may be used and modified on a subsequent job.

Remarks and Recommendations

The following remarks are applicable to the shell normal option:

1. The type of structure that exhibits the most change in results is a thick curved shell with large in-plane shear forces and twisting moments.
2. Most other problems, such as flat plates and curved shells with pressure loads, show changes in results of less than 1%. More degrees of freedom may be constrained using this improved formulation. This formulation results in zero in-plane rotational stiffness values.
3. It has been observed from testing that the automatic normal vector calculations produced answers nearly equal to and as accurate as those using the explicit SNORM vector input. In other words, the results were insensitive to small differences in the direction of the vectors. The important fact is that the shell normal vector requires the connected elements to use a consistent normal direction.
4. The CQUAD8 and CTRIA6 elements are not included in the shell normal processing. If they are modeled correctly, they do not require shell normal processing. Connecting these elements to the lower-order flat elements is not recommended.
5. Curved shell elements have no stiffness in the rotational degree of freedom when the new formulation is used-consequently, mechanisms may be introduced. Mechanisms occur when elements, RBEs, or MPCs are connected to the out-of-plane rotation of the shell grid points. Note that PARAM,AUTOSPC,YES does not constrain these mechanisms.
6. In linear solution sequences, the values of “param,k6rot,0.” and “param,snorm,20.” are the default.
7. Transverse shear flexibility (MID3 on the PSHELL property entry) should be left on when normals are used.

Three-Dimensional Elements

Although the CHEXA and CPENTA elements are designed to behave reasonably well as thin shell elements, it is recommended that they not be used in this capacity. The high ratio of extensional stiffness in the direction normal to the effective transverse shear stiffness can produce significant round-off errors.

As in the case of plate elements, if midside nodes are present for the solid elements, they should be located as close to the center of the edge as possible. Again, if midside nodes are desired, the general recommendation is to include all of them.

R-Type Elements

A high degree of precision must be maintained when specifying coefficients for MPCs in order to avoid introduction of unintentional constraints to rigid body motions. Rigid elements (e.g., the RBE2, RBAR, etc.) should be used whenever possible because their constraint coefficients are internally calculated to a high precision. Furthermore, these R-type elements require much less user interaction.

9.3 Mesh Density

The mesh density in a finite element model is an important topic because of its relationship to accuracy and cost. In many instances, the minimum number of elements is set by topological considerations, for example, one element per member in a space frame or one element per panel in a stiffened shell structure. In the past, when problem size was more severely limited, it was not uncommon to lump two or more frames or other similar elements in order to reduce the size of the model. With computers becoming faster and cheaper, the current trend is to represent all major components individually in the finite element model.

If the minimum topological requirements are easily satisfied, the question remains as to how fine to subdivide the major components. The question is particularly relevant for elastic continua, such as slabs and unreinforced shells. In general, as the mesh density increases, you can expect the results to become more accurate. The mesh density required can be a function of many factors. Among them are the stress gradients, the type of loadings, the boundary conditions, the element types used, the element shapes, and the degree of accuracy desired.

The grid point spacing should typically be the smallest in regions where stress gradients are expected to be the steepest. **Figure 9-6** shows a typical example of a stress concentration near a circular hole. The model is a circular disk with an inner radius = a and an outer radius = b . A pressure load p_i is applied to the inner surface. Due to symmetry, only half of the disk is modeled. In the example, both the radial stress and the circumferential stress decrease as a function of $1/r^2$ from the center of the hole. The error in the finite element analysis arises from differences between the real stress distribution and the stress distribution within the finite elements.

Three different mesh densities are used in this example as shown in **Figure 9-6**. The first one is a coarse mesh model with the elements evenly distributed. The second model consists of the same number of elements; however, the mesh is biased toward the center of the hole. The third model consists of a denser mesh with the elements evenly distributed. These three models are then analyzed with three different element types-CQUAD4, CQUAD8, and CQUAD4 with the corner stress option. The circumferential stress at the inner radius is always greater than p_i , which is the applied pressure load at the inner radius, and approaches this value as the outer radius becomes larger. The theoretical circumferential stress σ_θ (see Timoshenko and Goodier, *Theory of Elasticity*, Reference 3.) is given by the following equation:

$$\sigma_\theta = \frac{p_i a^2 \left(1 + \frac{b^2}{r^2} \right)}{(b^2 - a^2)}$$

where:

a = inner radius

b = outer radius

r = radial distance as measured from the center of the disk

p_i = pressure applied at the inner radius

The stresses are then plotted as a function of the radius in a nondimensional fashion—stress $p_i /$ versus r/a . The results are summarized in [Table 9-1](#) and [Figure 9-7](#).

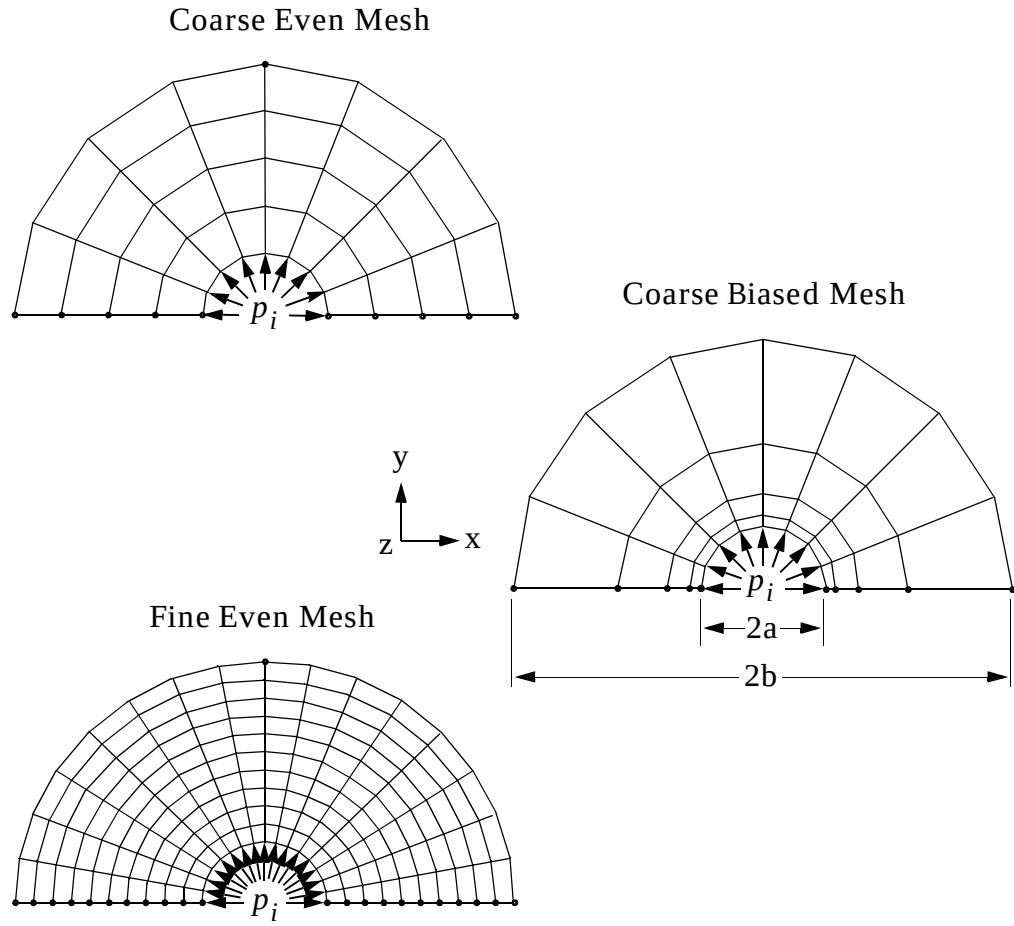
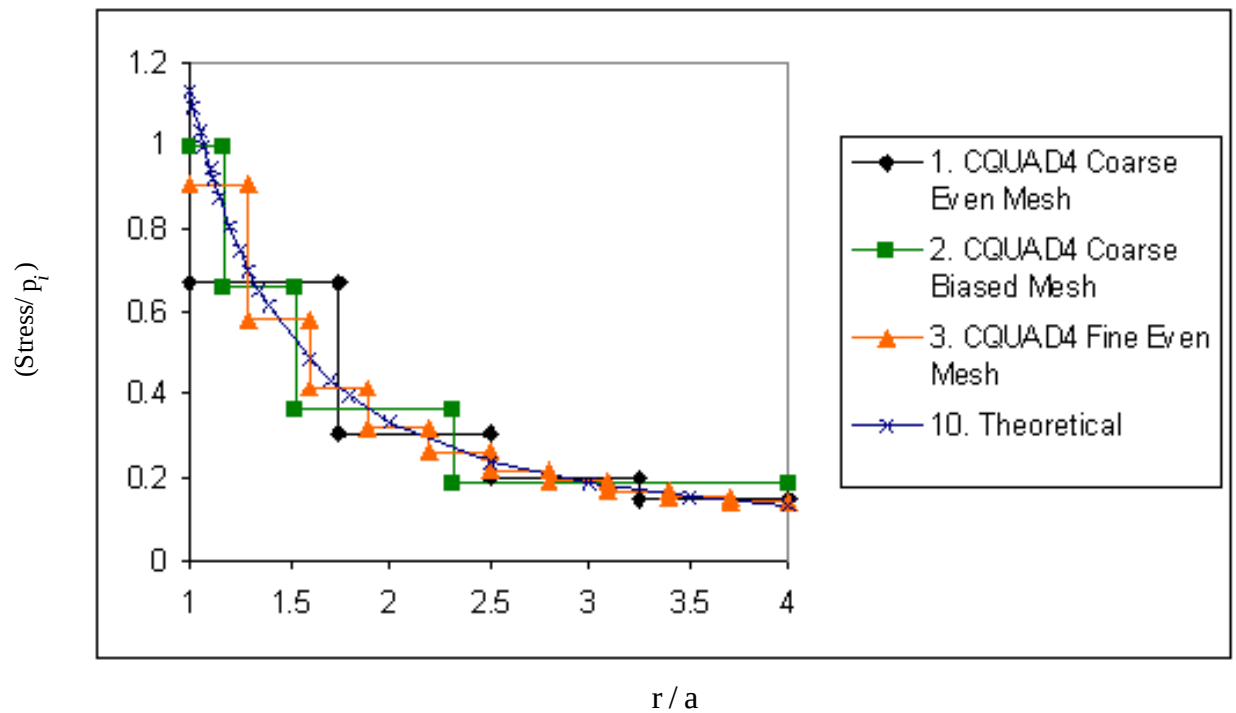
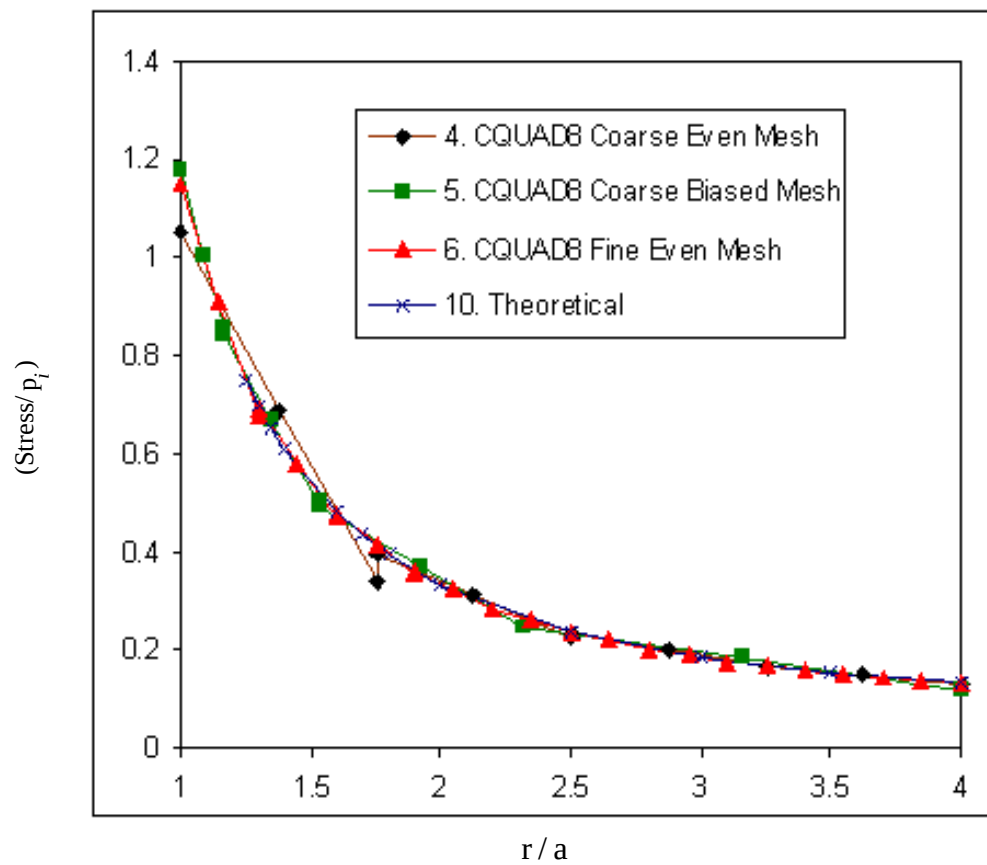


Figure 9-6 Circular Disk with Different Meshes



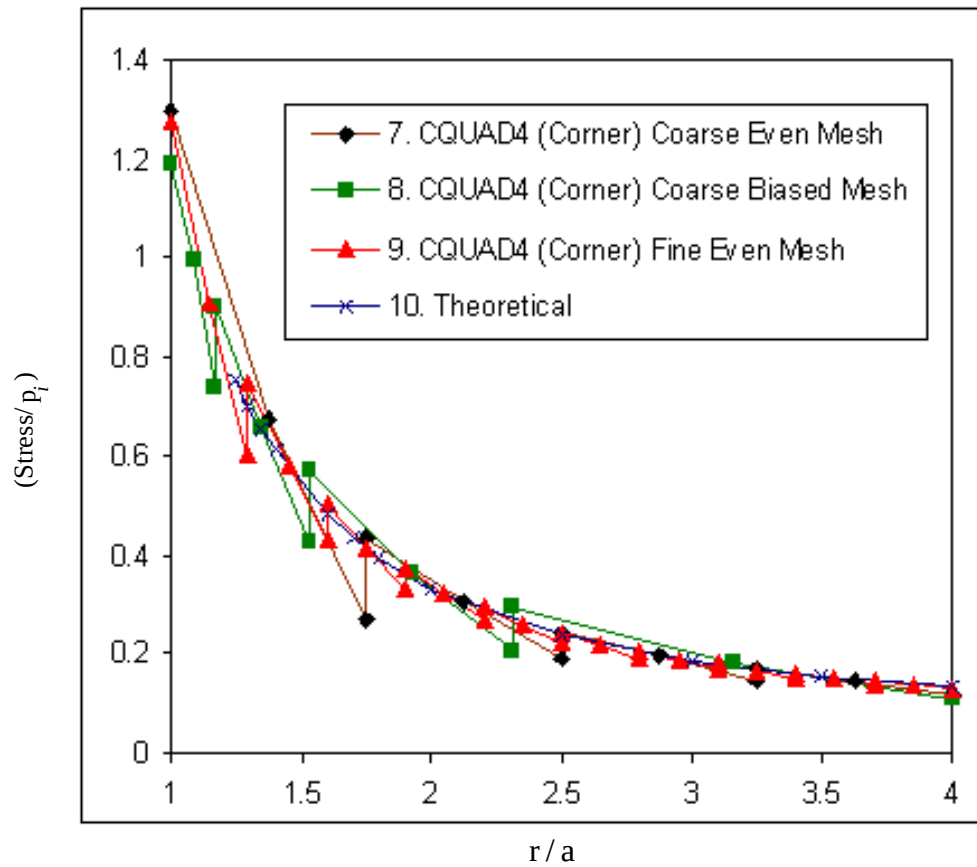
(a)

Figure 9-7 Stresses with Different Elements and Meshes



(b)

Figure 9-7 Stresses with Different Elements and Meshes (continued)



(c)

Figure 9-7 Stresses with Different Elements and Meshes (continued)

Table 9-1 Stresses Close to $r = a$ for a Circular Disk

Case No.	Element Type	DOF (L-Set)	Description	Stress Theoretical
1	CQUAD4	194	Coarse, Even Mesh	0.594
2	CQUAD4	194	Coarse, Biased Mesh	0.881
3	CQUAD4	868	Fine, Even Mesh	0.801
4	CQUAD8	550	Coarse, Even Mesh	0.929
5	CQUAD8	550	Coarse, Biased Mesh	1.041
6	CQUAD8	2538	Fine, Even Mesh	1.015
7	CQUAD4 with Corner Option	194	Coarse, Even Mesh	1.142
8	CQUAD4 with Corner Option	194	Coarse, Biased Mesh	1.047
9	CQUAD4 with Corner Option	868	Fine, Even Mesh	1.127
10	THEORETICAL	--	--	1.000

For this particular case, since the stresses are proportional to $1/r^2$, you expect the highest stress to occur at the inner radius. In order to take advantage of this piece of information, the obvious thing to do is to create a finer mesh around the inner radius. Looking at the results for the first two cases in [Table 9-1](#), it is quite obvious that just by biasing the mesh, the results are 30% closer to the theoretical solution with the same number of degrees of freedom.

A third case is analyzed with a finer but unbiased mesh. It is interesting to note that for case number 3, even though it has more degrees of freedom, the result is still not as good as that of case number 2. This poor result is due to the fact that for the CQUAD4 element, the stresses, by default, are calculated at the center of the element and are assumed to be constant throughout the element. This assumption is best illustrated by the results shown in [Figure 9-7\(a\)](#). Looking at [Figure 9-6](#), it is also obvious that the centers of the inner row of elements are actually further away from the center of the circle for case number 3 as compared to case number 2. The results for case number 3 can, of course, be improved drastically by biasing the mesh.

You can request corner outputs (stress, strain, and force) for CQUAD4 in addition to the center values (“[Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#)” in Chapter 3 of the *MSC.Nastran Reference Guide*). Corner results are extrapolated from the corner displacements and rotations by using a strain rosette analogy with a cubic correction for bending. The same three models are then rerun with this corner option--their results are summarized in cases 7 through 9. Note that the results can improve substantially for the same number degrees of freedom.

Corner output is selected by using a corner output option with the STRESS, STRAIN, and FORCE Case Control commands. When one of these options is selected, output is computed at the center and four corners for each CQUAD4 element, in a format similar to that of CQUAD8 and CQUADR elements. See Reference [8](#). for more information.

There are four corner output options available: CORNER, CUBIC, SGAGE, and BILIN. The different options provide for different approaches to the stress calculations. The default option is CORNER, which is equivalent to BILIN. BILIN has been shown to produce better results for a wider range of problems.

To carry it a step further, the same three models are then rerun with CQUAD8 (cases 4 through 6). In this case, the results using CQUAD8 are better than those using the CQUAD4. This result is expected since CQUAD8 contains more DOFs per element than CQUAD4. Looking at column three of [Table 9-1](#), you can see that due to the existence of midside nodes, the models using CQUAD8 contain several times the number of DOFs as compared to CQUAD4 for the same number of elements. The results using CQUAD4 can, of course, be improved by increasing the mesh density to approach that of the CQUAD8 in terms of number of DOFs.

It is important to realize that the stresses are compared at different locations for Cases 1 through 3 versus Cases 4 through 9. This difference occurs because the stresses are available only at the element centers for Cases 1 through 3, but the stresses are available at

the corners as well as the element centers for Cases 4 through 9. When looking at your results using a stress contour plot, you should be aware of where the stresses are being evaluated.

How fine a mesh you want depends on many factors. Among them is the cost you are willing to pay versus the accuracy you are receiving. The cost increases with the number of DOFs. The definition of cost has changed with time. In the past, cost is generally associated with computer time. With both hardware and software becoming faster each day, cost is probably associated more with the time required for you to debug and interpret your results. In general, the larger the model is, the more time it takes you to debug and interpret your results. As for acceptable accuracy, proceeding from case 8 to case 6, the error is reduced from 4.7% to 1.5%; however, the size of the problem is also increased from 194 to 2538 DOFs. In some cases, a 4.7% error may be acceptable. For example, in cases in which you are certain of the loads to within only a 10% accuracy, a 4.7% error may be acceptable. In other cases, a 1.5% error may not be acceptable.

In general, if you can visualize the form of the solution beforehand, you can then bias the grid point distribution. However, this type of information is not necessarily available in all cases. If a better assessment of accuracy is required and resources are available (time and money), you can always establish error bounds for a particular problem by constructing and analyzing multiple mesh spacings of the same model and observe the convergences. This approach, however, may not be realistic due to the time constraint.

The stress discontinuity feature, as described in “[Model Verification](#)” in Chapter 10, may be used for accessing the quality of the mesh density for the conventional h-version elements.

With the introduction of p-version element and p-version adaptivity in Version 68, the mesh refinement can be performed “automatically” inside MSC.Nastran. Discussion of the p-version element is beyond the scope of this user’s guide. However a basic definition of the h-version and p-version elements is warranted.

h-elements. In traditional finite element analysis, as the number of elements increases, the accuracy of the solution improves. The accuracy of the problem can be measured quantitatively with various entities, such as strain energies, displacements, and stresses, as well as in various error estimation methods, such as simple mathematical norm or root-mean-square methods. The goal is to perform an accurate prediction on the behavior of your actual model by using these error analysis methods. You can modify a series of finite element analyses either manually or automatically by reducing the size and increasing the number of elements, which is the usual h-adaptivity method. Each element is formulated mathematically with a certain predetermined order of shape functions. This polynomial order does not change in the h-adaptivity method. The elements associated with this type of capability are called the h-elements.

p-elements. A different method used to modify the subsequent finite element analyses on the same problem is to increase the polynomial order in each element while maintaining the original finite element size and mesh. The increase of the interpolation order is internal, and the solution stops automatically once a specified error tolerance is satisfied. This

method is known as the p-adaptivity method. The elements associated with this capability are called the p-elements. For more details on the subject of the p-version element, refer to “**p-Elements**” on page 216 of the *MSC.Nastran Reference Guide*.

9.4 Mesh Transitions

Mesh transition can be a complicated subject. It may simply be used to refine the mesh in a particular area, connect different element types (for example, a CBAR element to a solid element), or provide transitions required to model the geometry of the structure. Two guidelines for mesh transitions are as follows:

- Never place a mesh transition in an area of interest or in an area where there is a large variation in stress.
- Mesh transitions should be located away from the areas of interest in a region.

Due to incompatibilities between finite element types, any transition between different element types (even a transition from CQUAD4 to CTRIA3 elements) can result in local stress anomalies. Normally, these stress anomalies are localized and dissipate quickly as you move away from the transition. However, a problem arises when the transition occurs in an area of interest. In this case, the local stress rises (or decreases) due to the effect of the transition; in other words, the results may be conservative (or unconservative) in an area near a transition. However, if this localized stress variation occurs away from areas of interest, the increase (or decrease) in stress caused by the transition should cause no concern.

Transition from a Coarse Mesh to a Fine Mesh

The transition from a coarse mesh to a fine mesh, or vice versa, may not always be an easy task. One common method of performing a transition is to use an intermediate belt of triangular elements as shown in [Figure 9-8\(a\)](#). It is also very tempting to selectively delete midside nodes of higher-order elements (e.g., CQUAD8) and use them as transition elements as shown in [Figure 9-8\(b\)](#). This method is not recommended because it severely distorts the stress distribution in the elements adjacent to the change in mesh size.

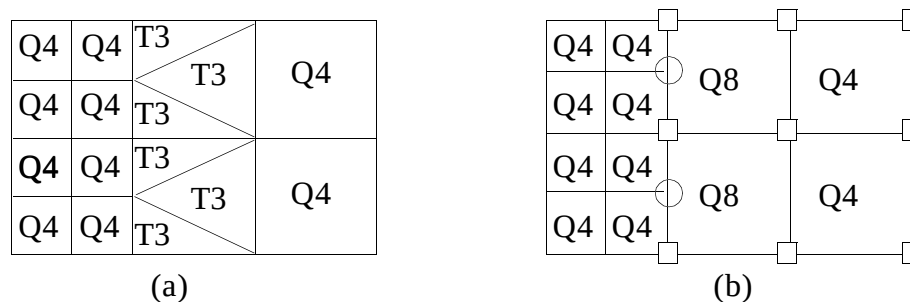


Figure 9-8 Transition Methods

When using a higher-order element, such as a CQUAD8 or a CHEXA (nine to 20 nodes) for any purpose, the general recommendation is to include all midside nodes. A CQUAD8 element with all midside nodes deleted is excessively stiff and therefore is inferior to a CQUAD4. For the CHEXA element, you should use it with eight nodes (no midside nodes), or use it with 20 nodes (all midside nodes). The midside nodes, if used, should be located as close to the center of the edge as possible.

As an alternative mesh transition method, a spline element can be used. In MSC.Nastran, the RSPLINE element is an elastic interpolation element that connects N end points. Displacements for intermediate grid points are interpolated from the equations of an elastic beam passing through these grid points. The spline element assumes a linear interpolation for displacement and torsion along the axis of the spline, a quadratic interpolation for rotations normal to the axis of the spline, and a cubic interpolation for displacements normal to the axis of the spline.

Consider an example problem as shown in **Figure 9-9**. This problem involves a plate fixed at the left end with a dimension of 14 in by 9 in, and modeled with 1 in by 1 in CQUAD4 elements. For clarity, only grid points of interest are labeled. Five different loadings-as shown in **Table 9-2** are applied to this structure along the free edge (right end). For convenience, a RBE3 element is created along the free edge by defining a dependent point (grid point 500) at the center. This grid point is connected to all the edge grid points. The load can then be applied to grid point 500 and distributed to the rest of the edge grid points. See “**The RBE3 Element**” on page 301 for a description of the RBE3 element.

Hypothetically consider that you are interested in the stresses along the line one inch from the fixed edge. Since you are not interested in the stresses outside of this region, you can represent the rest of the region with a coarser mesh. Two different transition methods will be used for this purpose-one with CTRIA3s (**Figure 9-10**) and one with RSPLINEs (**Figure 9-11**). The transitions are performed along the line five inches away from the fixed edge. An additional transition is performed at $x = 11$ inches in order to maintain the same grid point density on the right edge for all three models. This condition assures identical load applications for all three models. Normally, this additional transition is seldom done in a real model.

Table 9-2 Loadings Used for the Mesh Transition Test

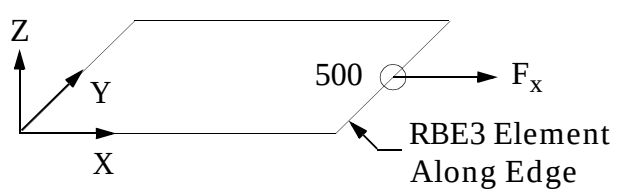
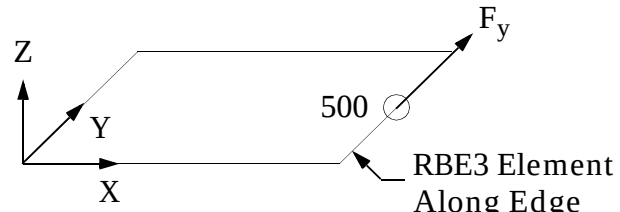
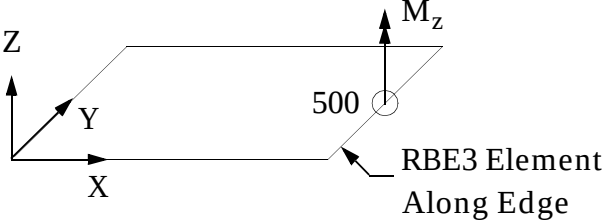
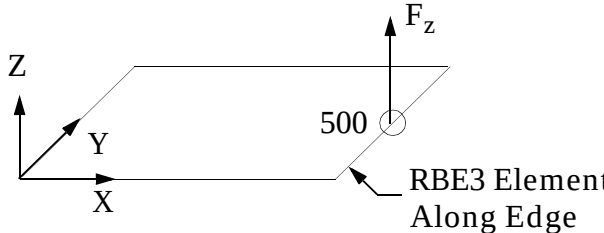
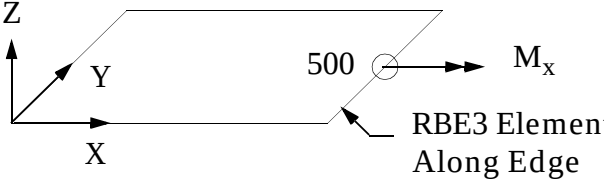
Loading Description	
1. Membrane Loads	
2. In-Plane Shear Loads (parabolic distribution)	

Table 9-2 Loadings Used for the Mesh Transition Test

3. In-Plane Bending Loads	
4. Vertical Loads	
5. Twisting Loads	

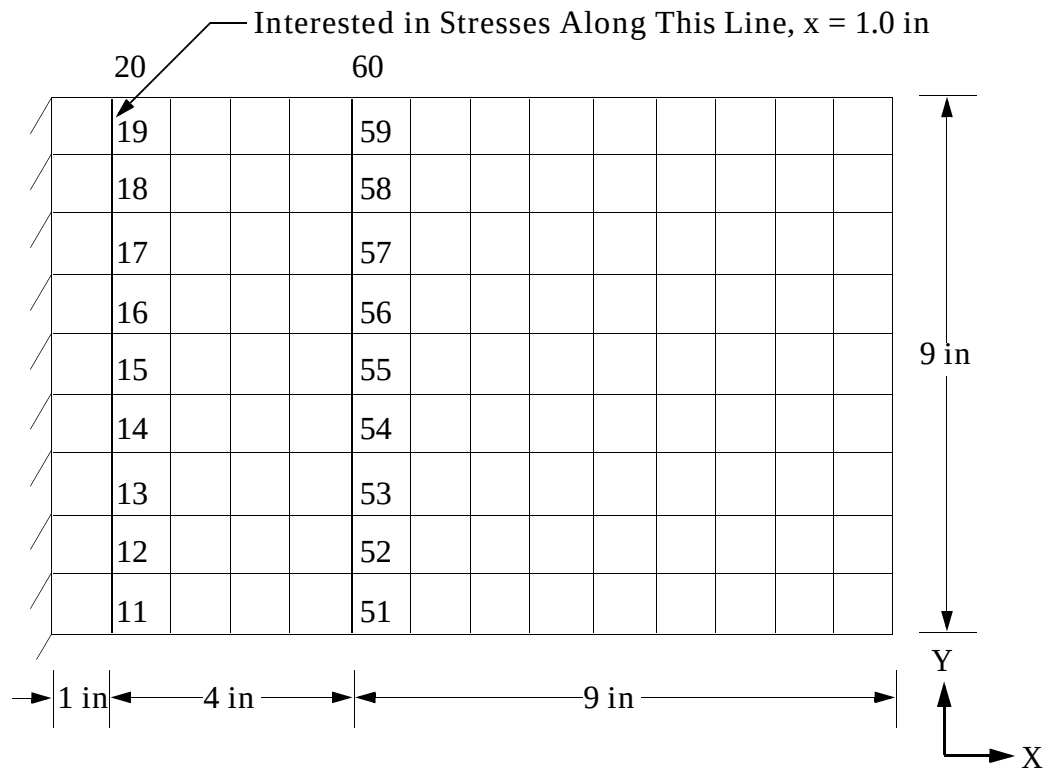


Figure 9-9 Baseline Model

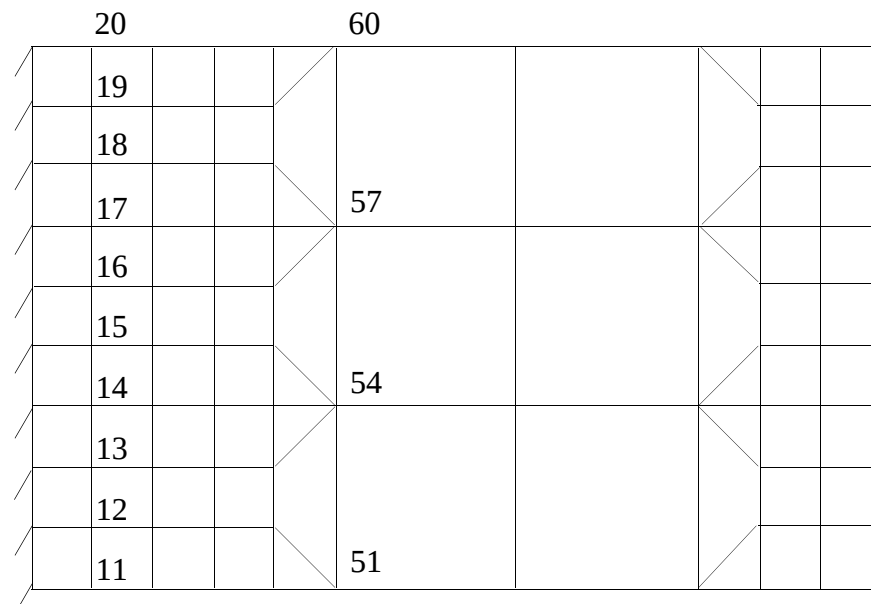


Figure 9-10 Transition with CTRIA3

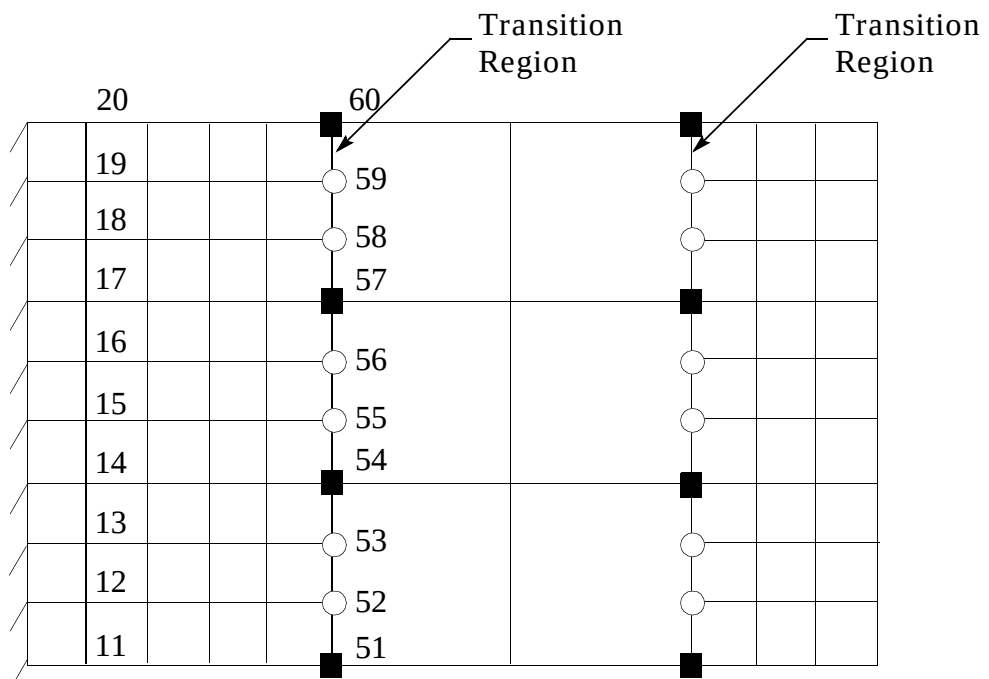


Figure 9-11 Transition with RSPLINE

The model with the mesh transition using CTRIA3s, as shown in [Figure 9-10](#), is quite straightforward and needs no further explanation. However, further discussion is warranted in the case of mesh transition using the RSPLINE element.

The input format for the RSPLINE element is as follows:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

Field	Contents
EID	Element identification number.
Gi	Grid point identification number.
Ci	Components to be constrained.

Remarks:

1. The first and last grid points must be independent.
2. Displacements are interpolated from the equations of an elastic beam passing through the grid points.

A single RSPLINE is used to connect grid points 51 through 60 as shown in [Figure 9-11](#). Grid points 51, 54, 57, and 60 are considered to be independent, while the other six points are dependent (grid points 52, 53, 55, 56, 58, and 59). This RSPLINE interpolates the motions at the intermediate points, indicated by circles in [Figure 9-11](#) (grid points 52, 53, 55, 56, 58, and 59), from the cubic equations of a beam spline passing through the points indicated by squares (grid points 51, 54, 57, and 60). The input for the RSPLINE element connecting grid points 51 through 60 is as follows:

1	2	3	4	5	6	7	8	9	10
RSPLINE	1000		51	52	12345	53	12345	54	
		55	12345	56	12345	57		58	
	12345	59	12345	60					

The results using these two transition methods are compared with the baseline model with no transition. These results are evaluated at two locations: at $x = 1.0$ in (location of interest) and at $x = 5.0$ in (transition region). The five different loading conditions mentioned previously were used. The Hencky-von Mises stresses using the grid point stress feature (“[Grid Point Stresses](#)” on page 349) and the CQUAD4 corner stress option are used for comparison. The results are shown in [Figure 9-12](#) through [Figure 9-16](#). As you can see, the results compare well, using either method of transition, with the baseline model at the area of interest (away from the transition region) for all five loading conditions.

At the transition region, depending on the loading and location, the transition using CTRIA3 elements compares better with the baseline model in some cases, whereas the transition using RSPLINE elements compares better with the baseline model in other cases. Note that there are only four stress values available in the case of using CTRIA3s since there are only four grid points along the line joining grid points 51 through 60 in [Figure 9-10](#).

The CTRIA3 elements used in this model are all regularly shaped elements; in most real-life problems, it is very likely that the transition areas may consist of CTRIA3 elements that are more distorted. In this case, the transition using the distorted CTRIA3 elements does not perform as well as the model in [Figure 9-9](#) consisting of regularly shaped CTRIA3 elements.

In either case, there are noticeable differences in the stresses at the transition region, as expected. However, at the area of interest and away from the transition region, the stresses are virtually the same for both cases as compared to the baseline model. Once again, mesh transitions should always be modeled away from the area of interest.

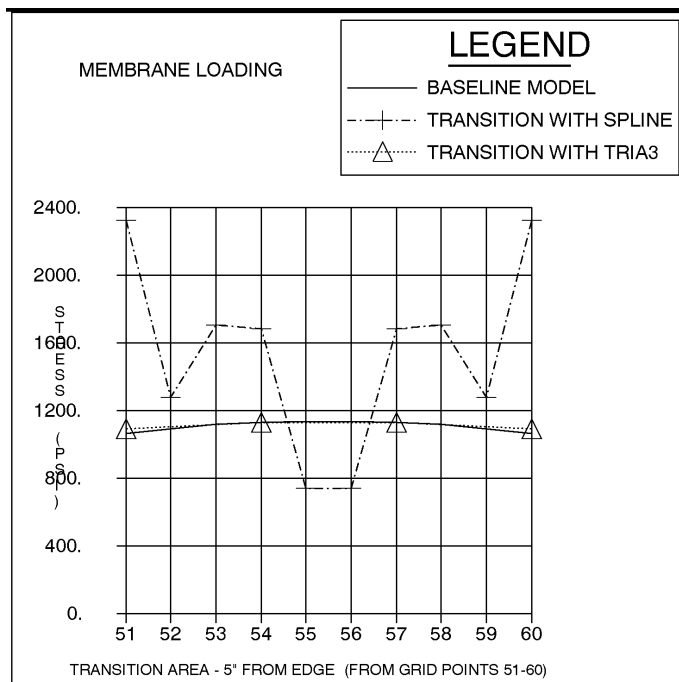
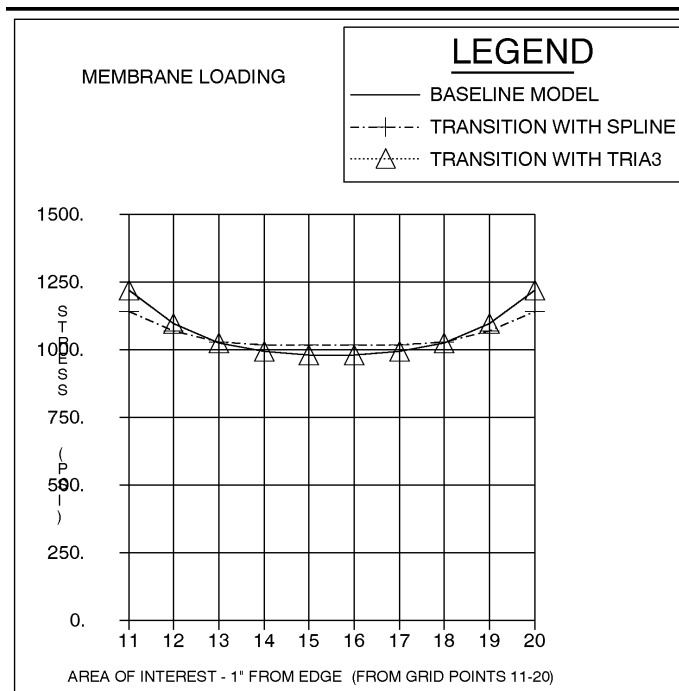


Figure 9-12 Resultant Stresses for Membrane Loads

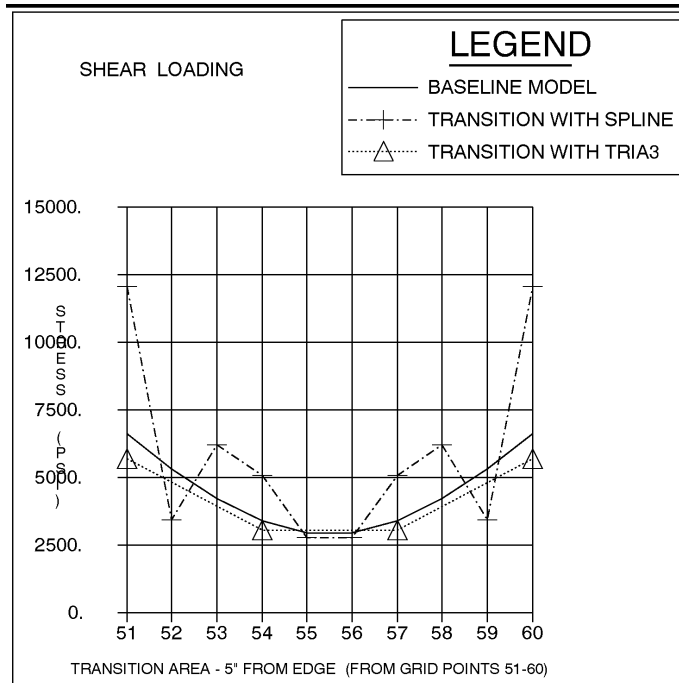
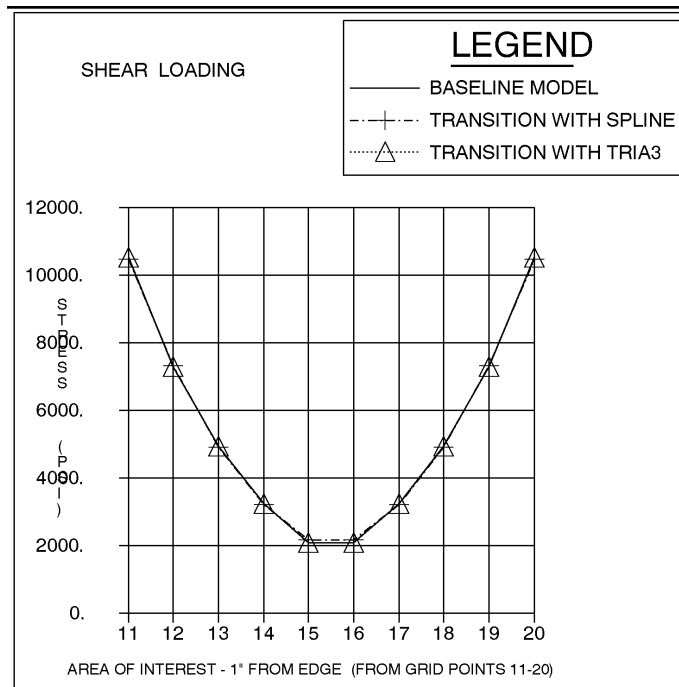


Figure 9-13 Resultant Stresses for In-Plane Shear Loads

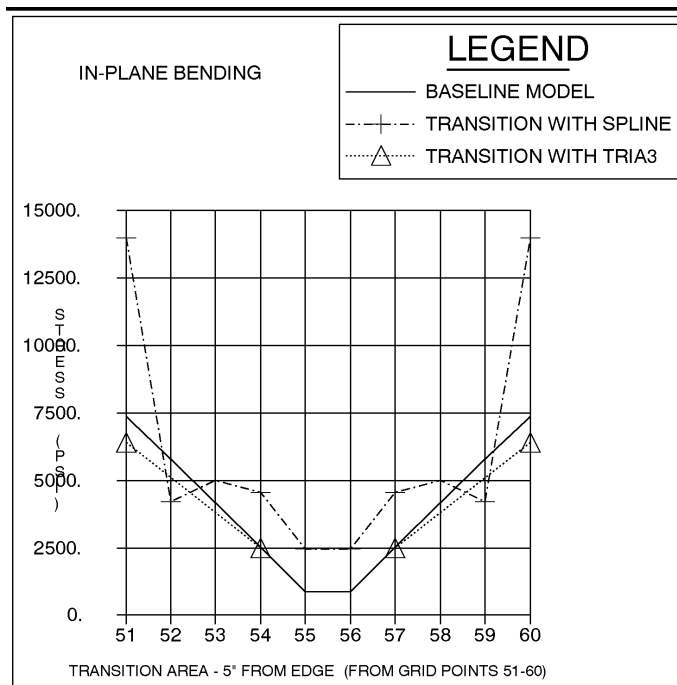
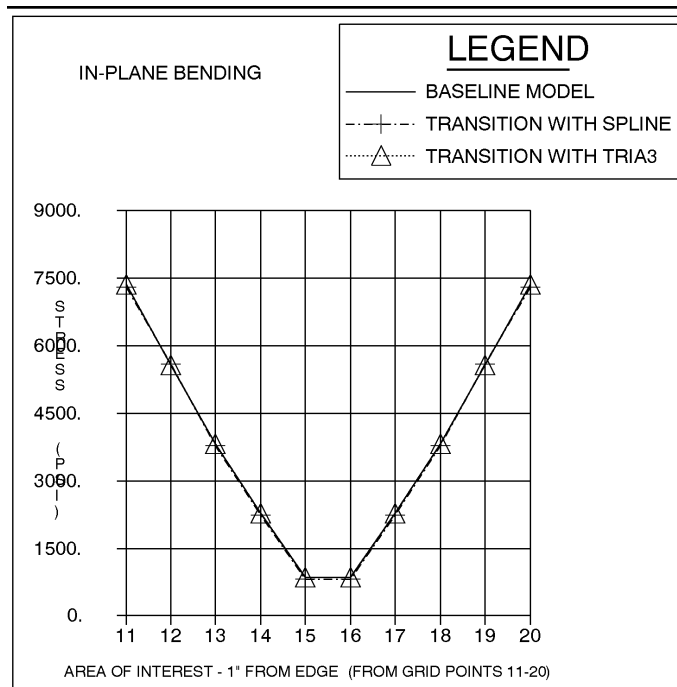


Figure 9-14 Resultant Stresses for In-Plane Bending Loads

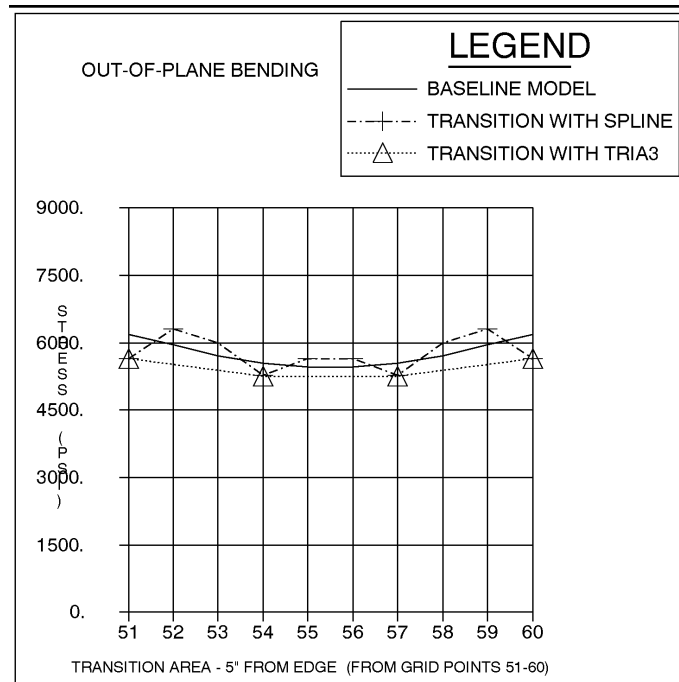
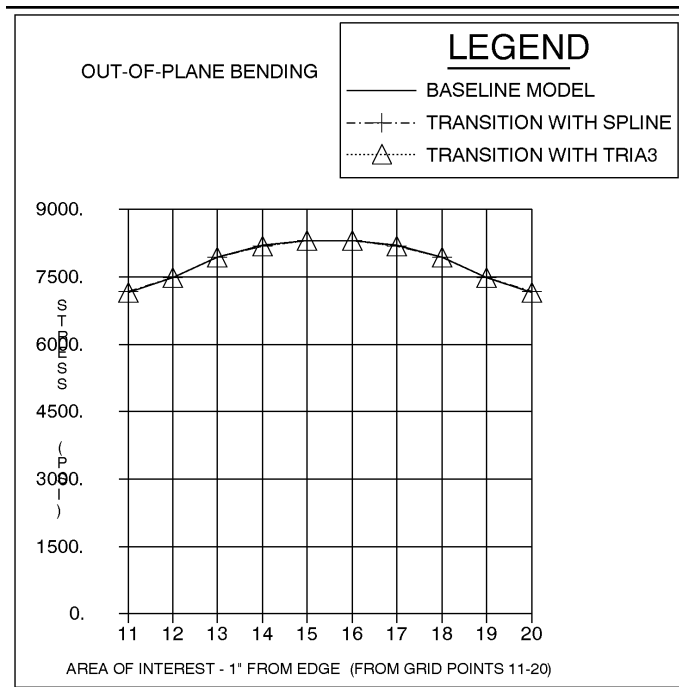


Figure 9-15 Resultant Stresses for Vertical Loads

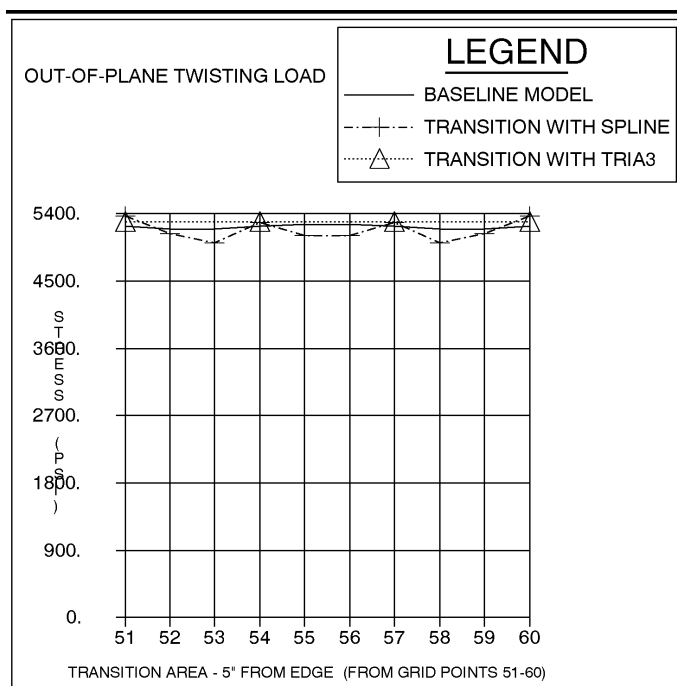
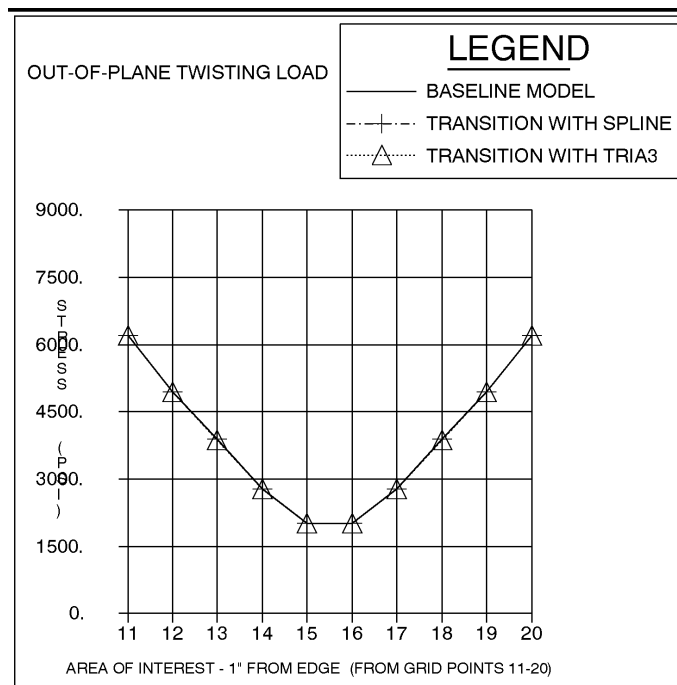


Figure 9-16 Resultant Stresses for Twisting Loads

Mesh Transition Between Dissimilar Element Types

Attaching a plate or bar element to a solid element is a case of transition between dissimilar element types. This process is more involved than it appears at first glance. Solid elements have stiffness only in the translational DOFs at the attachment grid points; they have no stiffness for rotational DOFs. A simple visualization is to think of the attachment of a solid element to a grid point as a “ball-and-socket” joint, that is, translational forces may be transmitted, but no moment may be transmitted through the connection.

This incompatibility of the element stiffness matrices represents a modeling problem whenever plate or bar elements are attached to solid elements. Both plate and bar elements have stiffness for rotational DOFs (although the plate element may not have a stiffness for the normal rotation). Therefore, special modeling must be performed whenever a plate or bar is connected to a solid element. Otherwise, the connection becomes a hinge (for plate elements) or a pinned connection (for bar elements).

Several methods are available to handle the transition between these elements. These methods range from adding extra elements (for example, adding an additional plate or bar that continues into the solid element) to using special (R-type) elements for the transition.

One method of handling this transition is to use RBE3 elements (see “**R-Type Elements**” in Chapter 8). The RBE3 is an interpolation element, which is ideally suited for this application. By using RBE3s, the rotations of the attached grid points is simply slaved to the translations of the adjacent grid points. Examples of using RBE3 elements to connect a solid element to a plate element and bar element are shown in **Figure 9-17**. The RBE3 elements attach the rotational DOFs to the translational DOFs on the solid element.

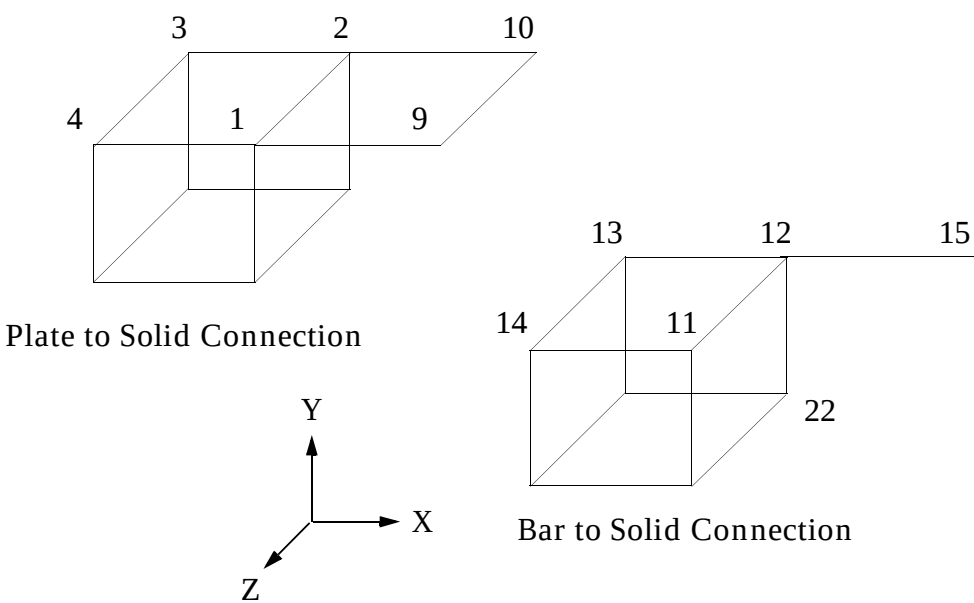


Figure 9-17 Typical Transition Between Dissimilar Elements

For the plate to solid connection, two RBE3 elements suffice:

1	2	3	4	5	6	7	8	9	10
\$RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	901		1	456	1.0	123	2	3	
	4								

RBE3	902		2	456	1.0	123	1	3	
	4								

For the bar to solid connection, one RBE3 element can make the connection:

RBE3	903		12	456	1.0	123	11	13	
	22								

These RBE3 elements transmit the loads to the independent DOFs. If RBE2 elements are used, then the connection is “rigid.”

The important thing to remember when handling these connections is that the solid elements have no stiffness for rotational DOFs, whereas the real structure does. This means that a special modeling effort is needed when any element with bending stiffness is connected to a solid element.

When using the RBE3 element, care must be taken to ensure that the independent DOFs are sufficient to transfer any applied loads. For the bar-to-solid connection in [Figure 9-16](#), if only two independent grid points are used, the element is “unstable,” that is, since only the translational DOFs are used as independent DOFs in the sample, the element is unstable for rotation about the axis connecting the two points. Therefore, three non-colinear grid points are used. A simple way to remember this is to ask, “If I constrain the DOFs that I list as independent on the RBE3, can I prevent any possible rigid body motion?” If the answer to this question is “yes,” then the RBE3 element is capable of transferring any applied loads. In this way, you can avoid possible problems in processing the RBE3 elements.

Shell-to-Solid Transition Element (RSSCON)

A new method using the RSSCON for modeling shell-to-solid transition is introduced in Version 69 (see [Figure 9-18](#)). This capability conveniently eliminates the need to define RBE3s or MPCs (multipoint constraints) to constrain a shell element’s translational and rotational degrees of freedom to a solid element’s translational degrees of freedom. When using the RSSCON capability, the shell element mesh must line up with the solid element mesh so that there is an exact element-to-element correspondence.

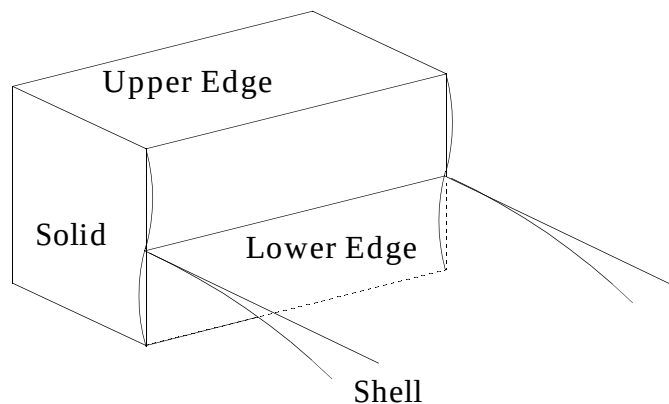


Figure 9-18 Clamped Connection of a Shell to a Solid

The RSSCON Bulk Data entry supports both p-adaptive elements and h-elements. The p-adaptive elements are outside the scope of this user's guide. Through the RSSCON Bulk Data entry, the shell elements CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR can be connected to the solid elements CHEXA, CPENTA, and CTETRA. Elements with midside nodes are also supported.

RSSCON generates a multipoint constraint, which puts the shell degrees of freedom in the dependent set (m-set). The three translational degrees of freedom and the two rotational degrees of freedom of the shell edge are connected to the three translational degrees of freedom of the upper and lower solid edge. Poisson's ratio effects and temperature loads are modeled correctly. The generated multipoint constraints produce six zero-energy modes for rigid-body motion.

The RSSCON Bulk Data entry defines the connection of a shell element to a solid element. Details of the RSSCON Bulk Data entry are described in the [MSC.Nastran Quick Reference Guide](#). Within MSC.Nastran, however, there are two options for making this connection using the RSSCON Bulk Data entry, as shown in [Figure 9-19](#):

1. The ELEM option, which allows you to specify the element ID of the shell and the element ID of the solid to which the shell is to be connected.
2. The GRID option, which allows you to specify the grid point ID of the shell and the upper and lower grid point IDs of the solid. Two triplets of grid point IDs may be specified on one RSSCON Bulk Data entry.

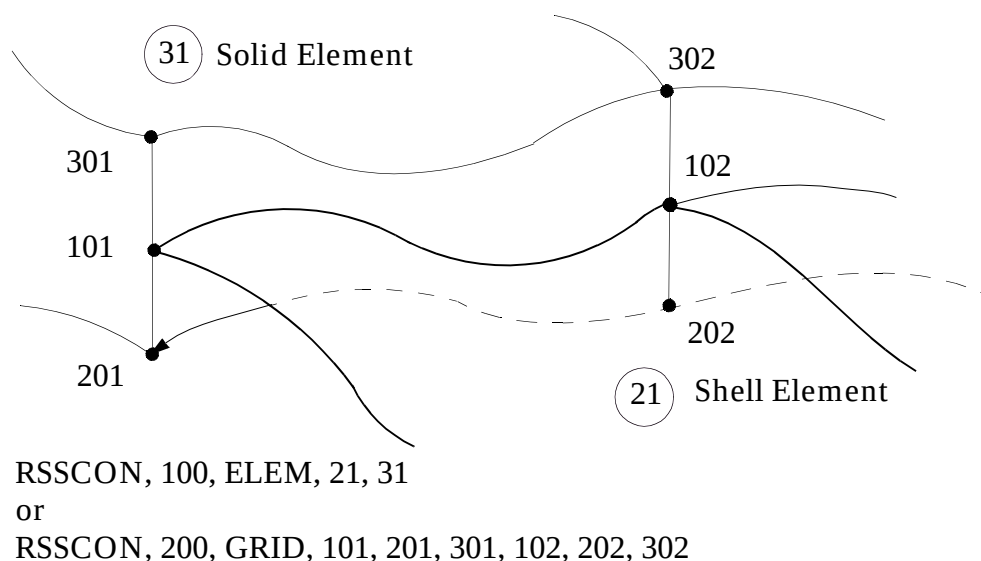


Figure 9-19 ELEM and GRID Option on the RSSCON Bulk Data Entry

The best modeling practice is illustrated in [Figure 9-20](#). The height of the connected solid element should be chosen equal to the thickness of the shell. If the height of the connected solid element is much larger than the thickness of the shell element, then the connection modeled with RSSCON will be stiffer than the continuum model. For example, in a mesh where shell grid points are identical or coincide with solid grid points, the RSSCON Bulk

Data entry may model a connection that is too stiff (see [Figure 9-21\[a\]](#)). Note also that in the model of [Figure 9-21\[b\]](#), the GRID option must be used to connect a shell element to more than one solid element.

The RSSCON connector element can accommodate a variety of mesh topologies, as shown in [Figure 9-22](#).

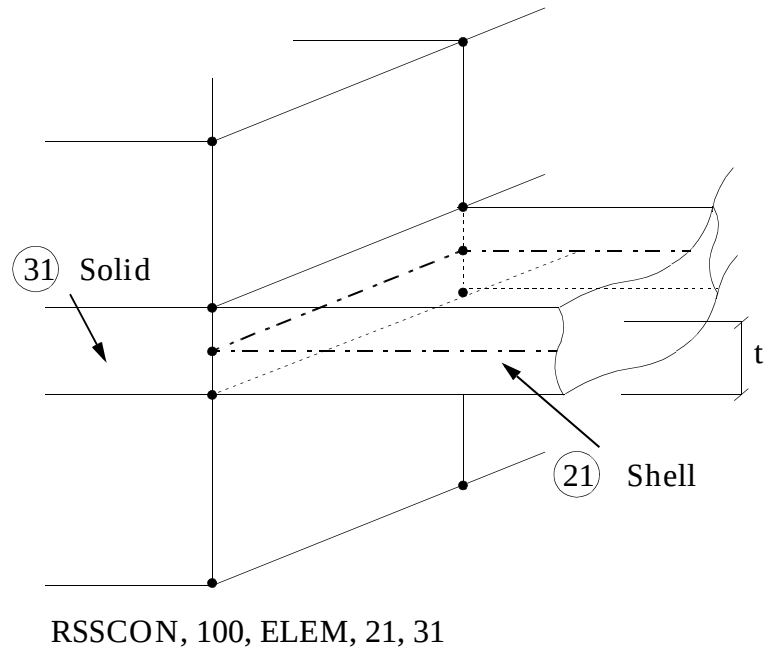


Figure 9-20 Best Modeling Practice for RSSCON

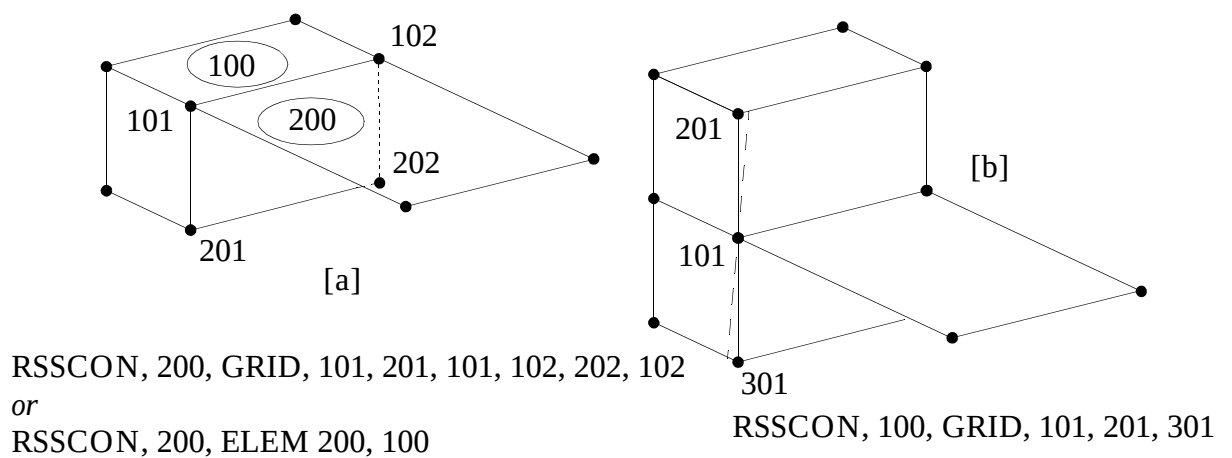


Figure 9-21 RSSCON in Meshes Where Solid and Shell Elements Share Grid Points

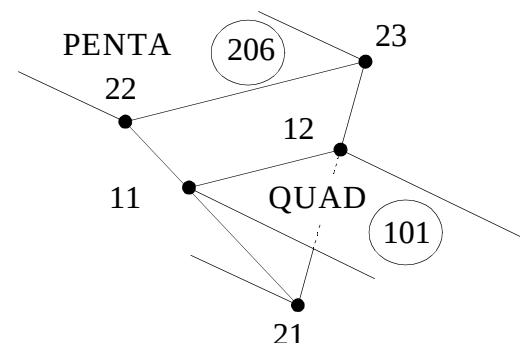
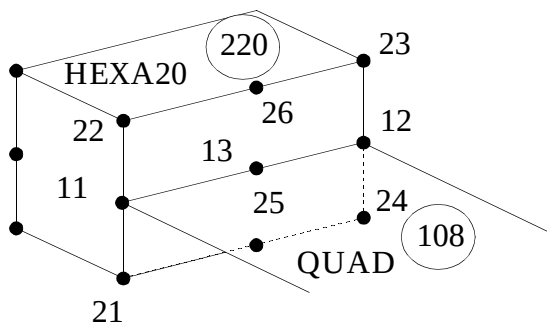
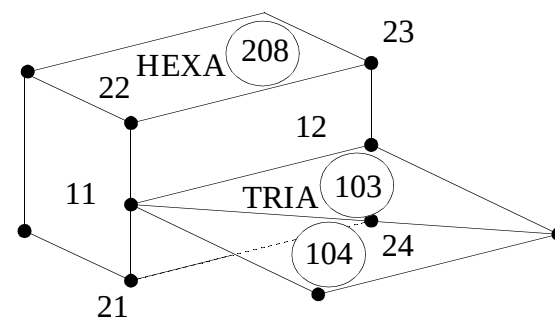
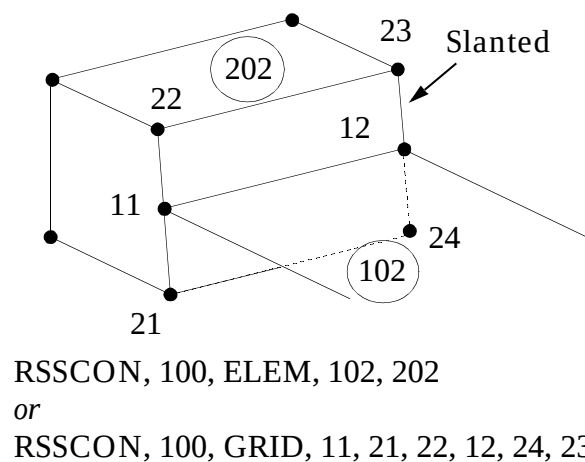


Figure 9-22 Modeling Options with RSSCON

The only modeling requirement is that the shell grid point must lie on the line connecting the lower and upper solid grid points. By default, the RSSCON allows the shell grid point to have an offset from this line of up to 5% of the distance between the two solid grid points. If a shell grid point does not lie on the line joining the two solid grid points, but within this 5% tolerance, the shell grid point will be moved to lie on the line. On the other hand, if a shell grid point lies outside this 5% tolerance, the run terminates with a fatal message. The default 5% tolerance can be modified (in Version 69.1 and newer systems) by adding "param,tolrsc, ϵ " to the Bulk Data Section, where ϵ is the desired tolerance percentage. The default value for ϵ is 0.05. The old and new location of the moved shell grid points can be printed using the parameter "param,sep1xovr,16".

When the GRID option is used, MSC.Nastran does not verify that the grid points are valid shell or solid grid points. When the ELEM option is used, if a solid-shell connection has horizontal edges that are curved in the shell plane, MSC.Nastran assumes that the

geometry of the shell element is compatible with that of the solid element. In the case of vertical edges, the shell element grid point must lie on the line between the upper and lower solid element grid point. A default offset tolerance of 5% (user modifiable via "param,tolrsc, ϵ ") of the distance between the solid grid points is allowed. For limitations of the RSSCON related to the p-elements, see “[Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#)” on page 167 of the *MSC.Nastran Reference Guide*.

By default, the moved shell grid points are not printed.

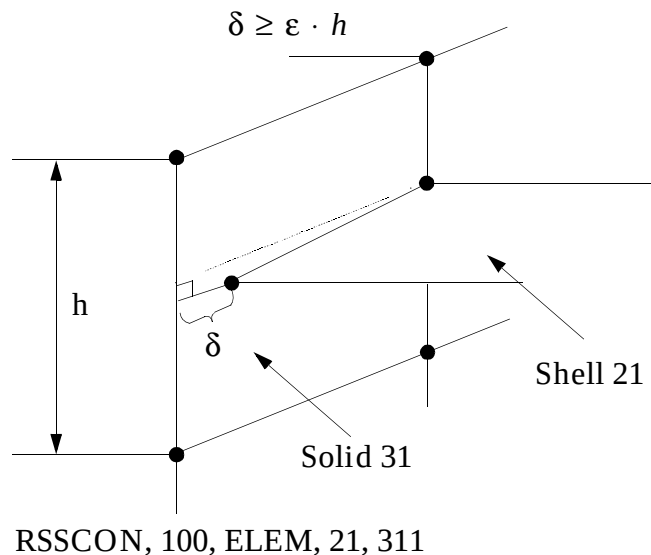


Figure 9-23 RSSCON Modeling Practice Parameters

RSSCON Modeling Recommendations

The RSSCON connector produces excellent results if the user follows good modeling practices. The geometry of the RSSCON connector is checked, and fatal messages are issued for invalid connections. Additional recommendations are given below.

1. Midside-noded elements should *only* be connected to other midside-noded elements. For example, a CQUAD8 element connected to a CHEXA20 element is acceptable; however, a CQUAD4 element connected to a PENTA15 element is not allowed. Elements with missing midside grid points are not allowed.
2. For midside-noded elements, avoid using the RSSCON where shells connect to triangular solid faces.
3. Do not mix coordinate systems when using midside nodes. In other words, when using midside nodes, do not define a local coordinate system containing all the shell grid points and the basic coordinate system to define the solid grid points. Otherwise, incorrect answers will be generated.
4. Only one edge of a single shell element may connect to a solid surface.
5. Do *not* connect more than one shell element to one solid element when using the ELEM option.
6. Do not attempt solid-shell connections to severely warped (greater than 20%) midside-noded solid elements.

7. The plane of the shell element should not be parallel to the plane of the connecting face of the solid element.
8. The connecting edge of the shell element must lie in the face of the solid element. In addition, the vertex grid points of the shell edge connecting to a quadrilateral solid face must line up with its opposite edges; for triangular solid faces, the connections may be made between any of the edges. The tolerance in the geometry may be modified; see `PARAM,TOLRSC, $\epsilon$` described above.
9. When using the `ELEM` option in the case of partitioned superelement Bulk Data, you must ensure that the solid element, `RSSCON` connector, and shell element which define a shell-solid connection are all contained within the same superelement.

Cantilever Clamped to Wall. A cantilever (4.1 in long, 0.1 in thick, 0.5 in wide) is clamped to a wall and loaded at the tip. This example demonstrates how accurate the local stress distribution is when a `RSSCON` entry is used to model the clamped condition. Four different finite element models are used, and their results are compared with classical beam theory. The deflection and bending stress from beam theory are 0.1103 in and 9,840 psi, respectively, at the clamped end of the cantilever.

1. The first model is a plate model consisting of all `CQUAD4`s. The stress contours of this model are shown in .
2. The second model is a plate model consisting of all `CHEXAs`. The stress contours of this model are shown in [Figure 9-26](#).
3. The third model is a plate-solid model using `MPCs` to transition between the plate and solid model. The input file and stress contours of this model are shown in [Listing 9-1](#) and [Figure 9-27](#), respectively.
4. The fourth model is a plate-solid model using `RSSCONs` to transition between the plate and solid model. The input file and stress contours of this model are shown in [Listing 9-2](#) and [Figure 9-29](#), respectively.

An exploded view of the transition used in the third and fourth models are shown in [Figure 9-24](#). Both the deflections and stress contours of the four models are in good agreement with the theoretical results, as shown in [Table 9-3](#).

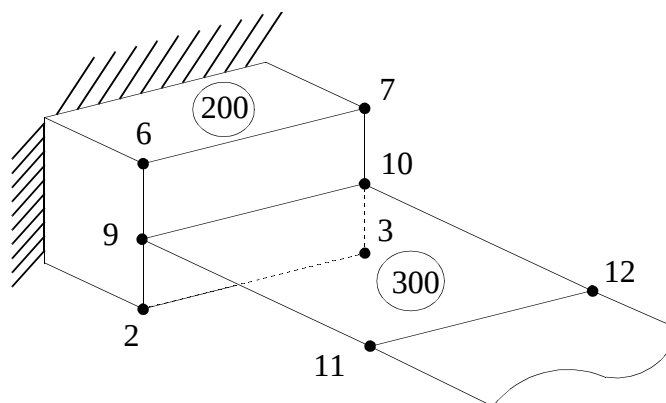


Figure 9-24 Stress Distribution for CQUAD4 Model

Table 9-3 Tip Displacements and Maximum Stresses at the Clamped End

Cantilever Clamped to Wall	Displacement at Tip	Stress at Clamped End (psi)
Beam Theory (Rigid Wall)	0.1103	9840.
All CQUAD4s	0.1103	9720.
All CHEXAs	0.1087	9720.
Plate-to-Solid Transition Using MPCs	0.1103	9720.
Plate-to-Solid Transition Using RSSCONs	0.1103	9720.

Listing 9-1 Input File for Solid-Plate Model Using MPCs for Transition

```

$
$   mpc1.dat
$
TIME    100
SOL     101
CEND
TITLE= cant cantilever clamped with HEXA
$
ECHO=BOTH
$SEALL=ALL
DISPLACEMENT =ALL
stress(corner) = all
OLOAD =ALL
spc= 100
mpc= 200
$
SUBCASE 10
LOAD = 100
BEGIN BULK
$
param,post,-1
$
grid,1,, 0., 0., 0.
grid,2,,0.1, 0.,0.
grid,3,,0.1,0.5,0.
grid,4,,0.,0.5,0.
grid,5,,0.,0.,0.1
grid,6,,0.1,0.,0.1
grid,7,,0.1,0.5,0.1
grid,8,,0.,0.5,0.1
grid,9,,0.1,0.,0.05
grid,10,,0.1,0.5,0.05

```

Listing 9-1 Input File for Solid-Plate Model Using MPCs for Transition (continued)

```

grid,11,,1.1,0.,0.05
grid,12,,1.1,0.5,0.05
grid,13,,2.1,0.,0.05
grid,14,,2.1,0.5,0.05
grid,15,,3.1,0.,0.05
grid,16,,3.1,0.5,0.05
grid,17,,4.1,0.,0.05
grid,18,,4.1,0.5,0.05
$
spc1, 100, 123, 1, 4, 5, 8
$
MPC, 200, 9,5, 1., 6, 1, -10.,,+MP01
+MP01, , 2,1, 10.
MPC, 200,10,5, 1., 7, 1, -10.,,+MP02
+MP02, , 3,1, 10.
MPC, 200, 9,4, 1., 2, 2, -10.,,+MP03
+MP03, , 6,2, 10.
MPC, 200,10,4, 1., 3, 2, -10.,,+MP04
+MP04, , 7,2, 10.
MPC, 200, 9,1, 1., 6, 1, -0.5.,,+MP05
+MP05, , 2,1, -0.5
MPC, 200, 9,2, 1., 6, 2, -0.5.,,+MP06
+MP06, , 2,2, -0.5
MPC, 200, 9,3, 1., 6, 3, -0.5.,,+MP07
+MP07, , 2,3, -0.5
MPC, 200,10,1, 1., 7, 1, -0.5.,,+MP08
+MP08, , 3,1, -0.5
MPC, 200,10,2, 1., 7, 2, -0.5.,,+MP09
+MP09, , 3,2, -0.5
MPC, 200,10,3, 1., 7, 3, -0.5.,,+MP10
+MP10, , 3,3, -0.5
$
chexa, 200, 210, 1, 2, 3, 4, 5, 6, +HX1
+HX1, 7, 8
CQUAD4, 300, 310, 9, 11, 12, 10
CQUAD4, 301, 310,11, 13, 14, 12
CQUAD4, 302, 310,13, 15, 16, 14
CQUAD4, 303, 310,15, 17, 18, 16
$
PSOLID 210100
PSHELL 310 100 .1 100 1.100
MAT1 100 1.+7 0.
$
FORCE 100 17 1. 0. 0.0 -1.
FORCE 100 18 1. 0. 0.0 -1.
$
ENDDATA

```


Listing 9-2 Input File for Solid-Plate Model Using RSSCONS for Transition

```

$
$   rsscon1.dat
$
TIME      100
SOL 101
CEND
TITLE = CANTILEVER CLAMPED WITH HEXA
$
ECHO=BOTH
  DISPLACEMENT = ALL
  stress(corner) = all
  OLOAD = ALL
SPC = 100
$
SUBCASE 10
  LOAD = 100
$
BEGIN BULK
$
param,post,-1
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.1,0.0,0.0
GRID,3,,0.1,0.5,0.0
GRID,4,,0.0,0.5,0.0
GRID,5,,0.0,0.0,0.1
GRID,6,,0.1,0.0,0.1
GRID,7,,0.1,0.5,0.1
GRID,8,,0.0,0.5,0.1
GRID,9,,0.1,0.0,0.05
GRID,10,,0.1,0.5,0.05
GRID,11,,1.1,0.0,0.05
GRID,12,,1.1,0.5,0.05
GRID,13,,2.1,0.0,0.05
GRID,14,,2.1,0.5,0.05
GRID,15,,3.1,0.0,0.05
GRID,16,,3.1,0.5,0.05
GRID,17,,4.1,0.0,0.05
GRID,18,,4.1,0.5,0.05
$
SPC1, 100, 123, 1, 4, 5, 8
$
RSSCON, 200, elem, 300, 200
$
CHEXA, 200, 210, 2, 3, 7, 6, 1, 4, +HX1
+HX1, 8, 5
CQUAD4, 300, 310, 9, 11, 12, 10
CQUAD4, 301, 310,11, 13, 14, 12
CQUAD4, 302, 310,13, 15, 16, 14
CQUAD4, 303, 310,15, 17, 18, 16
$
PSOLID  210      100
$
PSHELL  310      100      .1      100      1.      100
MAT1    100      1.+7      0.
$
FORCE   100      17      1.      0.      0.0      -1.
FORCE   100      18      1.      0.      0.0      -1.
$
ENDDATA

```

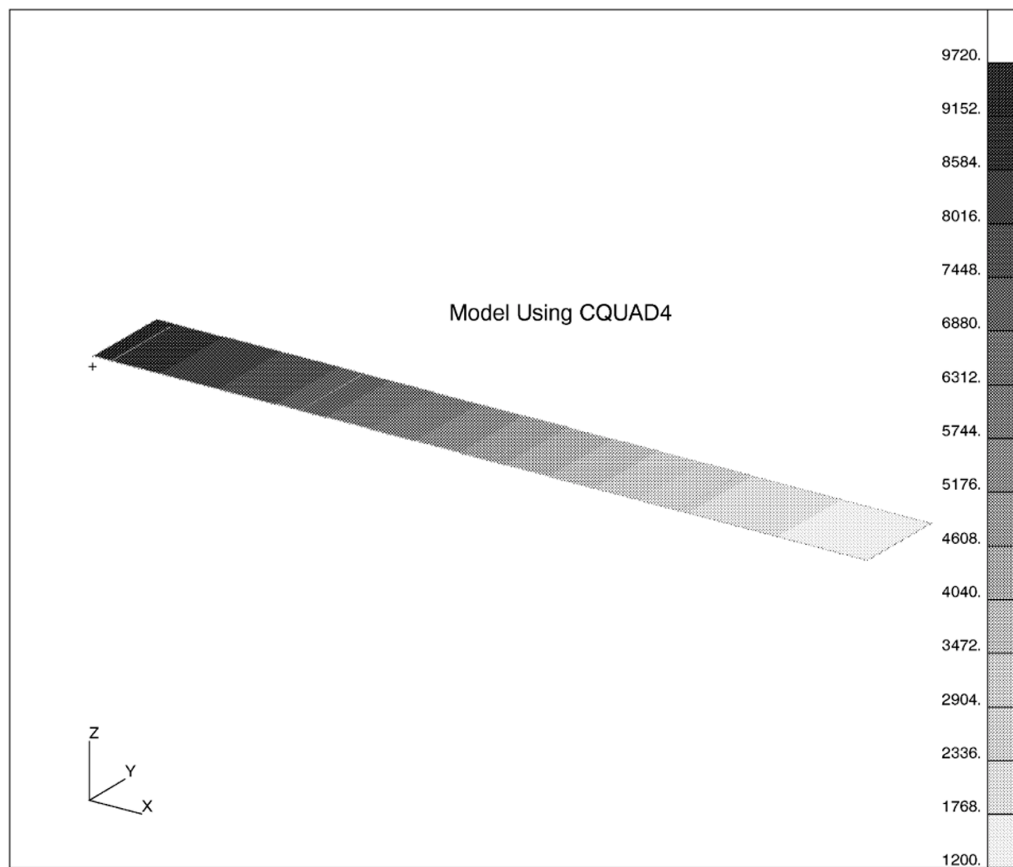


Figure 9-25 Stress Distribution for CQUAD4 Model

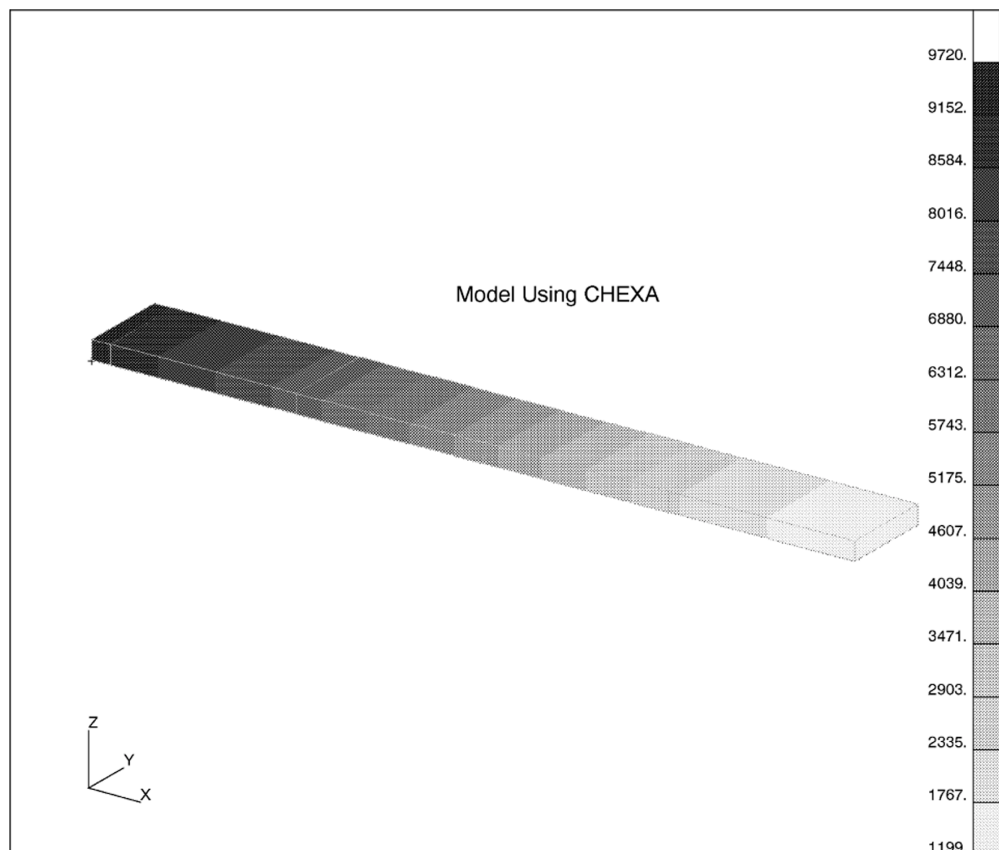


Figure 9-26 Stress Distribution for CHEXA Model

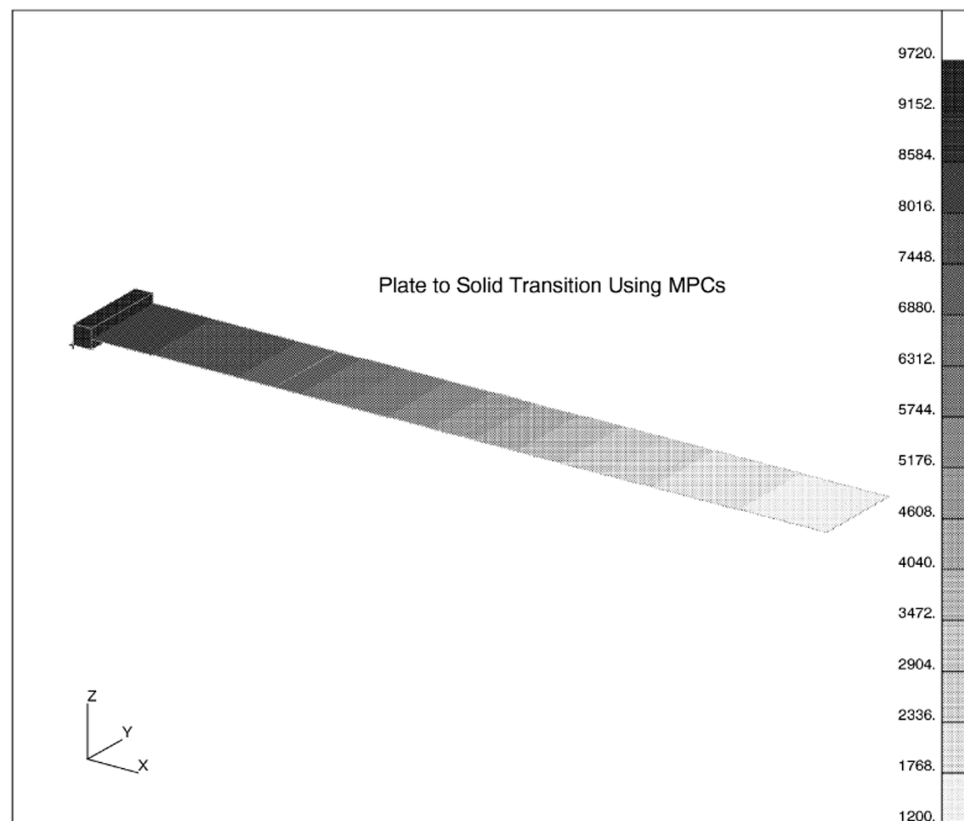


Figure 9-27 Stress Distribution for Solid-Plate Model Using MPCs for Transition

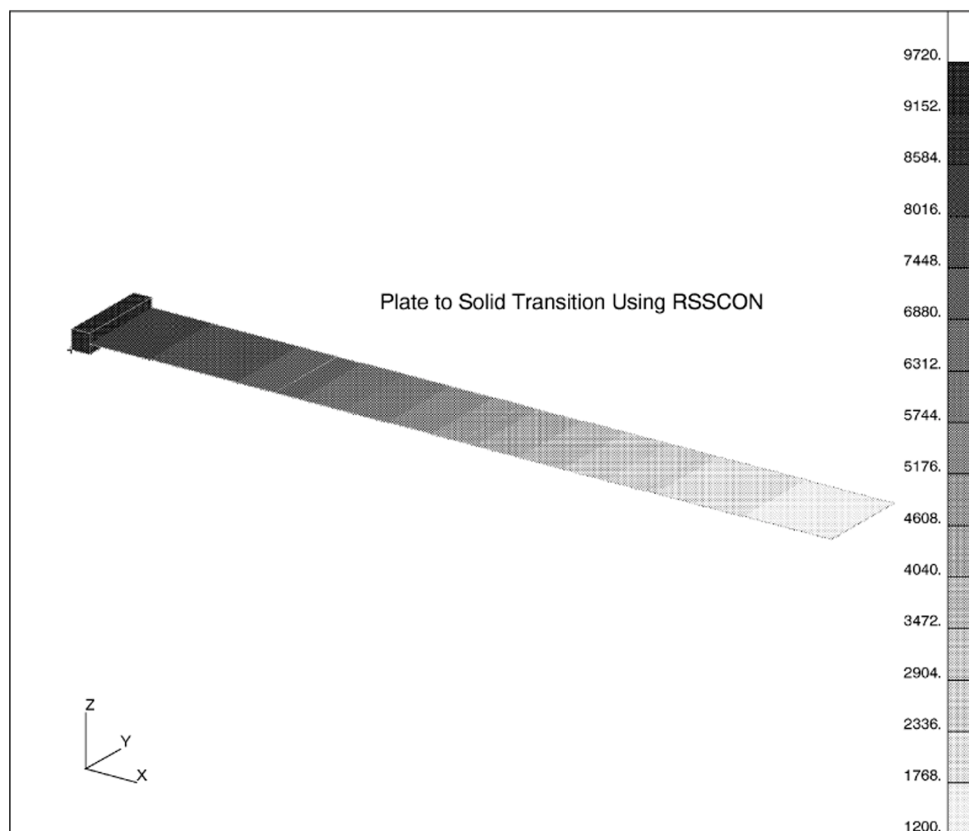


Figure 9-28 Stress Distribution for Solid-Plate Model Using RSSCONs for Transition

9.5 Grid Point Stresses

When you are working with a structure that is modeled with either plate or solid elements, it is quite common that you may want to look at either the printed output or contour plots of the element component stresses. If this is the case, extra care must be taken on your part to ensure that the component stresses, which you are evaluating, are output in some consistent manner.

Consider a simple problem as shown in **Figure 9-29**. This plate, which is 0.1 inch thick, is subjected to a uniaxial tensile load of 1 lb/in, and the boundary condition is as shown in **Figure 9-29**. This model is represented with a 3 x 3 mesh of CQUAD4 elements. If the elements are connected in a manner as shown in **Table 9-4**, then all the element σ_x values are in the direction of the applied loads. In this case, $\sigma_x = 10$ psi for all nine elements. On the other hand, if you were to change, for instance, the connectivity order for element 5 from grid points 6-7-11-10 to 7-11-10-6, the component stresses would be swapped. Note that this is the only difference between **Table 9-4** and **Table 9-5**. However, if you are not aware of it and you are making a contour plot of σ_x , the results will be quite different for the two cases.

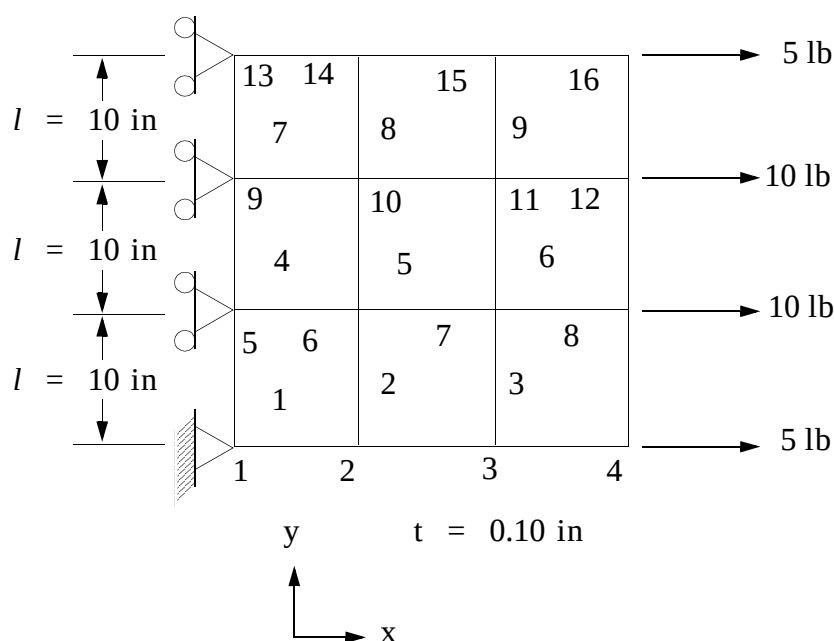


Figure 9-29 Uniaxial Load for a 3 x 3 Mesh

Table 9-4 Element Component Stresses

CQUAD4,7,1,9,10,14,13	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,8,1,10,11,15,14	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,9,1,11,12,16,15	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			
CQUAD4,4,1,5,6,10,9	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,5,1,6,7,11,10	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,6,1,7,8,12,11	$\sigma_v = 10$	$\sigma_y = 10$	$\sigma_v = 10$
\$			
CQUAD4,1,1,1,2,6,5	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,2,1,2,3,7,6	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,3,1,3,4,8,7	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			

Table 9-5 Element Component Stresses

CQUAD4,7,1,9,10,14,13	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,8,1,10,11,15,14	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,9,1,11,12,16,15	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			
CQUAD4,4,1,5,6,10,9	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,5,1,7,11,10,6	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,6,1,7,8,12,11	$\sigma_v = 10$	$\sigma_y = 10$	$\sigma_v = 10$
\$			
CQUAD4,1,1,1,2,6,5	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,2,1,2,3,7,6	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,3,1,3,4,8,7	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			

To avoid this type of surprise, it may be easier and probably more meaningful to look at the invariant stress quantities, such as Hencky-von Mises stresses, rather than the component stresses. The grid point stress (GPSTRESS) option in MSC.Nastran offers you another alternative. The grid point stress option calculates the stresses at the grid points from the adjoining plate and solid elements in a coordinate system defined by you.

This option enables you to request the output of stresses at grid points in surfaces of two-dimensional plate elements--namely, the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, and CTRIA6 elements--and in volumes containing CHEXA, CPENTA, and CTETRA solid elements.

By default, stresses are output at the center of each of these elements and also at the vertices of the CQUADR, CQUAD4 (with corner option), CQUAD8, CTRIAR, and CTRIA6 shell elements as well as the CHEXA, CPENTA, and CTETRA solid elements. This default output is not universally satisfactory because in the case concerning the surfaces of the CQUAD4 (without corner output request) and/or CTRIA3 elements, you often have a requirement for more accurate stress values at the grid points. However, the vertex stresses generated for these elements that connect to a common grid point are often not

identical. These deviations are most pronounced with a crude model and converge as the model becomes more detailed. The option described in this section provides a rational method that yields reasonable and unique stresses at each vertex grid point in surfaces or volumes defined by you.

Description of the Grid Point Stress Method

A scheme to interpolate and / or extrapolate over a surface or volume from a known set of stresses (the element stresses) requires that these known stress components be transformed into a consistent coordinate system prior to averaging these stress values. This coordinate system is defined by you, and it should be generally compatible with the surface of interest in order to obtain accurate grid point stresses. Generally, such a coordinate system is the natural coordinate system that you used in the generation of the model of the surface or volume in question or the basic coordinate system. Calculated grid point stress components are output in this coordinate system, which is called the output coordinate system of the surface or volume. This coordinate system should not be confused with the output coordinate system (CD field) as defined on the grid entries.

Each stress component (e.g., σ_x) is treated independently of other stress components in the calculation of average grid point stress components. The stress invariants at grid points (i.e., principal stresses, the Hencky-von Mises stress, and the mean pressure) are evaluated from the average values of the stress components at grid points. The grid point stress components are output in the output coordinate system of the surface.

Two methods are used to calculate grid point stress components for plate and shell elements—namely, topological and geometric, with topological as the default method. The topological method involves the averaging of stresses at a grid point due to the contribution of the elements directly attached to this grid point. This contribution does not include the effect of the element size. On the other hand, the geometric method accounts for the element size based on the numerical technique of least squares. If the results are substantially different when using the two different methods, it may be an indication that your model is not fine enough. In general, if your model is fine enough to capture the stress gradients, the two methods should yield similar results.

A single, simplified, topological method is used to calculate grid point stress components in volumes containing solid elements. However, prior to a discussion of these methods, it is necessary to define the four categories into which grid points are divided. These definitions are important for CQUAD4 (without corner output option) and CTRIA3 elements since the stress averaging method used depends on the type of grid point. However, they have very little meaning for CQUAD8, CTRIA6, and CQUAD4 (with corner output option) since the stress averaging is performed the same way regardless of the type of grid points for elements with vertex stresses.

- Interior grid points

Grid points that are connected only to interior line segments, i.e., line segments that are coincident with the edges of two or more elements. If the model in [Figure 9-29](#) is used as an example, grid points 6, 7, 10, and 11 are considered as the interior grid points.

- Corner grid points

Grid points that are only connected to a single element. For example, in the model in [Figure 9-29](#), grid points 1, 4, 13, and 16 are considered as the corner grid points.

- Edge grid points

Grid points that are connected to both interior and exterior line segments. An exterior line segment is coincident with the edge of only one element. For example, in the model in [Figure 9-29](#), grid points 2, 3, 5, 8, 9, 12, 14, and 15 are considered as the edge grid points.

- Exception grid points

Grid points at which discontinuities in stress can occur. Exception grid points are discussed later in this chapter.

Stresses at interior, corner, or edge grid points are considered to be continuous between directly connected elements. Stresses at exception points may be discontinuous between connected elements and, as will be seen, a different grid point stress may be output for each of the connected elements. Engineering judgment must be exercised in assessing the validity of grid point stresses at exception points. Note that the classification of the grid point types is performed automatically by MSC.Nastran based on their connectivities.

Topological Method - Plate Elements and Solid Elements

The topological method for the evaluation of grid point stresses only recognizes the existence of interior, edge, and corner points, i.e., exception points are not considered as a distinct category of grid points (exception grid points arise as a result of geometrical rather than topological characteristics). The relations used to compute the average grid point stresses in the surfaces of the plate elements by the topological method are described in [Table 9-7](#). The relations used to compute the average grid point stresses in the volumes containing solid elements are described in [Table 9-7](#). The topological method is the default method.

Table 9-6 Evaluation of the Average Grid Point Stresses for Plate and Shell Elements by the Topological Method

Type of Grid Point	Average Grid Point Stress Component	Remarks
Interior	$\sigma_i = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ where: σ_i = average stress at interior grid points N_e = number of directly connected elements $\sigma_{ei} = \begin{cases} \text{element center stresses if only CQUAD4 and / or CTRIA3 elements are connected;} \\ \text{element vertex stresses if CQUAD8, CTRIA6, and / or CQUAD4 (with corner output option) elements are connected} \end{cases}$	If CQUAD4 (without corner output option) and / or CTRIA3 elements connect to the same grid point as CQUAD8 and / or CTRIA6 elements, then only CQUAD8 and / or CTRIA6 elements are used in the calculation of average grid point stresses.
Edge	$\sigma_E = \frac{2}{N_e} \sum_{i=1}^{N_e} \sigma_{ei} - \frac{1}{N_i} \sum_{i=1}^{N_i} \sigma_i$ where: σ_E = average stress at edge grid points N_e = number of directly connected elements N_i = number of interior points connected to the edge points by line segment σ_i = average stress at interior grid points connected to the edge points by line segments $\sigma_{ei} = \begin{cases} \text{element center stresses if only CQUAD4 and / or CTRIA3 elements are connected;} \\ \text{element vertex stresses if CQUAD8, CTRIA6, and / or CQUAD4 (with corner output option) elements are connected} \end{cases}$	If an edge point is not connected to an interior point by line segments, then $N_i = 0$. In this case, the second term in the relation σ_E is set to zero, and the factor 2 in the first term is replaced by 1. Remarks for interior grid points also apply. The second term is not used for QUAD4 with corner request.

Table 9-6 Evaluation of the Average Grid Point Stresses for Plate and Shell Elements by the Topological Method (continued)

Type of Grid Point	Average Grid Point Stress Component	Remarks
Corner (CQUAD4 without corner output option)	Corner points connected to CQUAD4 element with the grid point diagonally opposite the corner point, an interior point, or an edge point $\sigma_c = 2\sigma_1 - \sigma_D$ where: σ_c = average stress at corner grid point σ_1 = CQUAD4 center stress σ_D = average stress at grid point diagonally opposite corner point (If the point diagonally opposite the corner point is not an interior or edge point, then $\sigma_c = \sigma_1$)	Remarks for interior grid points also apply

Table 9-6 Evaluation of the Average Grid Point Stresses for Plate and Shell Elements by the Topological Method (continued)

Type of Grid Point	Average Grid Point Stress Component	Remarks
Corner (CTRIA3)	Corner points connected to CTRIA3 elements and the other two grid points, namely, E1 and E2 of the connected CTRIA3, are edge points. $\sigma_c = 3\sigma_1 - (\sigma_{E1} + \sigma_{E2})$ where: σ_c = average stress at edge grid points σ_1 = CTRIA3 center stress σ_{E1}, σ_{E2} = stress at edge points E1 and E2 (If either E1 or E2 is not an edge point, then $\sigma_c = \sigma_1$.)	Remarks for interior grid points also apply
Grid points connected to CQUAD8, CTRIA6 and/or CQUAD4 (with corner output option)	$\sigma_c = \frac{1}{N_c} \sum_{i=1}^{N_c} \sigma_{ri}$ where: σ_c = average stress at r-th grid points N_c = number of CQUAD8, CTRIA6 and CQUAD4 (with corner output option) elements connected to the r-th grid point σ_{ri} = stress at the vertex of the i-th element connected to the r-th grid point	Remarks for interior grid points also apply.

Table 9-7 Evaluation of the Average Grid Point Stresses for Solid Elements by the Topological Method

Type of Grid Point	Average Grid Point Stress Component	Remarks
All	$\sigma_1 = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ where: σ_1 = average stress at interior or edge grid points N_e = number of directly connected elements σ_{ei} = element vertex stresses	Mixtures of CHEXA, CPENTA, and CTETRA elements may connect to an interior or edge grid point.

In the geometric method, grid point stresses for points that connect to CQUAD8, CTRIA6, and / or CQUAD4 (with corner output request) elements use the average of the element corner stresses. If CQUAD4 (without corner output request) or CTRIA3 elements connect to these points, their influence is not considered. The geometric method for the determination of average grid point stresses that connect only to CQUAD4/CTRIA3 elements utilizes the numerical technique of least squares.

Treatment of Exception Points

Exception points are defined as grid points at which stress discontinuities may occur. Such discontinuities may occur due to a substantial difference in slope between adjacent elements directly connected to the same line segment or if more than two elements are directly connected to the same line segment. **Figure 9-30** is an example of such a situation. Grid points 1, 2, 3, 4, and 5 are examples of exception grid points.

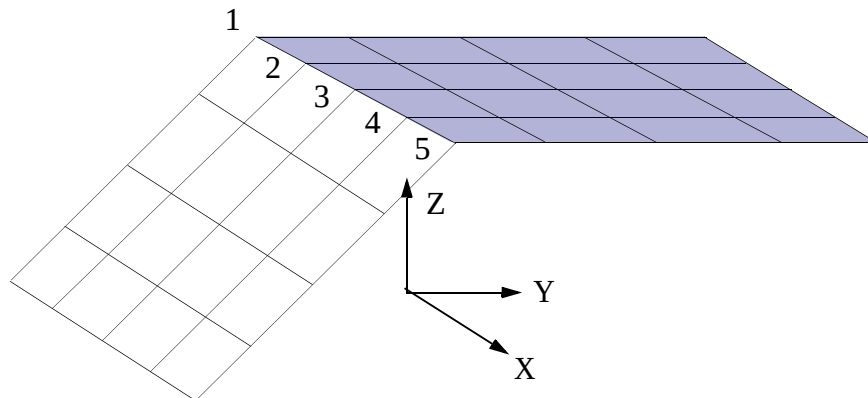


Figure 9-30 Adjacent Elements with Different Slopes Connected to the Same Line

Note that this situation is geometric rather than topological in nature and is, therefore, only considered under the geometric interpolation option.

Since exception points are defined as grid points at which stress discontinuities may exist, it is conceivable that a computed average grid point stress at an exception point may be different for each of the directly connected elements. However, it is more common to find that a subset of the elements, which are directly connected to an exception point, forms a subsurface on which stresses may be considered to be continuous. A unique average grid point stress may then be computed for the elements of the subsurface. For the problem shown in **Figure 9-18**, you may consider creating two subsurfaces—one subsurface consisting of elements within the shaded surface and another subsurface consisting of elements within the unshaded surface.

The calculation of average grid point stresses for a subsurface that contains two or more elements is performed when the geometric interpolation method is used for edge points. If a subsurface consists of a single element, the calculation of an average grid point stress is performed in the same manner as utilized in the calculation of average grid point stresses at corner points. It must be noted, however, that σ_D , σ_{E1} , and σ_{E2} may be interior, exterior, or corner points (see **Table 9-7**).

As stated previously, each element stress component (independently of all other stress components) is transformed into the output coordinate system before averaging. It is necessary that this output coordinate system vary slowly over the surface in order for the interpolated stresses to be accurate. This approach is certainly valid for stress interpolation over elements that are in a common plane, and it should be adequate for engineering analysis when the difference in slope between adjacent elements is small. If this slope is not small, it is suggested that you break them into multiple surfaces or volumes. When the elements do vary substantially from a smooth surface, you must utilize engineering judgment relative to the accuracy of the resultant stress data.

Average grid point stresses are only computed at element vertices. For elements with mid-edge grid points, you can estimate the stresses at these mid-edge grid points to be the average of the computed stresses at the grid point at the ends of the edge in question.

Grid Point Stress User Interface

The use of the grid point stress data recovery option is demonstrated in this section.

Executive Control Section. There is no special user interface required in Solution 101 for the Executive Control Section.

Case Control Section. The following Case Control commands are needed to activate the grid point stress feature:

GPSTRESS	Requests grid point stresses output.
STRFIELD	Requests grid point stresses for graphical postprocessing and stress discontinuities.
STRESS or ELSTRESS	Requests element stress output.
OUTPUT(POST)	Delimiter.

SET	Defines a set of elements that define a surface or volume.
SURFACE	Defines a surface of plate elements referenced by the SET command.
VOLUME	Defines a volume of solid elements referenced by the SET command.

In static analysis, one may place the GPSTRESS command above the subcase level or in individual subcases. If the GPSTRESS command is above the subcase level, it is utilized by all subcases and subcoms unless overridden by a GPSTRESS command in a subcase or subcom.

Note that if grid point stress data is desired, an ELSTRESS (or STRESS) request for all elements in the surfaces or volumes of interest must be included in the Case Control Section. This request is in addition to the GPSTRESS request. Such requests cause the output of element stress data as well as grid point stress data. If the printed output of element stress data is not desired, the requests “ELSTRESS(PLOT) = x” or “STRESS(PLOT) = x” may be used. For example, as shown in the problem in [Figure 9-29](#), if grid point stresses are requested for grid point 11, then element stresses must be requested for a set that includes at least elements 5, 6, 8, and 9. Furthermore, the set of elements in the SURFACE definition must also include elements 5, 6, 8, and 9. In this example, if one of the elements that is connected to grid point 11 (e.g., element 6) is left out in the SET definition, then User Warning Message 4621 is issued. The GPSTRESS values calculated in this case may be meaningless.

The OUTPUT(POST) section of the Case Control Section contains specific requests for grid point stress data for the surfaces or volumes of elements. Any number of element surfaces or volumes may be defined, but only those surfaces or volumes referenced through the GPSTRESS command will have grid point stress data generated and output.

Bulk Data Section. If postprocessing of grid point stress is desired, the parameter PARAM,POST,x is needed in the Bulk Data Section. There is no other special requirement.

Again, the model in [Figure 9-29](#) is used for this illustration. In this case, you may want, for example, to have two separate surfaces. The first surface contains elements 1, 2, 4, 5, 7, and 8; the second surface contains all elements. The following input file is used to accomplish these goals.

```
$
$ FILENAME - gpstress.dat
$
ID GPSTRESS ONE
SOL 101 $
TIME 5
CEND
TITLE = GPSTRESS EXAMPLE
.
.
.
STRESS= ALL
SET 10 = 100,200
GPSTRESS = 10
```

```

STRFIELD = 10
$
OUTPUT(POST)
  SET 1 = 1,2,4,5,7,8
  SET 2 = ALL
  SURFACE 100 SET 1 NORMAL Z
  SURFACE 200 SET 2 NORMAL Z
$
BEGIN BULK
$
$   BRING IN THE REST OF THE BULK DATA INPUT
$
include 'bulk.dat'
$
ENDDATA

```

In the above example, surface 100 contains elements 1, 2, 4, 5, 7, and 8 that were referenced by set 1. Surface 200 contains all of the elements that were referenced by set 2. The element positive normal is in the direction of the z-axis of the basic coordinate system. By default, the grid point stresses are transformed to the basic coordinate system. The grid point stresses are called out by the GPSTRESS command, which points to set 10, which, in turn, points to surfaces 100 and 200. Element stresses are requested for all of the elements, which, of course, encompass elements 1, 2, 4, 5, 7, and 8. The STRFIELD command is required to generate postprocessing data for your graphic package. A partial output of a typical grid point stress is provided below.

STRESSES AT GRID POINTS - - SURFACE 100										
SURFACE X-AXIS X NORMAL(Z-AXIS) Z REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0										
GRID	ELEMENT	STRESSES IN SURFACE SYSTEM				PRINCIPAL STRESSES			MAX	
ID	ID	FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	SHEAR	VON MISES
1	0	Z1	1.000E+01	-2.009E-14	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		Z2	1.000E+01	-2.009E-14	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		MID	1.000E+01	-2.009E-14	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
2	0	Z1	1.000E+01	-8.986E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		Z2	1.000E+01	-8.986E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		MID	1.000E+01	-8.986E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
.										
15	0	Z1	1.000E+01	-1.631E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		Z2	1.000E+01	-1.631E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01
		MID	1.000E+01	-1.631E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01

9.6 Consistent Loading

In finite element analysis, the loads are often applied to the elements rather than to the grid points. Examples of such loadings include the edge load on a CQUAD4 element or a pressure load on one of the faces of a solid element.

When you convert these element loads to grid point loads, a common tendency is to spread them equally at the connected grid points. Let us look at an example of a typical plate element with uniform in-plane loading at one of the edges as shown in [Figure 9-31](#). The total load (P) is to be uniformly distributed along the edge of the element with an intensity of P/l , where l is the length of the edge. This same load is to be applied to six different elements—CQUAD4, CTRIA3, CQUAD8, CTRIA6, CQUADR, and CTRIAR. If you lump these loads equally to the grid points, the load at each node is $P/2$ for the CQUAD4, CTRIA3, CQUADR and CTRIAR, and $P/3$ for the CQUAD8 and CTRIA6. Since the geometry and boundary condition are identical for the six models, and the loads are “seemingly equivalent,” you will expect the results (displacements, stresses, etc.) to be identical. However, in general, this is not the case. The results for the cases using CQUAD8, CTRIA6, CQUADR, and CTRIAR are different from the results using CQUAD4 and CTRIA3. The reason for this is because these “seemingly equivalent” loads are actually not quite equivalent. They are considered as lumped loads. In order to obtain the correct load distribution, these loads need to be converted to consistent loads.

The equivalent loads at the grid points computed from the element loads are known as consistent loads, and they are calculated by applying the principle of virtual work. The same shape function that is used in deriving the element stiffness is used for arriving at this load—and hence the word consistent load. They are a function of the element types and the applied loads. Depending on the element, these lumped loads in general are not equal to the consistent loads. The consistent loads for each of these elements for a uniform in-plane edge load of P/l are shown next to each element. In this case, the lumped loads and consistent loads are the same for the CQUAD4 and CTRIA3. They are not the same for the CQUAD8, CTRIA6, CQUADR and CTRIAR. As you can see, they are substantially different from the lumped loads that were discussed in the previous paragraph. It is interesting to note that the loads are distributed as $1/6$, $4/6$, and $1/6$ along the edge for the CQUAD8 and CTRIA6, which is quite different from the lumped load approach. For the CQUADR and CTRIAR, an additional moment $Pl/12$ is needed to arrive at the consistent loads. When the corresponding consistent loads shown in [Figure 9-31](#) are applied to each of the respective elements, the force distributions are then equivalent for all the elements.

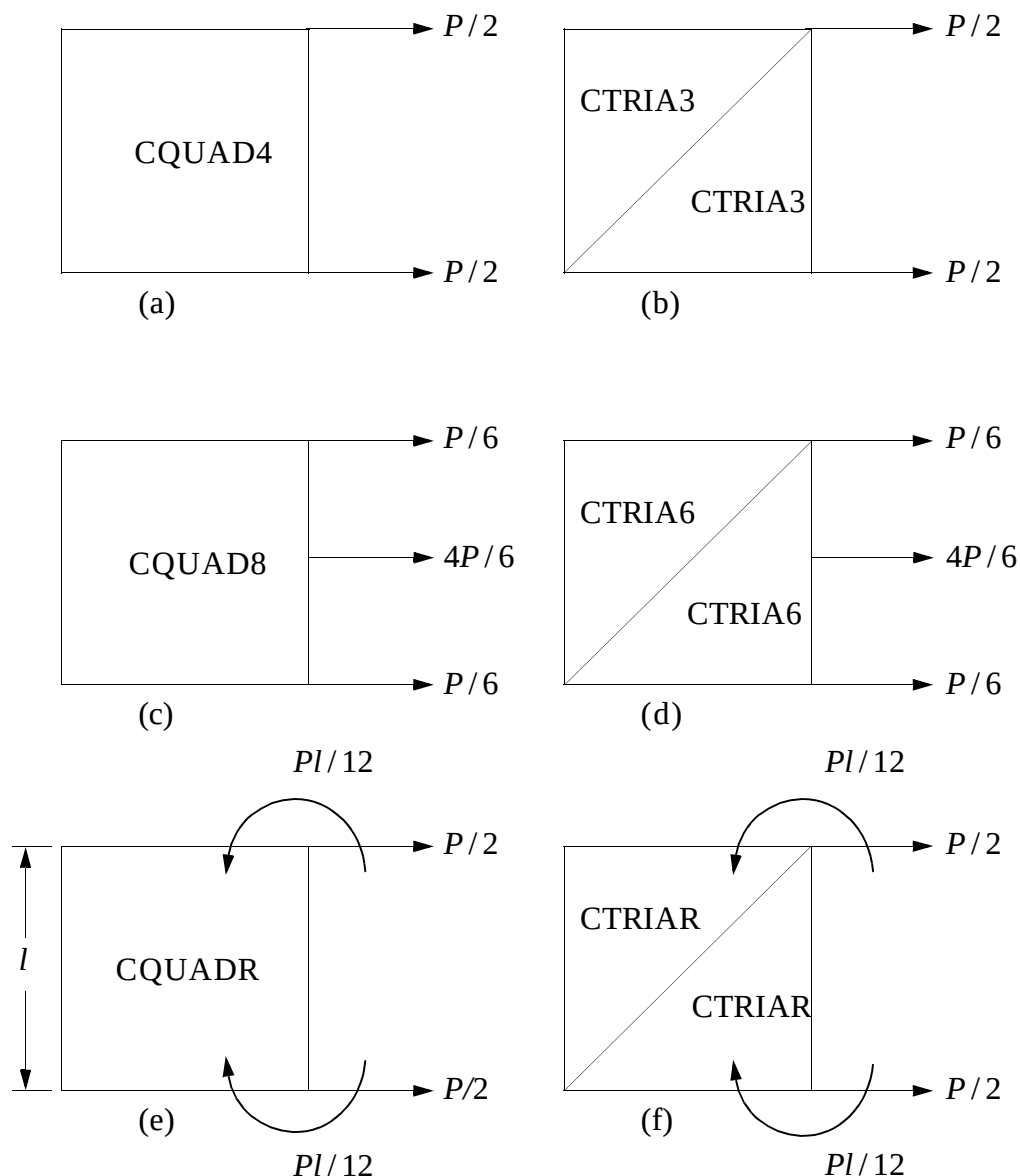


Figure 9-31 Consistent Loads Due to Uniform In-Plane Element Edge Load

Consider another example with solid elements. A load P is to be applied evenly as an outward pressure load (P/A) to a surface of each of the four solid elements shown in [Figure 9-32](#). Again, if you lump these loads equally to the grid points, the lumped load at each node is $P/4$ for the CHEXA (with eight nodes) and CPENTA (with six nodes), and $P/8$ for the CHEXA (with 20 nodes) and CPENTA (with 15 nodes). The consistent loads are shown in [Figure 9-32](#). The consistent and lumped loads are the same for the CHEXA (with eight nodes) and the CPENTA (with six nodes) for this case. The consistent loads for the CHEXA (with 20 nodes) and CPENTA (with 15 nodes) are shown in [Figure 9-32\(c\)](#) and [Figure 9-32\(d\)](#), respectively. The load distribution along each edge is $-.0833P$, $.3333P$ and $-.08333P$, or a $-1/4/-1$ ratio, which is quite counter intuitive, especially for the sign change. Fortunately, you do not have to calculate this consistent load for the pressure load. This consistent load is generated automatically inside MSC.Nastran by using the PLOAD4 entry.

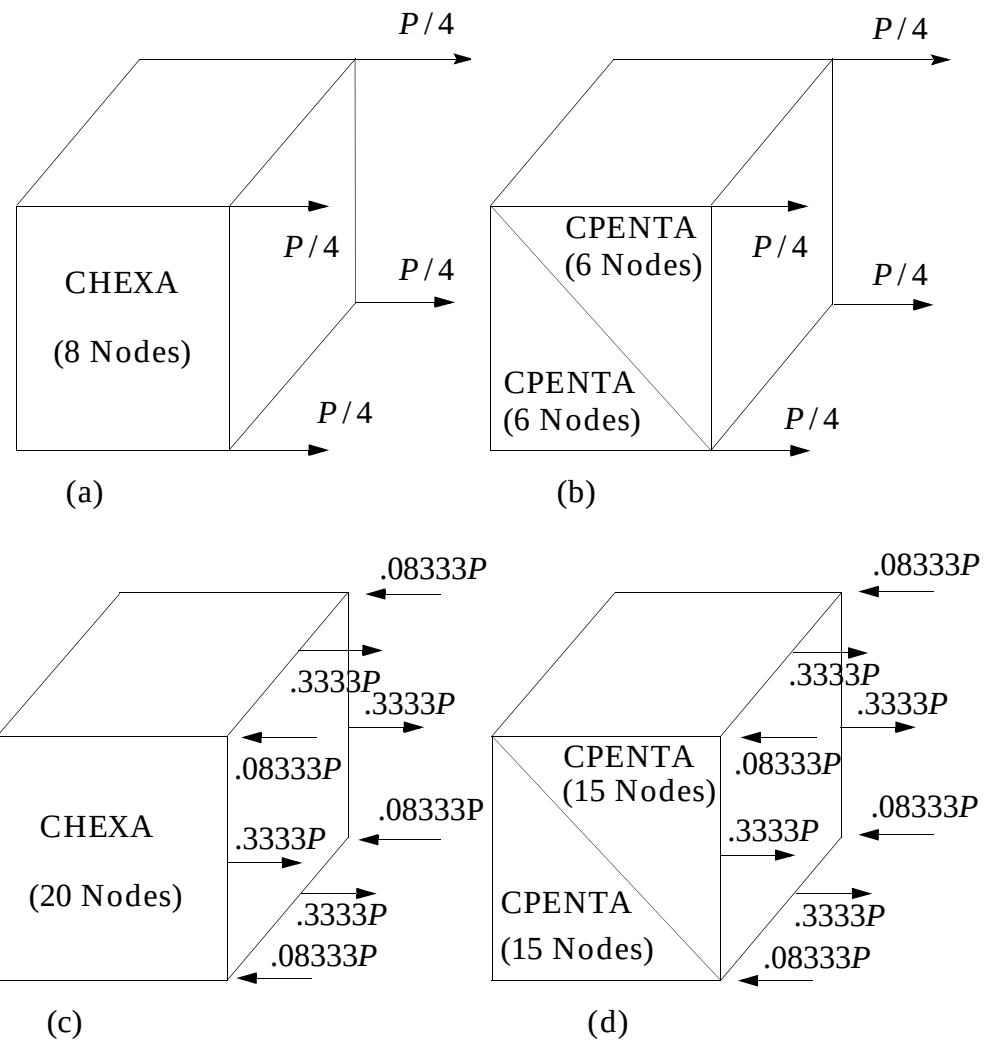


Figure 9-32 Consistent Loads Due to Uniform Loads on a Solid Element Face

As you can see, consistent loads are functions of the element types and applied loads.

9.7 Symmetry

When you think of a symmetrical structure, you most likely think of a structure that has one or more planes of reflective symmetry. Although there are other kinds of symmetry available in MSC.Nastran, reflective symmetry is the only type that is discussed in this section. MSC.Nastran also provides a series of special solution sequences that automates some of these other types of analyses. They are known as the cyclic symmetry features and are covered in “[Cyclic Symmetry](#)” on page 559. If a structure is symmetric, then the size of your finite element model can be reduced, which, in turn, reduces the time and cost of your analysis. For each plane of symmetry that you have in your model, the model size can be reduced by a factor of approximately two. [Figure 9-33\(a\)](#) and [Figure 9-33\(b\)](#) illustrate structures that contain one and two planes of symmetry, respectively. In the first case, only half the model needs to be represented. In the second case, only a quarter of the model needs to be represented.

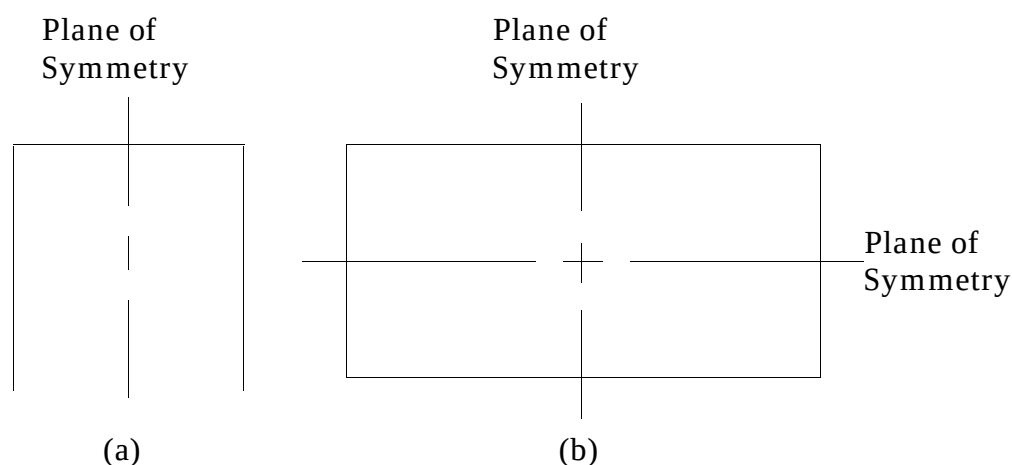


Figure 9-33 Structures with Planes of Symmetry

If the loads applied to the structure are symmetric relative to the plane of symmetry, then the full model can be replaced with half the model by applying a symmetric boundary condition. On the other hand, if the loads are antisymmetric, the same simplification can be achieved by applying the antisymmetric boundary condition.

A symmetric boundary condition implies that the displacements normal to the plane of symmetry and rotations about the axes in the plane of symmetry are zero at the plane of symmetry. An antisymmetric boundary condition implies that the displacements in the plane of symmetry and rotations normal to the plane of symmetry are zero at the plane of symmetry.

Any general loading condition can be broken into a combination of symmetric and antisymmetric loads relative to the plane of symmetry. You can work through an example of applying symmetry using the model in [Figure 9-34\(a\)](#). This load can be broken into two separate loads as shown in [Figure 9-34\(b\)](#) and [Figure 9-34\(c\)](#).

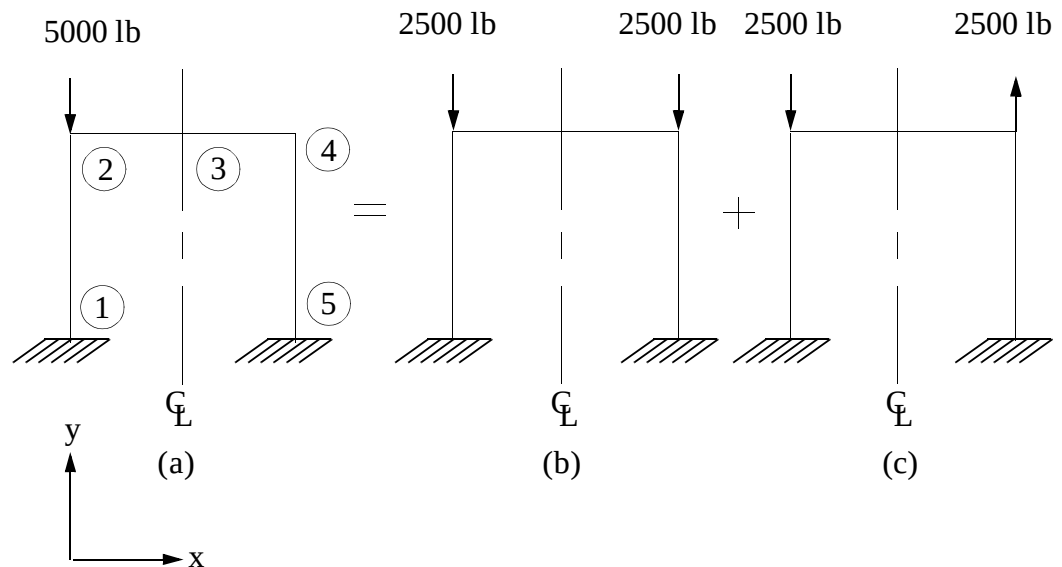


Figure 9-34 Decomposing a General Loading into Symmetric and Antisymmetric Loadings

Note that **Figure 9-34(a)** is a case of a general loading applied to a structure with one plane of symmetry. **Figure 9-34(b)** and **Figure 9-34(c)** represent a symmetric loading and an antisymmetric loading, respectively, applied to the same structure. By using symmetry, only one-half of the structure, as shown in **Figure 9-34(b)**, needs to be modeled by applying the symmetric boundary condition at the plane of symmetry (see **Figure 9-35**).

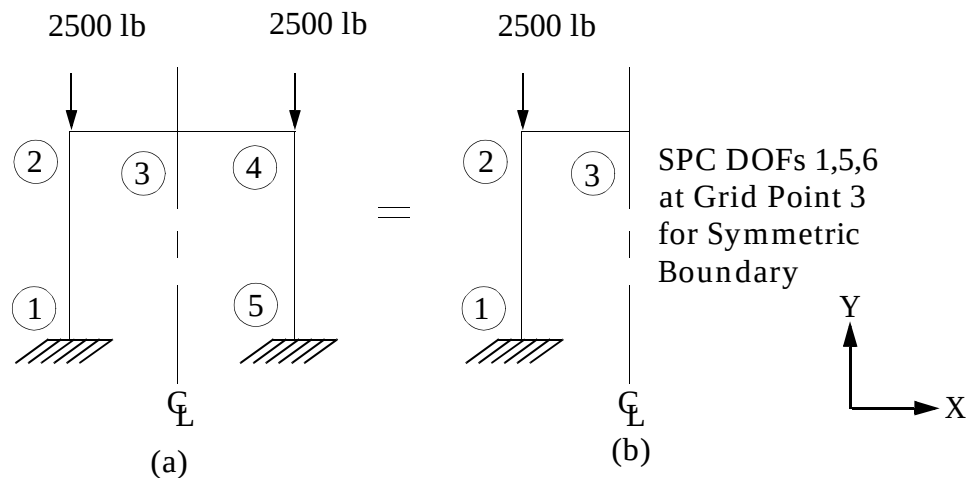


Figure 9-35 Symmetric Model

Similarly, only one-half of the structure shown in **Figure 9-34(c)** needs to be modeled by applying the antisymmetric boundary condition as shown in **Figure 9-36**.

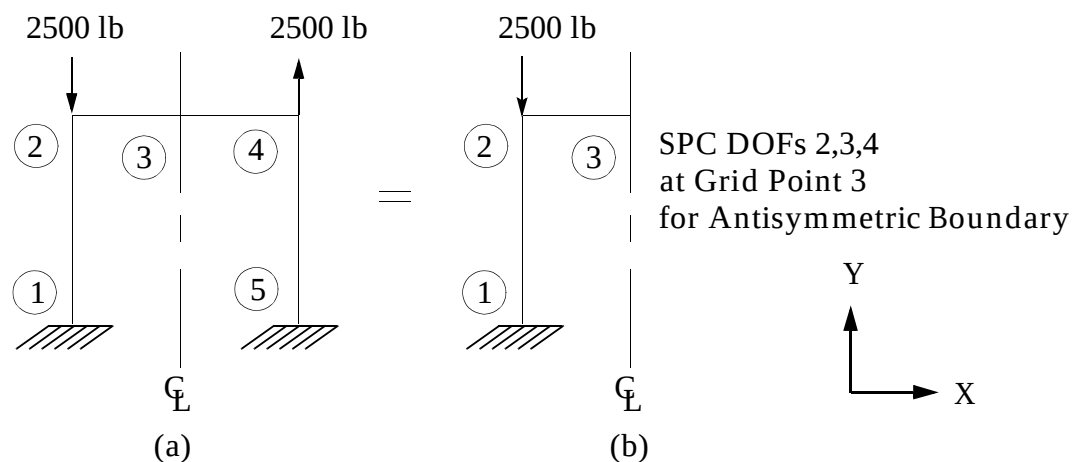


Figure 9-36 Antisymmetric Model

User Interface

No special user interaction is required in the Executive Control Section or Bulk Data Section. Depending on the number of planes of symmetry, approximately one-half or less of the structure needs to be modeled. For the above frame model, only one-half of the model needs to be analyzed.

This feature is basically activated by Case Control commands. The above frame example is used to illustrate the required Case Control commands. [Listing 9-3](#) contains the input file for this problem.

Listing 9-3 Input File for Using Symmetric and Antisymmetric Boundary Conditions

```

$
$ FILENAME - symbar.dat
$
ID SYM ANTI
TIME 5
SOL 101
CEND
$
TITLE = SYMMETRIC AND ANTISYMMETRIC
SUBCASE 1
  LABEL = SYMMETRIC CONSTRAINTS - Y LOAD
  SPC = 1
  LOAD = 2
$
SUBCASE 2
  LABEL = ANTISYMMETRIC CONSTRAINTS - Y LOAD
  SPC = 2
  LOAD = 2
$
SUBCOM 3
  LABEL = LEFT SIDE OF MODEL - Y LOAD
  SUBSEQ 1.0, 1.0
  DISP=ALL
$
SUBCOM 4
  LABEL = RIGHT SIDE OF MODEL - Y LOAD
  SUBSEQ 1.0, -1.0
  DISP=ALL
$
BEGIN BULK
CBAR    1      100      1      2      -1.0      0.0      0.0
CBAR    2      100      2      3      0.0      1.0      0.0
FORCE   2      2          2500.  0.0      -1.      0.0
GRID    1          0.0      0.0      0.0          123456
GRID    2          0.0      10.0     0.0
GRID    3          5.0      10.0     0.0
MAT1    1      3.+7      0.3
PBAR    100     1      5.0      5.0      5.0      10.
SPC1    1      156      3
SPC1    2      234      3
ENDDATA

```

The first subcase calls out the symmetric boundary condition and load as shown in [Figure 9-35\(b\)](#). The second subcase calls out the antisymmetric boundary condition and load as shown in [Figure 9-36\(b\)](#).

The third subcase is a “results combination” subcase. No boundary or load conditions need to be applied for this loading condition (see “[Modeling Guidelines](#)” on page 311). This subcase produces results for the portion of the structure that you have modeled (the left-hand side of the frame in this case) by performing a linear combination of the first two subcases. This combination is achieved by adding 100% of the results for the first subcase to 100% of the second subcase using the SUBSEQ command. Note that in this case of results combination, the SUBCOM instead of the SUBCASE command is used.

The fourth subcase is also a “results combination” subcase. It produces results for the portion of the structure that you did not model (the right-hand side of the frame in this case). These results are achieved by subtracting 100% of the results of the second subcase from 100% of the results from the first subcase. As in Subcase 3, the SUBCOM command instead of the SUBCASE command is used in this case. Once again, no load or boundary condition is needed for this subcase; this is an optional subcase. It is not required if you do not want to obtain results for the other half of the structure that you did not model.

The displacements for SUBCOM 3 and SUBCOM 4 are shown in [Figure 9-37](#) along with the results obtained from a full model run. The results for SUBCOM 3 (left-hand side of the frame) correlate with the full model results. The results for SUBCOM 4 (right-hand side of the frame) correlate with the full model results except for the sign changes for components x , θ_y , and θ_z . The sign changes occur because the results of the right half (reflected half) are produced in terms of its left half using the left-hand coordinate system. Note that these are the same degrees of freedom that are constrained for the symmetric boundary condition.

LEFT SIDE OF MODEL - Y LOAD								SUBCOM 3
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0		.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0		.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0		.0	.0	3.453039E-05
RIGHT SIDE OF MODEL - Y LOAD								SUBCOM 4
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0		.0	.0	.0
2	G	1.381215E-04	-5.524853E-06	.0		.0	.0	-2.762431E-05
3	G	1.381215E-04	-1.666667E-04	.0		.0	.0	-3.453039E-05

FULL MODEL - NEGATIVE Y LOAD								SUBCASE 1
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0		.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0		.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0		.0	.0	3.453039E-05
4	G	-1.381215E-04	-5.524862E-06	.0		.0	.0	2.762431E-05
5	G	.0	.0	.0		.0	.0	.0

Figure 9-37 Results Comparison Using Symmetry Versus a Full Model

In the past, when computers were not as fast as they are now, the use of symmetry to reduce the finite element model size was more popular than it is today. The advantage of using symmetry is obviously the reduction in model size. The disadvantage is that it requires more effort on your part and it is more prone to errors since you have to provide

equivalent symmetric / antisymmetric boundaries and loading conditions to your model. In MSC.Nastran, the use of symmetry is built into some of the solution sequences-they are known as the cyclic symmetry solution sequences (“[Cyclic Symmetry](#)” on page 559). Similar approaches can be used with superelement analysis.

The SYM / SYMCOM Case Control command combination can be used instead of the SUBCASE / SUBCOM combination. If you use this combination, the only changes you have to make is to replace the SUBCASE and SUBCOM Case Control commands with the SYM and SYMCOM Case Control commands, respectively. When the SUBCASE / SUBCOM combination is used, output is available in both the SUBCASE and SUBCOM subcases. On the other hand, when the SYM / SYMCOM combination is used, output requests are only available in the SYMCOM subcases. There is no real advantage to using the SYM / SYMCOM command as compared to the SUBCASE / SUBCOM command.

CHAPTER 10

Model Verification

- Introduction to Model Verification
- Preprocessor Checks
- Strain Energy Output
- Diagnostic Tools
- Stress Error Estimators
- Postprocessor Checks

10.1 Introduction to Model Verification

In “[Modeling Guidelines](#)” on page 311, details on various aspects of modeling were presented. This chapter is dedicated to various tools that are available for verifying and improving the quality of your models. They are broken into five different categories:

- Preprocessor checks.
- Strain energy output.
- Diagnostic tools.
- Stress error estimators.
- Postprocessor checks.

Some of these categories overlap each other. Most of these checks provide you with a tremendous amount of information, but yet their usages are quite straightforward and require very little effort on your part.

10.2 Preprocessor Checks

The use of computer graphics in structural analysis is no longer a luxury--it is an integral part of the standard tools for all finite element users. This section is intended to point out some of the handy tools that are available in most commercial preprocessors.

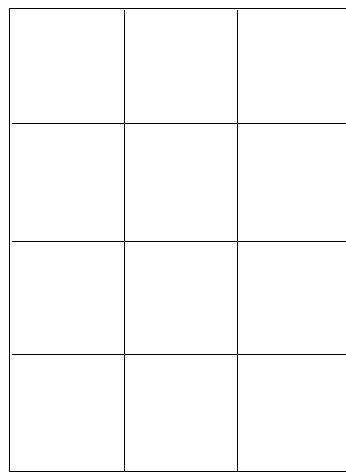
Preprocessors are used extensively for the generation of finite element models. They also contain features that can help you to improve the quality of your model and detect errors in your model prior to analysis. These features can result in substantial savings in terms of time and money.

Some of the illustrations in this section may not be as dramatic as you would otherwise observe using the actual graphic package. A black and white picture in this user's guide would certainly not appear as impressive or useful as the actual color graphics that you can experience interactively in front of a graphics terminal. However, the illustration serves as a guide for features that you should keep in mind when using these packages. Depending on the graphics package that you use, some of these features may or may not be available to you.

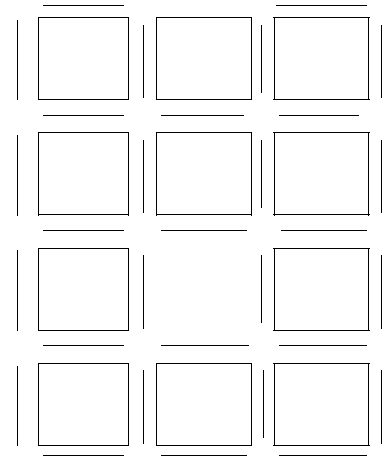
Shrink Plot

The shrink option allows you to shrink your elements by a specified percentage. This feature is an excellent tool for identifying missing elements. As an example, let us look at a plate model with 12 CQUAD4 elements surrounded by 31 CBAR elements as shown in [Figure 10-1](#). The normal unshrunk plot looks something like [Figure 10-1\(a\)](#). The CBAR elements are not visible since they lie on top of the edges of the CQUAD4 elements.

Now replot this figure using the shrink option. By using the shrink option as shown in [Figure 10-1\(b\)](#), it becomes quite obvious that you are missing a CBAR element at the top and a CQUAD4 element close to the center. Had you not used the shrink option, these missing elements would probably have gone undetected. You may argue that had the element labels been turned on, the missing elements would probably have been noticed. This is certainly a valid argument in this case since the model is a simple one. However, for complex models, the picture would have become too crowded to be useful had all the labels been turned on. This same shrink feature can also be applied to solid elements in a similar manner.



(a)



(b)

Figure 10-1 Shrink Plot

Hidden Line/Shaded Plots

For a complicated model, looking at a wireframe model by itself can be quite confusing since you lose the depth of field. However, hidden line and shaded plots can serve as the perfect complement to the wireframe plots. The box model in [Figure 10-2](#) is a perfect example. This model is made of CHEXA elements and has a hole in the front. By looking at the wireframe plot on the top left-hand corner of [Figure 10-2](#), you may not notice that there is a hole in the front. However, the hidden line (top right), shaded (bottom left), and shaded with shrink option (bottom right) plots all reveal the hole in front of the box. Furthermore, when using these features, you can see the structure being plotted from back to front, which gives you a better perspective of what the structure looks like. As the structure becomes more complicated, you will appreciate these features even more.

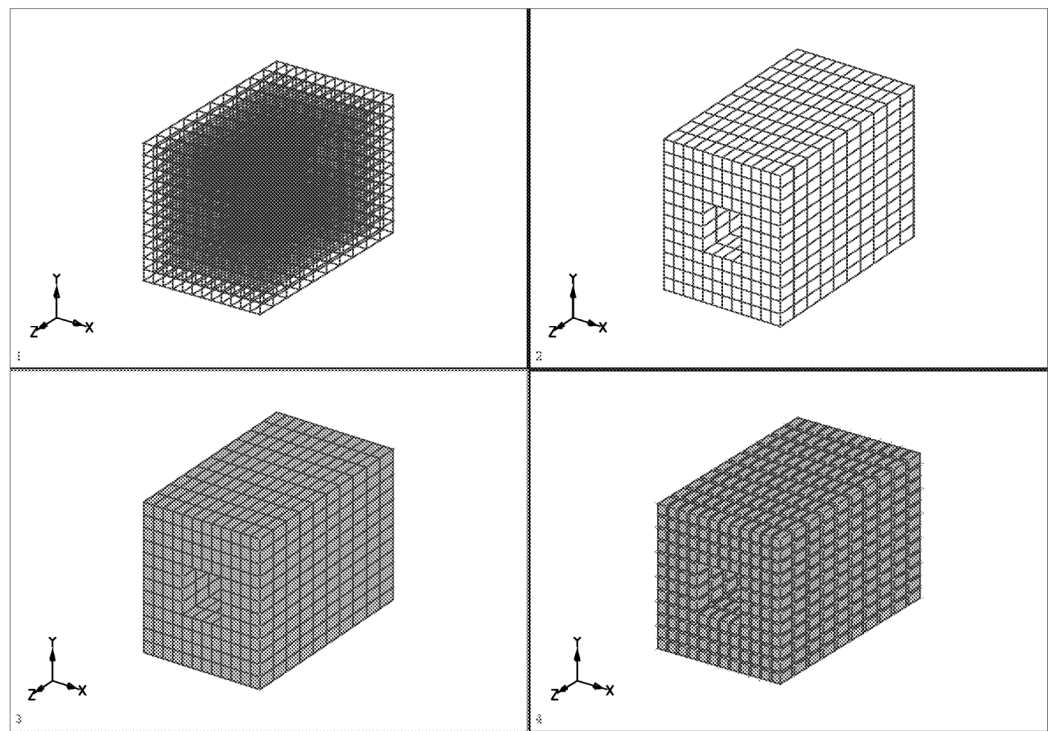


Figure 10-2 Wireframe, Hiddenline, and Shaded Plots

Free Edge/Face

A free edge is an edge that is connected by one single two-dimensional element (e.g., CQUAD4). The existence of free edges does not necessarily indicate a modeling problem. **Figure 10-3** represents a crude wing model. The bold lines around the outside are examples of legitimate free edges. On the other hand, whether the internal vertical bold line is a legitimate free edge or not depends on your design intent. This free edge indicates that elements 12 and 14, 11, and 13 are not connected to each other. If this is not your intention, then this free edge indicates a potential modeling error.

This can happen, for example, if you create this wing with four separate surfaces as shown in **Figure 10-4**. Most preprocessors typically create geometric surfaces and then generate the finite element mesh for each of these surfaces. Since elements 12 and 14 are created from two separate geometric surfaces, their connecting grid points have distinct IDs. After these elements are generated, if you want them to share the same edge, then you must perform some sort of equivalencing operation. The exact operation depends on the graphics package itself. This free edge indicates that you have either forgotten to perform this equivalencing operation or the equivalencing tolerance level is not tight enough for the program to perform this task.

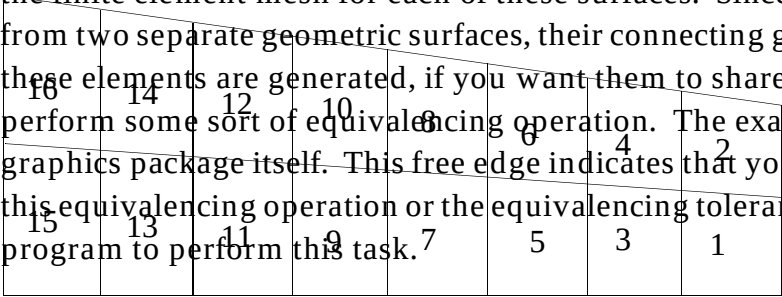


Figure 10-3 Wing Free Edge Check

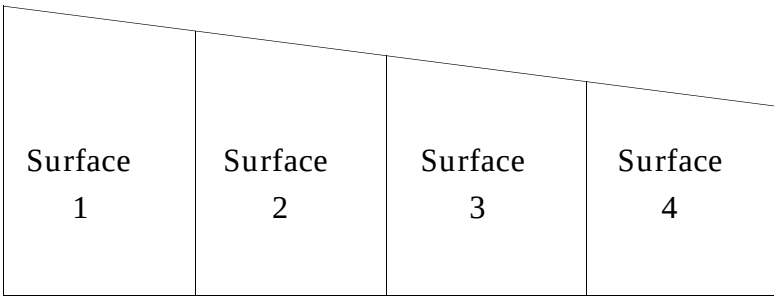


Figure 10-4 Wing Geometric Surfaces

The free face concept is similar to the free edge concept except that it applies to three-dimensional elements instead of two-dimensional elements. A free face is a face that is occupied by one single three-dimensional element (e.g., CHEXA). The example in **Figure 10-5** contains two volumes. After these two volumes are meshed with solid elements (e.g., CHEXAs), a free face check is performed. The outside surfaces of these two volumes indicate that they are free faces that are legitimate. In addition, it indicates that the adjoining surface (hashed area) is also a free surface. This indicates that the two volumes are not connected, which may or may not be your intention. If it is not your intention, there is probably a modeling error.

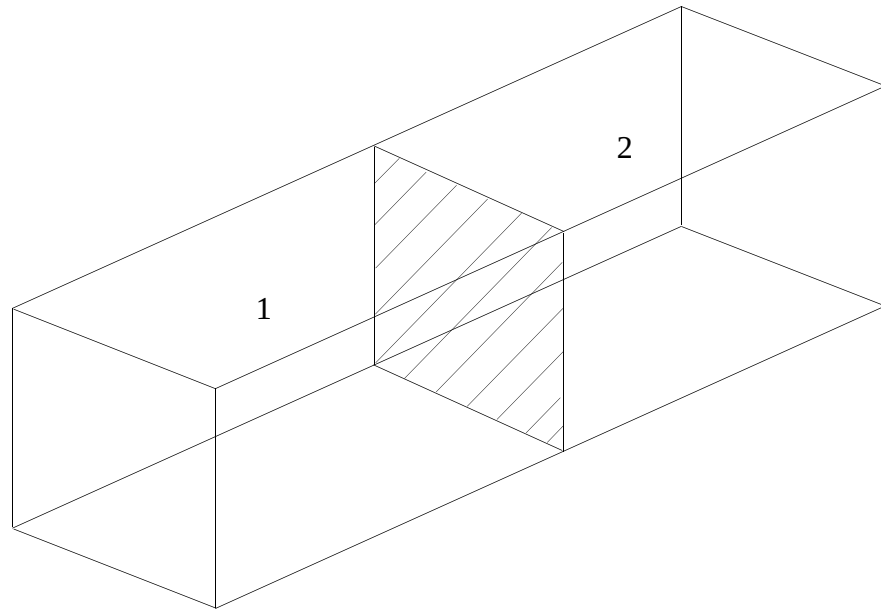


Figure 10-5 Possible Modeling Error as Indicated by Free Face

Zipper Effect

When adjoining surfaces are connected at very few points, the interface has a tendency to open up when it is loaded. This looks similar to a zipper and hence is classified as the “zipper effect.” Use the surfaces in [Figure 10-4](#) and create new meshes. Surfaces 1, 2, 3, and 4 contain meshes of 4×2 , 4×2 , 3×2 , and 3×2 , respectively. As you can see in [Figure 10-6](#) the interface between surfaces 2 and 3 is only connected at two grid points--grid points A and B. The loads can only be transferred between these two portions of the structure through these two locations. In other words, the load path may be quite different from what the actual structure does.

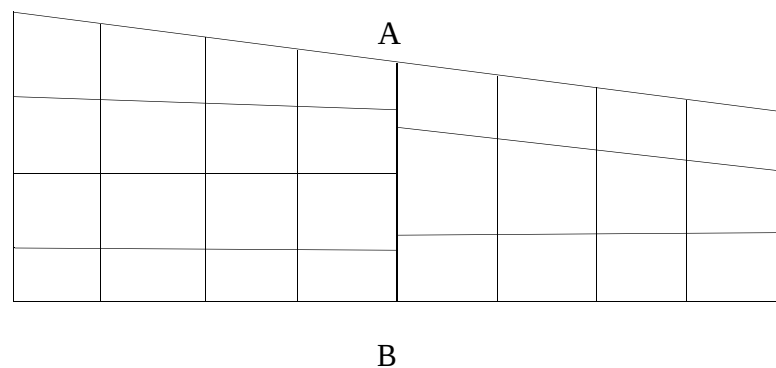


Figure 10-6 Model with Zipper Effect

The “zipper effect” can occur in both plate and solid element models. Therefore, when generating meshes for a structure with multiple surfaces and /or volumes, you should keep this in mind so that the interfaces are connected properly. You should always perform a quick check by zooming in at all the interface locations.

CBAR/CBEAM Orientation and Offset Check

One common error that many users make is the incorrect orientation of the CBAR/CBEAM inertia properties. Definitions of the CBAR/CBEAM orientation vectors are provided in “[MSC.Nastran Elements](#)” on page 65 and [MSC.Nastran Common Questions and Answers](#). If the graphics package that you are using plots these orientation vectors, you should take advantage of this feature to ensure that you have oriented these elements properly. You should also check the CBAR/CBEAM element offset plots, if they are available.

Duplicate Grid Points/Elements

The “duplicate grid point check” feature highlights grid points that occupy the same location. Having grid points occupy the same space may or may not be your intention. An example of unintentional grid points occupying the same location can be a result of not performing an equivalencing operation after meshing more than one curve, surface, or volume. An application of intentional duplicate grid points can be a model used to simulate a bolted joint. This effect can be achieved by connecting a stiff spring between these two points that occupy the same location but are connected to different portions of the structure.

The “duplicate element check” feature is similar to the “duplicate grid check” except the check is done on the elements rather than the grid points. These errors are often due to unintentionally meshing the same line, surface, or volume more than once.

Properties/Material Plots

You can also assign different colors to elements based on their property IDs (e.g., PSHELL IDs). If you have assigned an incorrect property, graphically this becomes quite obvious. Differentiating by color can also be used to highlight material properties (e.g., MATi IDs).

Consistent Plate Normals

When you create a model, you should always generate the elements in a consistent manner. In some cases, it makes your bookkeeping job a bit easier. However, in other cases, if you do not generate your model in a consistent manner, you can then inadvertently apply the loads in the wrong directions. An example of such a case is when the applied load is a pressure load. The plate model, consisting of four CQUAD4s as shown in [Figure 10-7](#), is used to illustrate this point. As mentioned in Chapter 4, each CQUAD4 has a positive normal direction associated with it. This direction is defined by the way you connect the element using the right-hand rule. The symbols used to denote the directions of the normals are shown in [Figure 10-8](#).

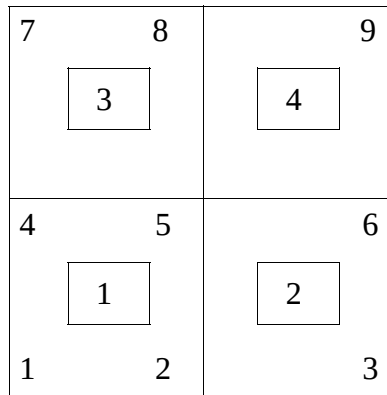


Figure 10-7 CQUAD4 Model

Now apply a 100 psi pressure load to these four elements. The direction of the pressure load is pointing into the paper. **Listing 10-1** contains the relevant partial input file for this job. In this case, the grid points for each element are defined in a consistent manner. They are connected in a clockwise direction. Therefore, the positive normals are all pointing into the paper as indicated by **Figure 10-8(a)**.

Listing 10-1 Consistent CQUAD4 Connectivity

```
ID RUN1 PLOAD4
SOL 101
CEND
TITLE = CONSISTENT ELEMENT NORMAL DIRECTIONS
LOAD = 100
.
.
BEGIN BULK
$
CQUAD4,1,10,1,4,5,2
CQUAD4,2,10,2,5,6,3
CQUAD4,3,10,4,7,8,5
CQUAD4,4,10,5,8,9,6
$
PLOAD4,100,1,100.0,,,THRU,2
PLOAD4,100,3,100.0
PLOAD4,100,4,100.0
.
.
ENDDATA
```

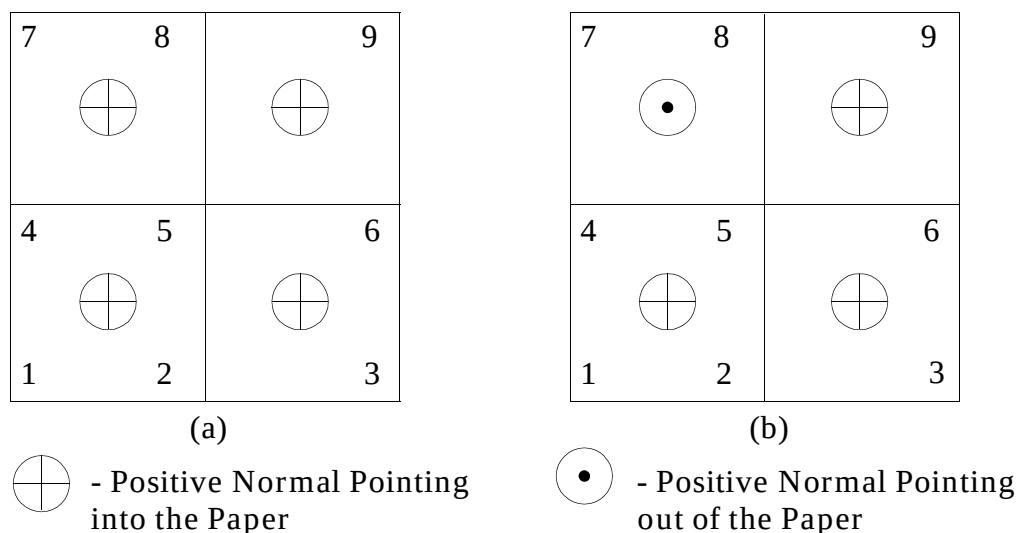


Figure 10-8 CQUAD4s with Pressure Loads

On the other hand, perhaps for some reason, you connect element 3 in a counterclockwise direction (**Figure 10-8(b)**) instead of a clockwise direction like the other three elements. If this is the case, you then need to change the sign of the pressure load for element number 3 in order to have identical loads for both cases (see **Listing 10-2**). Determining the positive direction of the pressure load for the CQUAD4 element can be achieved by checking the connectivity order using the right-hand rule. If the sign for the pressure load applied to element number 3 is not switched, then you have three elements with the pressure load pointing into the paper and the fourth element with the pressure load pointing out of the paper.

Listing 10-2 Inconsistent CQUAD4 Connectivity

```
ID RUN1 PLOAD4
SOL 101
CEND
TITLE = INCONSISTENT ELEMENT NORMAL DIRECTIONS
LOAD = 100
.
.
BEGIN BULK
$
CQUAD4,1,10,1,4,5,2
CQUAD4,2,10,2,5,6,3
CQUAD4,3,10,4,5,8,7
CQUAD4,4,10,5,8,9,6
$
PLOAD4,100,1,100.0,,,,,THRU,2
PLOAD4,100,3,-100.0
PLOAD4,100,4,100.0
.
.
ENDDATA
```


This type of inconsistency can occur if the whole model is generated with a preprocessor using multiple surfaces and the consistency of the connectivity direction is not retained. This problem is also likely to occur if you edit some of the elements manually and forget to account for the sign change in the pressure load. Many graphics packages offer this element normal check feature. This type of check requires very little effort, and in many cases the graphics processor may also allow you to reverse the direction of the normals.

The CQUAD4 element outputs are in the element coordinate system. Different connectivity orders affect the way the results are printed. If you are not aware of this fact, you may interpret the results improperly (see the example in “[Stress Error Estimators](#)” on page 418).

10.3 Strain Energy Output

Frequently you are faced with the decision of modifying your structure in order to meet stringent design criteria. In most instances, your objective is to maximize the benefits with minimal changes. The element strain energy output is an excellent tool for identifying areas of modification that will reap the most benefits for design changes.

The element strain energy is basically the elastic energy stored in the structural element. As an example, if you hold onto one end of the spring and push slowly on the other end starting from rest, the load deflection curve looks something like **Figure 10-9** for small deflections.

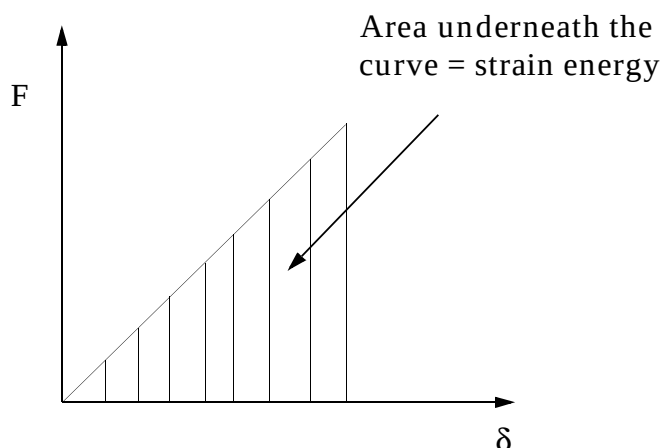


Figure 10-9 Load Deflection Curve

This energy is defined as

$$U = 0.5 \cdot F \cdot \delta$$

Since $F = k \cdot \delta$,

$$U = 0.5 \cdot k \delta^2 \quad \text{Eq. 10-1}$$

By working through a couple of examples, you can see how the element strain energy output is used to identify areas to be modified in order to reduce deflections.

The problem of interest is shown in **Figure 10-10**. It consists of two springs in a series with a tip load applied at grid point 2. The stiffnesses of the springs and the load applied are as shown in **Figure 10-10**. The goal is to reduce the tip deflection at grid point 2. Without performing any calculation, it is quite obvious that stiffening K_2 is more efficient than stiffening K_1 . The next step is to calculate the strain energy and see if it also guides you in the same direction.

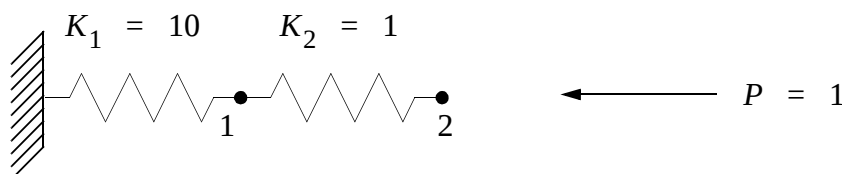


Figure 10-10 Spring Sample Problem

The deflections at grid points 1 and 2 can be calculated as follows:

$$\delta_1 = \frac{P}{K_1}; \delta_2 = \frac{P}{K_1} + \frac{P}{K_2} \quad \text{Eq. 10-2}$$

From **Eq. 10-1** and **Eq. 10-2**,

$$U_1 = 0.5 (K_1)(\delta_1)^2 = 0.5 (10)\left(\frac{1}{10}\right)^2 = 0.05$$

$$U_2 = 0.5(K_2)(\delta_2 - \delta_1)^2 = 0.5(1)\left[\left(\frac{1}{10}\right) + \left(\frac{1}{1}\right) - \left(\frac{1}{10}\right)\right]^2 = 0.5$$

As you can see, U_2 is an order of magnitude larger than U_1 . Therefore, it concurs with our intuition that stiffening K_2 is more effective than stiffening K_1 for reducing the deflection at the tip.

The second example is a classic cantilever beam with a vertical tip load applied at the end as shown in **Figure 10-11**. This finite element model is made up of five bar elements of equal length with square cross-sectional properties (0.05m x 0.05m) as shown in **Table 10-1**.

Table 10-1 Cross-Sectional Properties of a Cantilever Beam

Elem. No.	L (m)	W (m)	D (m)	A (m ²)	I ₁ (m ⁴)	I ₂ (m ⁴)	J (m ⁴)
1	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
2	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
3	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
4	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
5	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7

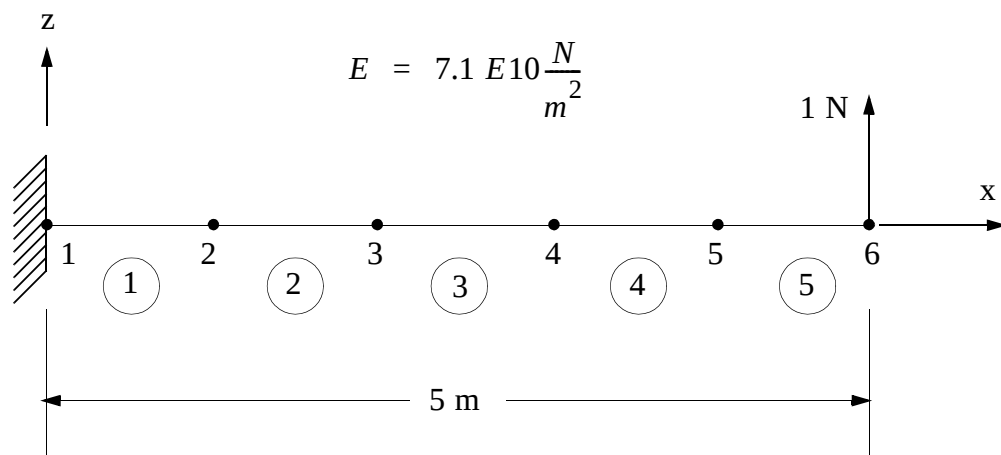


Figure 10-11 Cantilever Beam

As is commonly known, the largest deflection occurs at the tip of the cantilever beam. The vertical deflection at grid point 6 is equal to $.126833 \cdot 10^{-3}$ meters. The goal is to minimize this deflection at grid point 6. To make the problem more interesting, hypothetically impose the following constraint such that all that is available is one structural member that is $0.06\text{m} \cdot 0.06\text{m}$ and 1m long. In other words, you can only replace one of the existing five CBAR elements with this new CBAR element. Which one should you replace to minimize the deflection at grid point 6? Since grid point 6 has the largest deflection, combined with our experience with the spring problem, you may be tempted to replace CBAR element number 5 with this new element. If you do this, the vertical displacement at grid point 6 is then reduced from $.126833 \cdot 10^{-3}$ meters down to $.122165 \cdot 10^{-3}$ meters, a reduction of only 0.414%. Now run this problem using MSC.Nastran and see what the element strain energy output suggests. [Listing 10-3](#) contains a listing of the MSC.Nastran input file used for this purpose. Note that the only additional request required is the following command in the Case Control Section:

```
ESE = ALL
```

or

```
SET a = x,y,.....
```

```
ESE = a
```

or

```
ESE(PLOT) = ALL
```

For large models, the element strain energy request can potentially generate a large amount of printout. If this is not desired, you can use the “plot” option shown above to create a postprocessing file containing element strain energy data without generating printed output (see [Section ?](#)). The rest of the input file is standard MSC.Nastran input.

Listing 10-3 Input File for the Strain Energy of a Cantilever Beam

```

$
$  FILENAME -  secant.dat
$
ID CANT ESE
SOL 101
TIME 10
CEND
$
TITLE = BASELINE MODEL
SUBTITLE =
$
SPC = 10
LOAD = 10
$
SUBCASE 1
SET 1 = 6
DISP = 1
ESE = ALL
$
BEGIN BULK
$
CBAR    1      10      1      2      10
CBAR    2      100     2      3      10
CBAR    3      100     3      4      10
CBAR    4      100     4      5      10
CBAR    5      20      5      6      10
FORCE   10      6      1.      0.      0.      1.
GRID    1      0.      0.      0.
GRID    2      1.      0.      0.
GRID    3      2.      0.      0.
GRID    4      3.      0.      0.
GRID    5      4.      0.      0.
GRID    6      5.      0.      0.
GRID   10      0.      0.      10.
MAT1    1      7.1+10   .33      2700.
PBAR   10      1      2.5-3   5.208-7 5.208-7 8.789-7
PBAR   20      1      2.5-3   5.208-7 5.208-7 8.789-7
$
$  REPLACE PBAR,10, ...
$  WITH THE FOLLOWING FOR .06 X .06 CROSS SECTION
$
$PBAR,10,1,3.6-3,1.08-6,1.08-6,1.823-6
$
PBAR   100      1      2.5-3   5.208-7 5.208-7 8.789-7
SPC1   10      123456  1
$
ENDDATA

```

The MSC.Nastran element strain energy output for this run is summarized in [Figure 10-12](#). As it turns out, element number 1 has the highest element strain energy among the five elements. In fact, it contains 48.8% of the total strain energy of the whole structure. On the other hand, element number 5 contains the lowest element strain energy among the five elements--only 0.8%. In other words, element number 5 is the least effective element to modify if you wish to increase the stiffness of the structure for the same amount of weight increase.

This job is now rerun by replacing element number 1 with the new 0.06×06 element. The deflection at grid point 6 is now reduced from 1.126833×10^{-3} meters down to 8.421097×10^{-4} meters, or a reduction of 25.27% as compared to a mere 0.414% reduction if you had replaced element number 5 instead.

E L E M E N T S T R A I N E N E R G I E S				
ELEMENT-TYPE = BAR		* TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM	=	5.634164E-04
SUBCASE	1	TOTAL ENERGY OF ALL ELEMENTS IN SET	-1 =	5.634164E-04
			*	
	ELEMENT-ID	STRAIN-ENERGY	PERCENT OF TOTAL	STRAIN-ENERGY-DENSITY
	1	2.749472E-04	48.8000	1.099789E-01
	2	1.667712E-04	29.6000	6.670850E-02
	3	8.563929E-05	15.2000	3.425572E-02
	4	3.155132E-05	5.6000	1.262053E-02
	5	4.507331E-06	.8000	1.802932E-03
TYPE = BAR	SUBTOTAL	5.634164E-04	100.0000	

Figure 10-12 Element Strain Energy Output for a Cantilever Beam Model

As you can see, the element strain energy output is an extremely useful tool in helping you to identify the most efficient locations for modification. The same concept used for the previous two simple examples can be applied to complex models in the same manner. Again, the only additional input needed is the “ESE” request in the Case Control Section. The output format is identical to that of [Table 10-2](#). Furthermore, most modern plot packages also support element strain energy output. For plate and solid elements, these strain energy plots look similar to stress contour plots.

A car model (courtesy of Lapcad Engineering), containing CQUAD4 and CTRIA3 elements, is used to illustrate the element strain energy contour plot. The Executive Control and Case Control Sections are shown in [Listing 10-4](#). As you can see, the amount of user input required is quite small. The Bulk Data for this model is inserted into the input file using the INCLUDE statement. A 1g loading is applied to each of the three orthogonal axes. The element strain energy contour plot for a 1g loading in the z-direction is shown on a deformed plot in [Figure 10-13](#).

Listing 10-4 Input File for the Strain Energy of a Car Model

```
$  
$ FILENAME - secar.dat  
$  
$   MODEL COURTESY OF LAPCAD ENGINEERING  
$  
ID     LSUG, CAR  
SOL 101  
TIME 60  
CEND  
TITLE = CAR MODEL WITHOUT SUSPENSION SPRINGS  
ECHO = UNSORT  
SPC = 10  
DISP(PLOT) = all  
ESE(PLOT) = all  
$  
SUBCASE 10  
LABEL = 1G LOAD IN X-DIRECTION  
LOAD = 1  
$  
SUBCASE 20  
LABEL = 1G LOAD IN Y-DIRECTION  
LOAD = 2  
$  
SUBCASE 30  
LABEL = 1G LOAD IN Z-DIRECTION  
LOAD = 3  
$  
BEGIN BULK  
$  
INCLUDE 'carbulk.dat'  
$  
ENDDATA
```

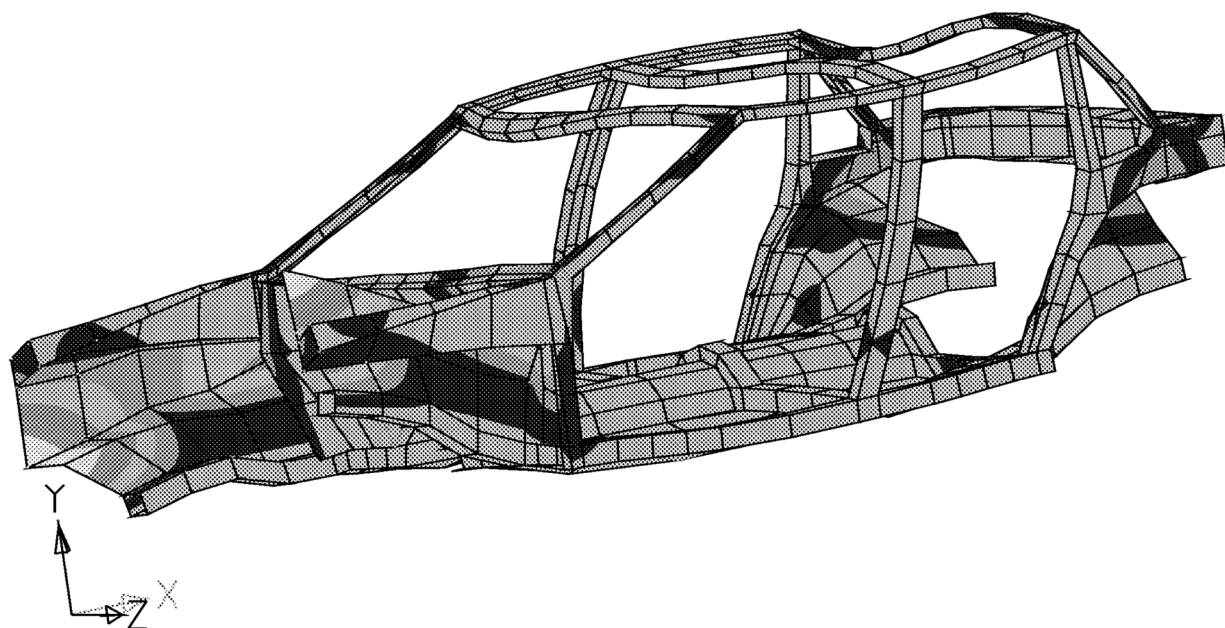


Figure 10-13 Element Strain Energy Contour Plot of an Automobile Model

(Geometry model courtesy of LAP CAD Engineering)

10.4 Diagnostic Tools

There are numerous diagnostic tools available in MSC.Nastran for debugging and understanding the solution process. Some of them are standard output; others are available with requests in the Executive Control Section, Case Control Section, and / or Bulk Data Section.

This section covers many of these tools. Whenever possible, the problem in Listing 10-5 is used to demonstrate these features. The problem in Listing 10-5 is similar to the cantilever beam shown in Figure 10-5 (Listing 10-3) with the exception of an additional load case (gravity load) and output request. The modifications and additions are indicated by the shaded regions.

Listing 10-5 Input File for Illustrating Diagnostic Checks

```
$
$ FILENAME - diag.dat
$
SOL 101
TIME 10
CEND
$
TITLE = DIAGNOSTIC TOOLS
$
SPC = 10
DISP = ALL
SPCF = ALL
OLOAD = ALL
$
SUBCASE 1
LABEL = VERTICAL TIP LOAD
LOAD = 10
$
SUBCASE 2
LABEL = GRAVITY LOADS
LOAD = 20
$
BEGIN BULK
$
PARAM, GRDPNT, 0
$
CBAR    1      10      1      2      10
CBAR    2      10      2      3      10
CBAR    3      10      3      4      10
CBAR    4      10      4      5      10
CBAR    5      10      5      6      10
FORCE   10     6      100.    0.    0.    1.
GRAV    20     9.8    0.    0.    1.
GRID    1      0.    0.    0.
GRID    2      1.    0.    0.
GRID    3      2.    0.    0.
GRID    4      3.    0.    0.
GRID    5      4.    0.    0.
GRID    6      5.    0.    0.
GRID    10     0.    0.    10.
$      2      3      4      5      6
MAT1    1      7.1+10 .33    2700.
PBAR    10     1      2.5-3  5.208-7 5.208-7 8.789-7
SPC1    10     123456 1
$
ENDDATA
```

Element Summary Output

The ELSUM Case Control command provides a quick summary table of properties for the elements in the model. These properties include element-id, material-id, length (or thickness), area, volume, structural mass, non-structural mass, total mass, and weight (weight mass * total mass). ELSUM provides an extension of the element summaries previously obtained with PARAM,EST,1.

The following examples highlight some of the new features available via the ELSUM command. Note that the the ELSUM command on line 11 in subcase 100 requests summary information for those elements defined in SET 1. The corresponding rod element output follows.

```

0
0
CASE CONTROL ECHO
COMMAND
COUNT
1 $
2 SET 1 = 401 THRU 403
3 DISP(PRINT,PUNCH)=ALL
4 ELST(PRINT,PUNCH)=ALL
5 ELFO(PRINT,PUNCH)=ALL
6 SPCF(PRINT,PUNCH)=ALL
7 ESE(PRINT,PUNCH)=ALL
8 SPC=50
9 $
10 SUBCASE 100
11 ELSUM=1
12 LOAD=100
13 ESE(PRINT)=ALL
14 $
15 BEGIN BULK

```

```

0
ELEMENT TYPE = ROD
ELEMENT PROPERTY SUMMARY
ID      MID      LENGTH      AREA      VOLUME      SM      NSM      TM      WEIGHT
401      202      1.000000E+01  6.000000E+00  6.000000E+01  4.800000E+02  5.000000E+00  4.850000E+02
4.850000E+02
402      201      1.000000E+01  4.000000E+00  4.000000E+01  1.600000E+02  8.000000E+00  1.680000E+02
1.680000E+02
403      202      1.000000E+01  6.000000E+00  6.000000E+01  4.800000E+02  5.000000E+00  4.850000E+02
4.850000E+02
SUBTOTAL MASS =      1.120000E+03  1.800000E+01  1.138000E+03
1.138000E+03
TOTAL MASS =      1.120000E+03  1.800000E+01  1.138000E+03
1.138000E+03

```

Element Geometry Checks

Overview

Mathematical models are usually composed of several different types of finite elements depending upon the simulation being performed and the type of structure being investigated. For each of these finite element formulations, thorough evaluations of the geometry presented are performed prior to and during element matrix generation operations. These geometry tests are typically used to determine whether (1) the element

geometry is adequate for finite element matrix generation at all, and if so, (2) how close the geometry could be to producing poor formulations. Taking a beam element for example, a fatal geometry condition would be locations of the two end points that result in an element with zero length. An example of the second type of geometry check for a beam would be an excessive ratio of the length of the beam with offset vectors to the length of the beam without them. In the first case, the geometry check is fatal because the finite element cannot be formulated from the given geometry and the simulation cannot be done. In the second case, the geometry check is informational and the simulation analysis can proceed. It is the user's responsibility to inspect these informational messages and determine if the results are acceptable.

Several different basic geometry tests are performed; for example, duplicate grid point location tests, mid-side node location tests, warped element tests, and aspect ratio tests. One or more of these types of tests is performed using the geometry supplied for each finite element. Obviously, not all tests apply to every element type. Each one of the geometry tests is accompanied by a message that can generate up to four lines of informational output for an element if the test tolerance is violated. For models that employ a small number of elements, this amount of output is not overwhelming. However, the size and complexity of mathematical simulation models have continued to grow over time. It is no longer surprising to see models that generate hundreds of thousands of lines of output from these tests alone. In an effort to reduce the amount of output generated by geometry checks, a new style of output for reporting the results of the element geometry tests has been introduced. The results for all tests for a single element are reported on a single line together with a visual indication of which tests have exceeded their tolerances and the severity of the failure. Default values for all test tolerances are defined such that geometry tests result in the same type of pass/fail decision as previous releases. In a few cases, additional tests are performed. At the same time, the new GEOMCHECK executive statement has been introduced to give users more control over the test tolerances, their severity levels and the number of messages output.

GEOMCHECK Statement

The GEOMCHECK statement provides users with a way to override the default values supplied for all tests. (Refer to the [MSC.Nastran Quick Reference Guide](#) for a full description of the GEOMCHECK executive statement.) Except for a very few isolated instances, bad geometry that does not allow the formulation of finite element matrix data is always fatal and is not under user control. In these cases, the informational message formats have not been modified. Every test that fails will produce a message for every element. For the cases where geometry does not prevent finite element matrix generation, additional tests that evaluate various geometry parameters are performed and the results reported to the user. For these tests, the user can control both the test tolerance and the severity level of test failure. Using the BAR element as an example, there is a test performed that evaluates the ratio of the BAR's length with offset effects to the length without offset effects. The tolerance for this test is 15%, meaning that if the length with offset is different from the length without the offset by more than fifteen percent, an informational message is issued and element processing continues. The user can change the tolerance if desired. The severity of the test failure can also be changed from

informational to fatal if desired. In that case, any BAR element that exceeds the offset ratio test tolerance will cause the job to stop after element matrix generation has been attempted for all elements.

The GEOMCHECK statement can only be used to modify geometry test activities for the QUAD4, QUADR, TRIA3, TRIAR, BAR, BEAM, HEXA, PENTA, and TETRA finite elements at this time. As stated previously, only informational tests can be affected using the GEOMCHECK statement. Results of geometry generated by MSC.Nastran that prevent the generation of a finite element matrix cannot be modified using the GEOMCHECK statement.

Keywords are available that allow control over the number of messages to be generated (MSGLIMIT) as well as the severity of the message (MSGTYPE) and its associated effect on the job. Only a severity of FATAL will cause the job to abort after the element matrix generation module has executed. In this case, the identifier associated with failed tests is **FAIL** and the run will also terminate.

Geometry Test Descriptions

Several different types of geometry tests are performed. For each finite element type, the geometry must be tested to ensure that it is adequate to allow generation of element matrices. No attempt is made here to describe those tests. Rather, tests that optionally evaluate certain characteristics of the geometry are described. These tests fall into several categories depending upon the element type. Thus, not all tests are applicable to every element type. The discussion that follows summarizes the optional geometry tests that are applicable to various element types.

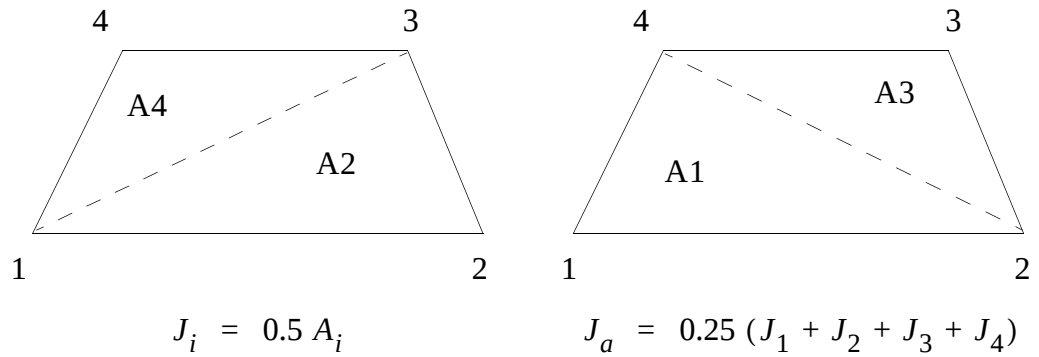
BAR and BEAM

The only optional test available for these element types is the offset test. The length of the element offset is compared to the original length of the element. If the ratio of these lengths is greater than the tolerance, an informational message is issued identifying the element and its length with and without the effects of offset.

CQUAD4 and CQUADR

Four optional tests are performed on the quadrilateral shaped elements. These tests are:

1. Interior Angle test: This test evaluates the interior angles measured at each of the four corner grid points. If any one of the four angles exceeds minimum or maximum tolerance levels, an informational message is produced.
2. Taper test: The QUAD4 taper ratio test compares the area of the triangles formed at the four corner nodes (the corner node and the two nodes attached to it via the two element edges) to the total area of the QUAD4. The minimum of the four values is taken as the taper ratio. If the ratio exceeds the tolerance, an informational message is produced.



Taper message is issued if $\frac{J_i - J_a}{J_a} > 0.5$

Figure 10-14 CQUAD4 Taper

3. Skew test: This test evaluates the distortion or “parallelogram shape effect” by measuring the angle between lines that join the midpoints of opposite sides of the element. If the angle exceeds the tolerance, an informational message is produced.

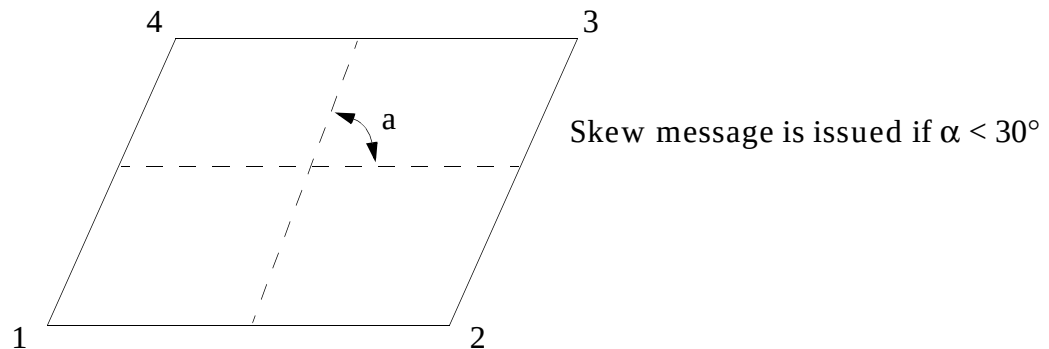


Figure 10-15 CQUAD4 Skew

4. Warp test: This test evaluates how far out of plane the four corner grid points are by measuring the distance of each point from a “mean” plane passing through the locations of the four points. The corner points are alternately H units above and H units below this mean plane. If the lengths of the diagonals of the element are denoted by D1 and D2, the warping coefficient is obtained from the equation $WC = H / 2(D1+D2)$. If this value exceeds the tolerance, an informational message is produced.

CTRIA3 and CTRIAR

Only one optional test is performed on the triangular shaped planar elements. It is the same as the interior angle test discussed previously under the CQUAD4 and CQUADR test description.

CHEXA, CPENTA and CTETRA

Five optional tests are performed on the solid element family. Not all tests are applicable to every element type. These tests are:

1. Aspect Ratio test: This test evaluates the ratio of the longest length (edge or height) to the shortest length (edge or height) encountered in the element. If the ratio exceeds the tolerance, an informational message is produced.
2. Edge Point Length Ratio test: This test evaluates the location of the “mid-side” nodes if any are present. The node should be located on the line connecting the two adjacent corner grid points at approximately the mid-way point. This determination is made in two ways. First, the distance of the node from the two corner nodes is measured and if the ratio of the distance from one node to the other exceeds the tolerance, an informational message is issued. Next, the angles between the lines connecting the mid-side node to its two adjacent corner nodes, and the line connecting the corner nodes themselves, is evaluated. If either of the angles exceeds 30 degrees, an informational message is issued.
3. Integration Point Jacobian Determinant test: This test evaluates the determinant of the Jacobian at each integration point. If it is zero, or changes sign from integration point to integration point, an informational message is issued. Note that detection of a zero value should always be fatal because element matrices calculated from such geometry rarely produce satisfactory analysis results.
4. Warped Face test: This test is applicable to CHEXA and CPENTA elements. It evaluates the warping coefficient of the quadrilateral faces of such elements in a manner similar to that discussed previously under the CQUAD4 element warping test description. If the warping coefficient exceeds the tolerance, an informational message is issued.

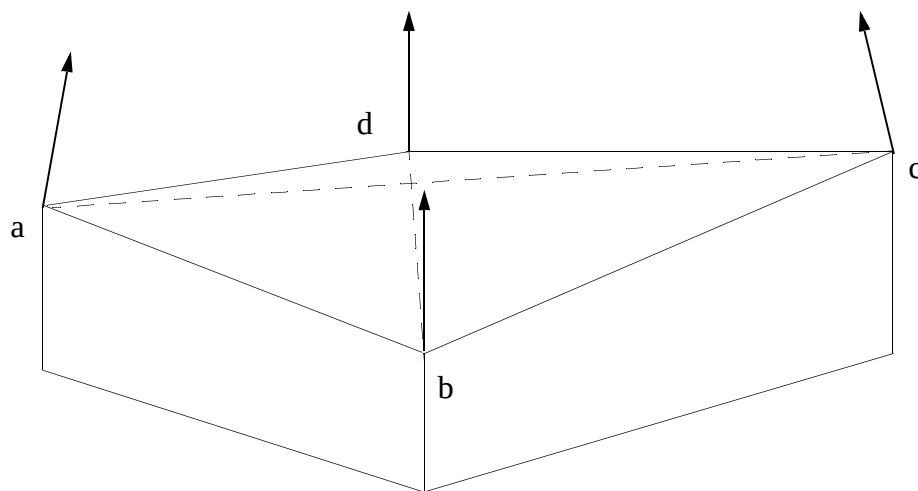


Figure 10-16 CHEXA - Warping

5. Grid Point Jacobian Determinant test: This test is applicable to CTETRA elements only. The test is the same as that done for the integration point test discussed previously except it uses the location of the corner grid points to perform the test. If any determinant is zero or changes sign, an informational message is produced.

User Interface:

The geometry check is requested by the GEOMCHECK Executive Control Statement and the format is as follows:

GEOMCHECK test_keyword [= tol_value], [MSGLIMIT=n], [MSGTYPE = FATAL
INFORM
WARN], [SUMMARY], [NONE]

The following table summarizes the acceptable specifications for the test_keyword:

Name	Value Type	Default	Comment
Q4_SKEW	Real ≥ 0.0	30.0	Skew angle in degrees
Q4_TAPER	Real ≥ 0.0	0.50	Taper ratio
Q4_WARP	Real ≥ 0.0	0.05	Surface warping factor
Q4_IAMIN	Real ≥ 0.0	30.0	Minimum Interior Angle in degrees
Q4_IAMAX	Real ≥ 0.0	150.0	Maximum Interior Angle in degrees
T3_SKEW	Real ≥ 0.0	10.0	Skew angle in degrees
T3_IAMAX	Real ≥ 0.0	160.0	Maximum Interior Angle in degrees
TET_AR	Real ≥ 0.0	100.0	Longest edge to shortest edge aspect ratio
TET_EPLR	Real ≥ 0.0	0.50	Edge point length ratio
TET_DETJ	Real	0.0	J minimum value
TET_DETG	Real	0.0	J minimum value at vertex point
HEX_AR	Real ≥ 0.0	100.0	Longest edge to shortest edge aspect ratio
HEX_EPLR	Real ≥ 0.0	0.50	Edge point length ratio
HEX_DETJ	Real	0.0	J minimum value
HEX_WARP	Real ≥ 0.0	0.707	Face warp coefficient
PEN_AR	Real ≥ 0.0	100.0	Longest edge to shortest edge aspect ratio
PEN_EPLR	Real ≥ 0.0	0.50	Edge point length ratio
PEN_DETJ	Real	0.0	J minimum value
PEN_WARP	Real ≥ 0.0	0.707	Quadrilateral face warp coefficient
BEAM_OFF	Real ≥ 0.0	0.15	CBEAM element offset length ratio
BAR_OFF	Real ≥ 0.0	0.15	CBAR element offset length ratio

Examples:

1. GEOMCHECK Q4_SKEW=15.0,MSGLIMIT=50

Set the tolerance for the CQUAD4 element skew angle test to 15.0 degrees and limit the number of messages to 50.

2. GEOMCHECK SUMMARY

Requests summary table output only using the default tolerance values.

Sample Output:

The following output from a small example provides a sample of the new message formats that are produced for geometry tests that can be controlled using the GEOMCHECK statement. For this output, most of the default test tolerances were modified so that the effect could be observed in the tolerance information lines of the messages that were produced. Also note that a small table is generated at the end of all messages that summarizes the number of tests that actually exceeded the tolerance value for each element type as well as a list of the elements that produced the worst violations. The summary table itself can be produced by using the keyword SUMMARY on the GEOMCHECK statement.

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
 FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
 User Action: Use the GEOMCHECK (Executive Control Statement) keyword=value to change tolerance values if desired.
 A MINIMUM OF 100 OFFSET LENGTH RATIO TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.

TOLERANCE LIMIT IS: BAR_/BEAM_OFF(SET) = .15 (xxxx = LIMIT VIOLATED)
 ELEMENT TYPE ID LENGTH W/O OFFSET LENGTH W/ OFFSET OFFSET LENGTH RATIO
 BAR 7101 1.00000E+00 3.66367E+00 2.66 xxxx

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
 FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
 A MINIMUM OF 4 OFFSET LENGTH RATIO TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

TOLERANCE LIMIT IS: BAR_/BEAM_OFF(SET) = -.02 (WARN = LIMIT VIOLATED)
 ELEMENT TYPE ID LENGTH W/O OFFSET LENGTH W/ OFFSET OFFSET LENGTH RATIO
 BEAM 8101 1.00000E+00 1.10372E+01 10.04 WARN
 BEAM 8102 1.00000E+00 1.00000E+00 .00 WARN

*** USER INFORMATION MESSAGE 7555 (EHEXGD)
 FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
 A MINIMUM OF 100 EDGE LENGTH ASPECT RATIO (AR) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 EDGE NODE POINT LENGTH RATIO (ER) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 EDGE ANGLE VALUES ARE INDICATED BY AN * AFTER THE VALUE. THE TOLERANCE LIMIT IS 30 DEGREES.
 ALL NEG/ZERO DET(JACOBIAN) VALUE (DJ) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY FAIL.
 A MINIMUM OF 100 WARPED FACE COEFFICIENT TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.

TOLERANCE LIMITS ARE: HEX_AR = 300.00, HEX_EPLR = .40, HEX_DETJ = .01, HEX_WARP = .65 (xxxx = LIMIT VIOLATED)
 ELEMENT TYPE ID LONGEST EDGE SHORTEST EDGE ASPECT RATIO EDGE POINT LENGTH RATIO MIN. JACOBIAN DETERMINANT FACE WARP COEFFICIENT
 HEXA 8602 4.00 1.00 4.00 N/A- ONLY 8 NODE .92 .52 xxxx
 HEXA 8620 4.00 1.00 4.00 .60xxxx .50 1.00

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
 FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
 A MINIMUM OF 100 SKEW ANGLE (SA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 MIN INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 MAX INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 WARPING FACTOR (WF) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 TAPER RATIO (TR) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.

TOLERANCE LIMITS ARE: SA = 30.00, IA(MIN) = 30.00, IA(MAX) = 150.00, WF = .05, TR = .50 (xxxx = LIMIT VIOLATED)
 ELEMENT TYPE ID SKEW ANGLE MIN INT. ANGLE MAX INT. ANGLE WARPING FACTOR TAPER RATIO
 QUAD4 105 29.74 xxxx 29.05 xxxx 150.95 xxxx .00 .05
 QUAD4 106 28.39 xxxx 27.76 xxxx 152.24 xxxx .00 .05
 QUAD4 107 27.15 xxxx 26.57 xxxx 153.43 xxxx .00 .05
 QUAD4 108 26.00 xxxx 25.46 xxxx 154.54 xxxx .00 .05
 QUAD4 109 24.94 xxxx 24.44 xxxx 155.56 xxxx .00 .05
 QUAD4 2105 29.74 xxxx 29.05 xxxx 150.95 xxxx .00 .05
 QUAD4 2106 28.39 xxxx 27.76 xxxx 152.24 xxxx .00 .05
 QUAD4 2107 27.15 xxxx 26.57 xxxx 153.43 xxxx .00 .05
 QUAD4 2108 26.00 xxxx 25.46 xxxx 154.54 xxxx .00 .05
 QUAD4 2109 24.94 xxxx 24.44 xxxx 155.56 xxxx .00 .05

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
 FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
 A MINIMUM OF 100 SKEW ANGLE (SA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 MIN INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 MAX INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 WARPING FACTOR (WF) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.
 A MINIMUM OF 100 TAPER RATIO (TR) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY xxxx.

TOLERANCE LIMITS ARE: SA = 30.00, IA(MIN) = 30.00, IA(MAX) = 150.00, WF = .05, TR = .50 (xxxx = LIMIT VIOLATED)

ELEMENT TYPE	ID	SKEW ANGLE	MIN INT. ANGLE	MAX INT. ANGLE	WARPING FACTOR	TAPER RATIO
QUADR	1105	29.74 xxxx	29.05 xxxx	150.95 xxxx	.00	.05
QUADR	1106	28.39 xxxx	27.76 xxxx	152.24 xxxx	.00	.05
QUADR	1107	27.15 xxxx	26.57 xxxx	153.43 xxxx	.00	.05
QUADR	1108	26.00 xxxx	25.46 xxxx	154.54 xxxx	.00	.05
QUADR	1109	24.94 xxxx	24.44 xxxx	155.56 xxxx	.00	.05

*** USER INFORMATION MESSAGE 7555 (ETETGD)

FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.

A MINIMUM OF 4 EDGE LENGTH TO HEIGHT RATIO (AR)TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

A MINIMUM OF 4 EDGE NODE POINT LENGTH RATIO (ER)TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

EDGE ANGLE VALUES ARE INDICATED BY AN * AFTER THE VALUE. THE TOLERANCE LIMIT IS 30 DEGREES.

A MINIMUM OF 4 NEG/ZERO DET(JACOBIAN) VALUE (DJ)TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

A MINIMUM OF 4 NEG DET(JAC) AT VERTEX NODE (DG)TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

TOLERANCE LIMITS ARE: TET_AR = 351.00, TET_EPLR = .45, TET_DETJ = -.01, TET_DETG = -.02 (xxxx = LIMIT VIOLATED)

ELEMENT TYPE	ID	LONGEST EDGE	SHORTEST HEIGHT	ASPECT RATIO	EDGE POINT LENGTH RATIO	MIN. JACOBIAN DETERMINANT	MIN. DET(JAC) AT VERTEX
TETRA	6601	1.41	.58	2.45	.25 WARN	.14	-.20 WARN

ELEMENT GEOMETRY TEST RESULTS SUMMARY

		TOTAL NUMBER OF TIMES TOLERANCES WERE EXCEEDED				EDGE POINT		
		ASPECT/	MINIMUM	MAXIMUM	SURFACE/FACE			
JACOBIAN								
ELEMENT TYPE	SKEW ANGLE	TAPER RATIO	INTER. ANGLE	INTER. ANGLE	WARP FACTOR	OFFSET RATIO	LENGTH RATIO	
DETERMINANT								
BAR	N/A	N/A	N/A	N/A	N/A	0	N/A	N/A
BEAM	N/A	N/A	N/A	N/A	N/A	0	N/A	N/A
HEXA	N/A	0	N/A	N/A	1	N/A	1	0
PENTA	N/A	0	N/A	N/A	0	N/A	1	0
QUAD4	10	0	10	10	0	N/A	N/A	N/A
QUADR	5	0	5	5	0	N/A	N/A	N/A
TETRA	N/A	0	N/A	N/A	N/A	N/A	1	1
TRIA3	0	N/A	N/A	0	N/A	N/A	N/A	N/A
TRIAR	0	N/A	N/A	0	N/A	N/A	N/A	N/A

N/A IN THE ABOVE TABLE INDICATES TESTS THAT ARE NOT APPLICABLE TO THE ELEMENT TYPE AND WERE NOT PERFORMED. FOR ALL ELEMENTS WHERE GEOMETRY TEST RESULTS HAVE EXCEEDED TOLERANCES,

HEXA	ELEMENT ID	8620	PRODUCED SMALLEST	EDGE COS(ANGLE)	OF	.60 (TOLERANCE =	.87).
HEXA	ELEMENT ID	8602	PRODUCED SMALLEST	FACE WARP FACTOR	OF	.52 (TOLERANCE =	.65).
PENTA	ELEMENT ID	3915	PRODUCED SMALLEST	EDGE COS(ANGLE)	OF	.78 (TOLERANCE =	.87).
QUAD4	ELEMENT ID	109	PRODUCED SMALLEST	SKEW ANGLE	OF	24.94 (TOLERANCE =	30.00).
QUAD4	ELEMENT ID	109	PRODUCED SMALLEST	INTERIOR ANGLE	OF	24.44 (TOLERANCE =	30.00).
QUAD4	ELEMENT ID	109	PRODUCED LARGEST	INTERIOR ANGLE	OF	155.56 (TOLERANCE =	150.00).
QUADR	ELEMENT ID	1109	PRODUCED SMALLEST	SKEW ANGLE	OF	24.94 (TOLERANCE =	30.00).
QUADR	ELEMENT ID	1109	PRODUCED SMALLEST	INTERIOR ANGLE	OF	24.44 (TOLERANCE =	30.00).
QUADR	ELEMENT ID	1109	PRODUCED LARGEST	INTERIOR ANGLE	OF	155.56 (TOLERANCE =	150.00).
TETRA	ELEMENT ID	6601	PRODUCED LARGEST	EDGE POINT LR	OF	.25 (TOLERANCE =	.45).
TETRA	ELEMENT ID	6601	PRODUCED SMALLEST	VERTEX DET(JAC)	-UWM 6828- OF	-.20 (TOLERANCE =	-.02).

Weight Center of Gravity and Moment of Inertia Check

Is the weight of my model correct? This question can be answered by activating the Grid Point Weight Generator (GPWG). The GPWG is activated with the parameter GRDPNT in the Bulk Data or Case Control Section.

The data entry format is
PARAM,GRDPNT

where “i” is an integer value defining a reference point. The value of “i” can be any grid point in the model, or if it is set to zero, the reference point is the origin of the basic coordinate system.

The output from the GPWG includes a rigid body mass matrix, various coordinate transformations, and the location of the center of mass. The output from the GPWG is generally more than what you need. The mass and center of gravity (CG) location is typically all that is used. A partial output for the problem in [Listing 10-5](#) is shown in [Figure 10-17](#).

DIRECTION						
MASS	AXIS	SYSTEM (S)	MASS	X-C.G.	Y-C.G.	Z-C.G.
	X		3.375000E+01	0.000000E+00	0.000000E+00	0.000000E+00
	Y		3.375000E+01	2.500000E+00	0.000000E+00	0.000000E+00
	Z		3.375000E+01	2.500000E+00	0.000000E+00	0.000000E+00

Figure 10-17 Partial Output from the Grid Point Weight Generator

MSC.Nastran does not keep track of your units. Therefore, you must input all of your properties using a consistent set of units. For example, if you are using meters (m) for defining locations on your grid entries, then your properties, such as areas (A), should be in terms of m². MSC.Nastran expects your mass input (MATi, CONMi, etc.) to be in terms of mass units. However, if you would rather input your mass in terms of weight units, then you must add the following entry to your Bulk Data Section:

PARAM,WTMASS,

In the parameter statement above, x is the appropriate conversion factor based on the units that you are using with a default value of 1.0. In other words, the weight density is related to the mass density by the relationship

$$\rho_m = \left(\frac{1}{g}\right) \cdot \rho_w = \text{WTMASS} \cdot \rho_w$$

where:

ρ_w = weight density

ρ_m = mass density

g = gravitational acceleration constant

In the example below, the mass came from the density entered on the MAT1 entry associated with the CBAR elements. In this case, the density is in terms of mass units. Since the grid point locations are defined in terms of meters, the density should then be in units of kg / m³. Using a consistent set of units, the Young's modulus (E) and density (ρ) are 7.1E10 N / m² and 2700 kg / m³ as indicated by Fields 3 and 6, respectively, on the following MAT1 entry.

\$	MASS DENSITY (kg/m ³)				
\$					
MAT1	1	7.1+10	.33	2700.	

On the other hand, if you want to input your density in weight units, then you should replace the above MAT1 entry with the following two entries.

```
$ WEIGHT DENSITY (N/m3)
$ ACCELERATION (m/sec2)
$
MAT1      1      7.1+10      .33      2.65+4
PARAM     WTMASS  0.1019
```

This density and all the other masses (excluding M2PP or M2GG input) are multiplied by this scale factor. In this case, $(0.1019 \cdot 2.65E4) = 2700$. The results (e.g., displacements, stresses, etc.) are the same using either of the above mass or weight units.

The only difference is in the grid point weight generator output. The GPWG output is in the same units of your density or point masses. In other words, if your input is in mass units, then the GPWG output is also in mass units. If your input is in weight units, then the GPWG output is also in weight units.

If you are using English units, then the following two sets of entries yield the same results. Of course, other units in your model must also be consistent, e.g., the grid point locations and cross-sectional properties.

```
$ MASS DENSITY (lb-sec2/in4)
$
MAT1      1      1.0 +7      .33      2.51-4
```

```
$ WEIGHT DENSITY (lb/in3)
$ ACCELERATION (in/sec2)
$
MAT1      1      1.0 +7      .33      0.097
PARAM     WTMASS  0.002588
```

The output from GPWG is for informational purposes only and is not used in any subsequent steps in the solution process. An extensive discussion of the grid point weight generator is contained in “**Four Concentrated Mass Model**” in Appendix B.

Mechanisms and Singularities

When performing the solution to a system of linear equations, singularities lead to conditions in which a unique solution is not possible. MSC.Nastran considers two types of singularities:

1. Grid point singularity that is identified by considering the stiffness terms of only one grid point.
2. A mechanism type of singularity that requires the consideration of the stiffness terms of more than one grid point.

Singularities cause ill-conditioned matrices that can be detected in several phases of the MSC.Nastran execution.

After matrix assembly, the grid point singularities are detected. At each grid point, a 3x3 partition of the stiffness matrix for each of the three translational and three rotational DOFs is solved as an eigenvalue problem to determine the principal stiffnesses. Each stiffness term is compared to the principal stiffness using the formula

$$\varepsilon = \frac{K_{ii}}{K_{max}}$$

where K_{ii} is the term in the i -th row and i -th column of the matrix and K_{max} is the principal stiffness. If ε is less than the value of PARAM,EPZERO, the global direction nearest i is considered singular. The default value for EPZERO is 10^{-8} . A list of potential singularities is printed in the grid point singularity table (see “[Strain Energy Output](#)” on page 379 for further details).

PARAM,AUTOSPC. If PARAM,AUTOSPC,YES is specified (this is the default in the Structured Solution Sequences, except SOLs 106 and 129), the potential singularities are automatically constrained if possible.

During decomposition, mechanisms can be detected based on the maximum ratio of the matrix diagonal to the factor diagonal

$$\text{MAXRATIO} = \frac{K_{ii}}{D_{ii}}$$

where K_{ii} is the i -th diagonal term of the original stiffness matrix and D_{ii} is the i -th diagonal term of the factor diagonal matrix. For a symmetric matrix K it can be represented as

$$[K] = [L][D][L]^T$$

where:

$[L]$ = lower triangular factor

$[D]$ = factor diagonal matrix

PARAM,MAXRATIO. Refer to the [MSC.Nastran Numerical Methods User's Guide](#) for further details on the subject of system of linear equations. All terms whose ratio exceed the value of PARAM,MAXRATIO are printed. The default for MAXRATIO is 10^7 . UIM 4158 prints the statistics for the decomposition that include the number of negative terms on the factor diagonal and the maximum ratio of matrix diagonal to factor diagonal at a specified row number and its corresponding grid point. User Warning Message (UWM) 4698 prints the degrees of freedom that have a factor diagonal ratio greater than the MAXRATIO value or have negative terms on the factor diagonal. Both of these messages are issued by the DECOMP module and are shown below:

```

*** USER INFORMATION MESSAGE 4158---STATISTICS FOR SYMMETRIC DECOMPOSITION OF DATA BLOCK KLL    FOLLOW
      NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL =      1
      MAXIMUM RATIO OF MATRIX DIAGONAL TO FACTOR DIAGONAL = 7.2E+15 AT ROW NUMBER      16
*** 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 HAVE
NEGATIVE TERMS ON THE FACTOR DIAGONAL.
      GRID POINT ID      DEGREE OF FREEDOM      MATRIX/FACTOR DIAGONAL RATIO      MATRIX DIAGONAL

      6714                T1                -7.19297E+15                6.02908E+07

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

```

If the MAXRATIO value is exceeded, your job terminates with DMAP Fatal Message 9050. You can override this fatal message by inserting “PARAM,BAILOUT,-1” in your input file. You should, however, be aware that a large value of the MATRIX/FACTOR DIAGONAL may be an indication of a potential modeling problem. Taking the $\log_{10}$ of MAXRATIO indicates how many significant digits may have been lost during the decomposition. The MAXRATIO may change slightly if a different sequencer is used. This change is due to the fact that the order of operation may change with a different sequencer that yields a different numerical roundoff.

After decomposition, a singularity may lead to an incorrect solution. In static analysis, MSC.Nastran solves

$$Ku = P$$

to obtain u (displacements). Using these displacements, MSC.Nastran then calculates a “residual” loading vector as follows:

$$Ku - P = \delta P$$

This residual vector should theoretically be null but may not be null due to numeric roundoff. To obtain a normalized value of the residual loading, an error measure ε is calculated by

$$\varepsilon = \frac{u^T \cdot \delta P}{u^T \cdot P}$$

δP can be printed by including PARAM,IRES,1 in your Bulk Data Section. The value ε (epsilon) is printed in User Information Message (UIM) 5293 as shown below. Epsilon values that are greater than 0.001 are flagged for a possible loss of accuracy due to numeric conditioning as shown below:

```

*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
      1            -4.5961966E-14      5.6341639E+00
      2            -9.2748066E-15      9.4523830E+00

```

One ϵ is generated for each loading condition. An acceptable value of ϵ depends on the model complexity and the machine that it runs on. An epsilon value of $|\epsilon|$ in the neighborhood of less than 10^{-9} is generally considered acceptable. More details on this subject are available in References 4. and 5., and “**Constraints**” on page 257 of this user’s guide.

Some general causes for singularity can include:

- Degrees of freedom without stiffness because of missing elements.
- A 2-D plate problem with the normal rotation unconstrained.
- A solid model with rotational DOFs at the corners unconstrained.
- Incorrect modeling of offset beams.
- Incorrect multipoint constraints.
- Mechanisms and free bodies, such as sloped plates, beam to plate connections, beam to solid connections, and plate-to-solid connections.
- Low stiffness in rotation.
- A stiff element adjacent to a very flexible element.

Applied Loads Check

Are the loads being applied to the structure correctly? There are two checks available in MSC.Nastran for this purpose. The first one is the summation of the total loads applied to the structure about the reference point x as specified on the PARAM,GRDPNT,x entry. If PARAM,GRDPNT,x is not specified, this summation is performed about the origin of the basic coordinate system. This quantity is known as the OLOAD RESULTANT output and is part of the standard MSC.Nastran output. It consists of seven lines of output per load case. The OLOAD RESULTANT output for the problem in [Listing 10-5](#) is shown in [Figure 10-17](#). Note that loads from scalar points are not included in the OLOAD RESULTANT output.

SUBCASE/ DAREA ID		LOAD TYPE	OLOAD RESULTANT					
			T1	T2	T3	R1	R2	R3
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	3.307500E+02	0.000000E+00	-8.268750E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-8.268750E+02	0.000000E+00

Figure 10-18 OLOAD Resultant

Look at Subcase 1 with a vertical tip load applied to grid point 6 in the +z direction. Since there are no loads applied to the structure in the x- or y-direction, the loads in those two directions are also equal to zero. The total z-load is equal to 100 Newtons contributed solely by the applied load at grid point 6, which is the only applied load for the whole structure. The total moment about the origin of the basic coordinate system is $-5 \cdot (100 \text{ N}) = -500 \text{ Nm}$ about the y-axis. This R2 moment is contributed by the Z load as indicated by the -500 in row FZ, column R2. There is no moment about the other two axes. The next seven lines in the OLOAD RESULTANT output is due to the gravity load, the seconding loading condition.

So far, only the resultant applied load was discussed. You can also request more detailed information regarding the exact location on the structure where loads are applied. This can be done with the OLOAD request in the Case Control Section. The format is as follows:

OLOAD = All

or

SET 10 = x,y,z

OLOAD = 10

In this particular case, a printout of all the applied loads is requested. As shown in **Figure 10-19**, there is one load applied to grid point 6 in the +z direction for Subcase 1. However, for the gravity load in Subcase 2, each grid point that has mass associated with it has a 1g load applied to it. Each CBAR element contributes $\{(2700)(1.0)(2.5\text{E-}3)(9.8)/2\} = 33.075 \text{ N}$ to each end of its connecting grid points. Therefore, a 33.075 N load is applied to the end points and a 66.15 N load is applied to each of the intermediate points since there are two CBAR elements connected to each of these intermediate grid points. As you can see, for a large model, OLOAD output for gravity loads can potentially generate many lines of output.

VERTICAL TIP LOAD						SUBCASE 1		
L O A D V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
6	G	.0	.0	1.000000E+02	.0	.0	.0	
GRAVITY LOADS						SUBCASE 2		
L O A D V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	3.307500E+01	.0	.0	.0	
2	G	.0	.0	6.615000E+01	.0	.0	.0	
3	G	.0	.0	6.615000E+01	.0	.0	.0	
4	G	.0	.0	6.615000E+01	.0	.0	.0	
5	G	.0	.0	6.615000E+01	.0	.0	.0	
6	G	.0	.0	3.307500E+01	.0	.0	.0	

Figure 10-19 OLOAD Output

Reaction Load

Are the reaction loads correct? This is a fundamental equilibrium check. Again, there are two checks available in MSC.Nastran for this purpose. The first one is the summation of the total reaction loads about the reference point specified on the PARAM,GRDPNT,x entry. If PARAM,GRDPNT,x is not specified, this summation is performed about the origin of the basic coordinate system. This operation is known as the SPCFORCE RESULTANT output and this output is obtained if the SPCFORCE output is requested. It consists of seven lines of output per load case. The SPCFORCE RESULTANT output for the above problem ([Listing 10-5](#)) is shown below:

=0		SPCFORCE RESULTANT						
SUBCASE/ DAREA	ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	-1.000000E+02	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	5.000000E+02	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	-1.000000E+02	0.000000E+00	5.000000E+02	0.000000E+00
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	-3.307500E+02	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	8.268750E+02	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	-3.307500E+02	0.000000E+00	8.268750E+02	0.000000E+00

Figure 10-20 SPCFORCE RESULTANT

Since the SPCFORCE RESULTANT and OLOAD RESULTANT are summed about the same reference point, you would expect the total applied loads and total reaction loads to be equal and opposite of each other. As indicated by [Figure 10-18](#) and [Figure 10-20](#), this is indeed the case.

Similar to the applied loads, you can also request more detailed information regarding where exactly on the structure is reacting to these applied loads (grid point locations and directions). These are also known as boundary loads. This operation can be done with the SPCFORCE request in the Case Control Section. The format is as follows:

SPCFORCE = All

or

SET 10 = 15, 16, 18, 25

SPCFORCE = 10

VERTICAL TIP LOAD							SUBCASE 1		
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	-1.000000E+02	.0	5.000000E+02	.0		
GRAVITY LOADS							SUBCASE 2		
F O R C E S O F S I N G L E - P O I N T C O N S T R A I N T									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	G	.0	.0	-3.307500E+02	.0	8.268750E+02	.0		

Figure 10-21 Single-Point Constraint Forces

In this case, since there is only one reaction point (fixed at grid point 1) in the model, the SPCFORCE output only contains one line of output per subcase (see [Figure 10-21](#)). Some guidelines to consider when examining the reaction loads include:

1. Are they equal to the applied loads? Is the structure in equilibrium?
2. Are the boundary loads coming out in the directions and magnitudes that you are expecting? As an example, if one of your reaction loads is to simulate a shear bolt and you are getting a tension reaction load at that location, the chances are that you have modeled the boundary condition incorrectly.
3. Is the load distribution over the structure what you expect? If not, you may want to go back and check your modeling of your reaction points to make sure that they are what you intended.
4. Grid points with reaction force equal to 0.0 on all six components are suppressed from the printed output.

1g Load

A 1g gravitation load applied separately in each of the three orthogonal directions is suggested for model checkout. This check is accomplished with three separate subcases. This is a basic check to determine grid points on the structure that are either loosely connected or have very small stiffnesses. A displacement output indicates that these points are moving excessively as compared to other parts of the model, especially if you are using a postprocessor. In order to use this feature, you must ensure that your structure contains masses—either by direct masses (e.g., CONM2) or indirect masses (e.g., from the density on the MATi entries), or nonstructural mass (e.g., a nonzero value on the NSM field on the PBAR entry).

Output of Maximum, Minimum Grid Point-Based Responses

Overview

Documenting the validation of a mathematical simulation model can be a very tedious and time consuming process. A portion of this process usually requires the generation of reports to completely describe the results of the performed analysis. Although a substantial amount of the reporting requirements can be satisfied using graphical results

plots, tabular data that summarizes the information depicted in the graphics are often required. Also, during the validation process itself, many tools are employed that assist users in evaluating the results produced by the simulation. Maximum / minimum surveys of output quantities are one such tool provided by the new Case Control command, MAXMIN. See the [MSC.Nastran Quick Reference Guide](#) for a complete description of this new command.

For many years, MSC.Nastran users have been able to obtain simple surveys of grid point related results output such as displacements, applied loads, and forces of single point constraint. The output consists of a summary that identifies the grid point at which the maximum and minimum values occurred in each of the six output directions. This information is provided for each subcase. Several enhancements have been made that result in even more powerful views of the data. The new max / min information appears as typical grid point related output. Users can request a number of the highest and lowest values for a particular component. All components of the grid related quantities are output together with the one component that is being surveyed. For example, if 10 maximum and 10 minimum values of T3 displacements are requested, the output will also contain the T1, T2, R1, R2, and R3 displacement values associated with the grid points identified in the T3 component survey. The output heading clearly identifies the component being surveyed. Output can be surveyed for SUBCASEs and SUBCOMs. Other features include the capability to specify sets of points to be surveyed and coordinate system to be used for the survey output. Only SORT1 output is currently available.

If output coordinate systems are specified for any of the grid points being surveyed, the user has the option of specifying one of several coordinate system views of the max / min output. The data can be displayed in the basic system, a local system, or even the global system. Any grid point that is output in the max / min survey will have its output presented in both the survey coordinate frame of reference and its original frame of reference if they are different. In this way, the user can see both the relative position of the grid point component in the max / min output as well as the original output quantities without having to research other portions of the displacement output produced by the analysis.

MAXMIN Sample Output

The following sample output demonstrates some features available with the MAXMIN Case Control command.

Inspection of the Case Control Echo reveals that the MAXMIN command at line 7 requests print and punch of max / min survey results for all available output quantities generated (via keywords PRINT, PUNCH and ALL). It also specifies that survey results be limited to component directions T1 and T2 for grid points defined in case control set 100. Because the command has been placed above the subcase level, it applies to all subcases and subcoms.

The MAXMIN command at line 11 requests a survey of the forces of single point constraint output (keyword SPCF) for component direction T1 only (keyword T1) in the basic coordinate system frame of reference (keyword CID selects BASIC). It also requests that the highest and lowest 15 values be output considering ALL grid points. Because it has been placed within subcase 1000, it only pertains to that subcase.

The MAXMIN command at line 14 requests print and punch of max/min survey results for single point forces of constraint. A vector magnitude survey is requested (keyword VMAG) for the four highest and lowest values. The results for these types of surveys appear in the columns of output normally containing component directions T1 and R1. Note that the specification of component directions to be surveyed (keyword COMP) has no effect in this case. All grid points will be included in the survey.

```

0
COMMAND          C A S E   C O N T R O L   E C H O
COUNT
1  TITLE = EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND
2  SUBTITLE = ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
3  LABEL = COMPONENT DOF SPECIFICATIONS ARE POSSIBLE
4  SPC=100
5  SET 100 = 2,51,59
6  DISP=100
7  MAXMIN (PUNCH,PRINT,ALL ,COMP=T1/T2) = 100 $
8  SPCF=ALL
9  OLOAD=ALL
10 SUBCASE 1000
11 MAXMIN (SPCF,CID=BASIC,COMP=T1,BOTH=15) = ALL
12 LOAD=1000
13 SUBCASE 2000
14 MAXMIN (PUNCH,PRINT,SPCF,VMAG=4,COMP=T1/T3) = ALL
15 LOAD=1000
16 SUBCOM 3000
17 SUBSEQ = 0.5,0.5
18 BEGIN BULK

```

Examination of the following actual MAXMIN survey output results reveals that the three heading lines contain important information that describe the options used to produce the results. The first line identifies the component direction that was searched, the type of output being searched, and the subcase/subcom identification number. The second line summarizes the options specified on the MAXMIN Case Control command used to perform the search. The third line contains column headings for the data values that follow in the output listing. Note that the field, TYPE, identifies whether it is a grid point (G) or scalar point (S) and the field, POINT ID, identifies the grid point or scalar ID number. In addition, the coordinate system reference frame for the output values is identified. If data values for a point were transformed to a reference frame different from the one originally defined in the output, then the data values for both frames of reference are output. This is shown in the output for the first grid point (ID = 2) in the sample output. Here it is seen that values for the forces of single point constraint are provided for the point in the basic and CID 100 systems. This occurs because the output for the point has been specified in CID=100, but the max/min survey was requested in the basic coordinate system. See the MAXMIM Case Control command for further details.

O	*** T1 ***			A P P L I E D L O A D		M A X / M I N		V A L U E		S U M M A R Y		RESULTS FOR SUBCASE		1000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1														
POINT ID.		TYPE	CID	*** T1 ***		T2		T3		R1		R2		R3
2	G	BASIC		0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
2	G		100	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
51	G	BASIC		0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
59	G	BASIC		0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00

[illegible][illegible]

```
1  EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND          AUGUST 22, 2000  MSC.NASTRAN  8/21/00  PAGE    22
  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0      *** T2 ***  A P P L I E D  L O A D  M A X / M I N  V A L U E  S U M M A R Y          RESULTS FOR SUBCASE    2000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2
POINT ID. TYPE  CID          *** T1 ***          *** T2 ***          T3          R1          R2          R3
   2   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
   2   G   100          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  51   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  59   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND          AUGUST 22, 2000  MSC.NASTRAN  8/21/00  PAGE    23
  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0      *** T1 ***  A P P L I E D  L O A D  M A X / M I N  V A L U E  S U M M A R Y          RESULTS FOR SUBCOM    3000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1
POINT ID. TYPE  CID          *** T1 ***          T2          T3          R1          R2          R3
   2   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
   2   G   100          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  51   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  59   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND          AUGUST 22, 2000  MSC.NASTRAN  8/21/00  PAGE    24
  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0      *** T2 ***  A P P L I E D  L O A D  M A X / M I N  V A L U E  S U M M A R Y          RESULTS FOR SUBCOM    3000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2
POINT ID. TYPE  CID          *** T1 ***          *** T2 ***          T3          R1          R2          R3
   2   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
   2   G   100          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  51   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  59   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND          AUGUST 22, 2000  MSC.NASTRAN  8/21/00  PAGE    25
  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0      *** T1 ***  S P C F O R C E  M A X / M I N  V A L U E  S U M M A R Y          RESULTS FOR SUBCASE    1000
MAXMIN OPTIONS: SET=ALL, CID=BASIC, MAX=15, MIN=15, COMP=T1
POINT ID. TYPE  CID          *** T1 ***          T2          T3          R1          R2          R3
   3   G   BASIC          -1.118073E+03          0.000000E+00          0.000000E+00          5.236134E+01          2.360384E-16          1.280828E+02
   3   G   100          0.000000E+00          0.000000E+00          -1.118073E+03          1.280828E+02          -2.360384E-16          5.236134E+01
  101  G   BASIC          -1.000000E+03          -1.733791E-11          0.000000E+00          0.000000E+00          0.000000E+00          8.325034E-02
  251  G   BASIC          -1.000000E+03          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  201  G   BASIC          -1.000000E+03          -1.731096E-11          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  151  G   BASIC          -9.999980E+02          0.000000E+00          -6.541988E-14          0.000000E+00          0.000000E+00          -2.454053E+02
   1   G   BASIC          -4.012542E+02          1.588410E-11          1.125718E-15          -5.499307E+00          -2.439500E-01          7.725024E+01
   51  G   BASIC          -3.474817E+02          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -4.013108E+01
   2   G   BASIC          -1.331908E+02          0.000000E+00          0.000000E+00          -4.686203E+01          2.439500E-01          2.724495E+02
   2   G   100          0.000000E+00          0.000000E+00          -1.331908E+02          2.724495E+02          -2.439500E-01          -4.686203E+01
  209  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  208  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  207  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  206  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  205  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  204  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
  203  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
   55  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -2.368870E-02
   56  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -7.088747E-04
   57  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -4.525901E-04
   58  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -1.690151E-04
   59  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          1.056287E-04
   60  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          8.333347E+01
   6   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          1.338836E-03
  102  G   BASIC          0.000000E+00          0.000000E+00          1.209136E-09          0.000000E+00          0.000000E+00          2.414764E+02
  103  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -3.667573E+00
  104  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -1.456001E+00
  105  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          -8.364819E+01
   5   G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          1.390992E-01
  152  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          6.736528E+00
  153  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          1.944593E+00
  700  G   BASIC          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00          0.000000E+00
```

78 S 0.000000E+00
88 S 0.000000E+00
1 EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND AUGUST 22, 2000 MSC.NASTRAN 8/21/00 PAGE 26
ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0 COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0 *** T1 *** S P C F O R C E M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 2000
MAXMIN OPTIONS: SET=ALL, CID=BASIC, VMAG=4, VMAG=4, COMP=T1
POINT ID. TYPE CID ***T1*** T2 T3 R1 R2 R3
700 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
257 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
202 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
203 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
251 G BASIC 1.000000E+03 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
201 G BASIC 1.000000E+03 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
101 G BASIC 1.000002E+03 0.000000E+00 0.000000E+00 8.325034E-02 0.000000E+00 0.000000E+00
3 G BASIC 1.118073E+03 0.000000E+00 0.000000E+00 1.383724E+02 0.000000E+00 0.000000E+00
3 G 100 0.000000E+00 0.000000E+00 -1.118073E+03 1.280828E+02 -2.360384E-16 5.236134E+01

78 S 0.000000E+00
88 S 0.000000E+00

1 EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND AUGUST 22, 2000 MSC.NASTRAN 8/21/00 PAGE 27
ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0 COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0 *** R1 *** S P C F O R C E M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 2000
MAXMIN OPTIONS: SET=ALL, CID=BASIC, VMAG=4, VMAG=4, COMP=R1
POINT ID. TYPE CID T1 T2 T3 ***RMAG*** R2 R3
700 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
256 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
210 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
209 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
52 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 2.110957E+02 0.000000E+00 0.000000E+00
102 G BASIC 1.209136E-09 0.000000E+00 0.000000E+00 2.414764E+02 0.000000E+00 0.000000E+00
151 G BASIC 9.999980E+02 0.000000E+00 0.000000E+00 2.454053E+02 0.000000E+00 0.000000E+00
2 G BASIC 1.331908E+02 0.000000E+00 0.000000E+00 2.764504E+02 0.000000E+00 0.000000E+00
2 G 100 0.000000E+00 0.000000E+00 -1.331908E+02 2.724495E+02 -2.439500E-01 -4.686203E+01

1 EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND AUGUST 22, 2000 MSC.NASTRAN 8/21/00 PAGE 28
ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0 COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0 *** T1 *** S P C F O R C E M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCOM 3000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1
POINT ID. TYPE CID *** T1 *** T2 T3 R1 R2 R3
51 G BASIC -3.474817E+02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 -4.013108E+01
2 G BASIC -1.331908E+02 0.000000E+00 0.000000E+00 -4.686203E+01 2.439500E-01 2.724495E+02
2 G 100 0.000000E+00 0.000000E+00 -1.331908E+02 2.724495E+02 -2.439500E-01 -4.686203E+01
59 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 1.056287E-04

1 EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND AUGUST 22, 2000 MSC.NASTRAN 8/21/00 PAGE 29
ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0 COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0 *** T2 *** S P C F O R C E M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCOM 3000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2
POINT ID. TYPE CID T1 *** T2 *** T3 R1 R2 R3
51 G BASIC -3.474817E+02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 -4.013108E+01
2 G BASIC -1.331908E+02 0.000000E+00 0.000000E+00 -4.686203E+01 2.439500E-01 2.724495E+02
2 G 100 0.000000E+00 0.000000E+00 -1.331908E+02 2.724495E+02 -2.439500E-01 -4.686203E+01
59 G BASIC 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 1.056287E-04

1 EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND AUGUST 22, 2000 MSC.NASTRAN 8/21/00 PAGE 30
ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0 COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0 *** T1 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 1000
MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1
POINT ID. TYPE CID *** T1 *** T2 T3 R1 R2 R3
2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00
2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00

1	EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND	AUGUST 22, 2000	MSC.NASTRAN	8/21/00	PAGE	31
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED					
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE					
0	*** T2 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 1000					
	MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2					
	POINT ID. TYPE CID T1 *** T2 *** T3 R1 R2 R3					
	59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
1	EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND	AUGUST 22, 2000	MSC.NASTRAN	8/21/00	PAGE	32
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED					
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE					
0	*** T1 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 2000					
	MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1					
	POINT ID. TYPE CID *** T1 *** T2 T3 R1 R2 R3					
	2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
1	EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND	AUGUST 22, 2000	MSC.NASTRAN	8/21/00	PAGE	33
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED					
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE					
0	*** T2 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCASE 2000					
	MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2					
	POINT ID. TYPE CID T1 *** T2 *** T3 R1 R2 R3					
	59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
1	EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND	AUGUST 22, 2000	MSC.NASTRAN	8/21/00	PAGE	34
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED					
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE					
0	*** T1 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCOM 3000					
	MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1					
	POINT ID. TYPE CID *** T1 *** T2 T3 R1 R2 R3					
	2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
1	EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND	AUGUST 22, 2000	MSC.NASTRAN	8/21/00	PAGE	35
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED					
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE					
0	*** T2 *** D I S P L A C E M E N T M A X / M I N V A L U E S U M M A R Y RESULTS FOR SUBCOM 3000					
	MAXMIN OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2					
	POINT ID. TYPE CID T1 *** T2 *** T3 R1 R2 R3					
	59 G BASIC 1.939060E-02 -2.595003E-02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G BASIC 0.000000E+00 -3.568517E-05 -2.158988E-07 0.000000E+00 0.000000E+00 0.000000E+00					
	2 G 100 -2.158988E-07 3.568517E-05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					
	51 G BASIC 0.000000E+00 1.852034E-04 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00					

Unconstrained Equilibrium Check

If a structure is truly unconstrained, then movement at one point of this structure should cause the whole structure to move as a rigid body. The following procedure can be used to perform this check.

1. Remove all the constraints.
2. Apply a unit enforced displacement in the x-direction at one selected grid point while constraining the other five components at this selected grid point to zero. This grid point should be close to the center of gravity (C.G.) of the structure, although this is not a requirement. The x-displacements at all the other grid points should also be equal to unity if the structure is truly unconstrained. If the

displacement at a certain grid point is not equal to unity in the x-direction, it is very likely that it is overstrained. The likely causes for this can include incorrect modeling of rigid elements, offset beams, non-colinear CELASi, etc. See “**Constraints**” on page 257 for details regarding enforced displacement.

3. Repeat the same procedure for the other two orthogonal directions. You can perform similar checks for rotations; however, it is more difficult to interpret the results.

At the conclusion of this check, you should remember to put your original constraints back into your model.

Automatic Identification of Unintentional Grounding

In addition to the unconstrained equilibrium check mentioned above, you may also request a more robust and more automatic grounding check of the stiffness matrix. This grounding check is also referred to as rigid body check because the stiffness matrix is multiplied by the rigid body transformation matrix. The grounding check identifies unintentional constraints and ill-conditioning in the stiffness matrix. The grounding check is very similar to--and is intended as a replacement for--the grounding check performed in the checka.vxx alter found in the SSSAlter library prior to MSC.Nastran 2001.

The grounding check is requested with the GROUNDCHECK Case Control command.

$$\text{GROUNDCHECK} \left[\left(\left[\begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], \text{PUNCH}, \text{SET} = \left\{ \begin{array}{c} \text{G}, \text{N}, \text{N} + \text{AUTOSPC}, \text{F}, \text{A} \\ \text{ALL} \end{array} \right\} \right) \right] = \left\{ \begin{array}{c} \text{Y} \\ \text{N} \end{array} \right.$$

$$\left[\text{GRID} = \text{gid}, \text{THRESH} = \text{e}, \text{DATA REC} = \left[\begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right], (\text{RTHRESH} = \text{r}) \right)$$

For example, the following command

GROUNDCHECK=YES

will request a grounding check of the g-set. The following command

GROUNDCHECK(SET=ALL)=YES

will request a grounding check of all DOF sets.

The check may be performed at any or all stages of the stiffness reduction based on the specification of the SET keyword

SET Keyword	DOF Set	Description
G	g-set	before single point, multipoint constraints, and rigid elements are applied
N	n-set	after multipoint constraints and rigid elements are applied
N+AUTO	n-set with AUTOSPC	same as the n-set with the rows/columns in the stiffness matrix corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out
F	f-set	after single point, multipoint constraints, and rigid elements are applied
A	a-set	after static condensation

The THRESH keyword specifies the maximum strain energy that passes the grounding check. DATAREC=YES requests data recovery of grounding forces. The RTHRESH=r keyword prints the grounding forces larger than r percent of the largest grounding force if DATAREC is set to YES.

User Information Message 7570 is issued by GROUNDCHECK for each DOF set requested. The strain energy is computed in each direction of the rigid body motion and by default, if the strain energy exceeds the tolerance, then “FAIL” is printed out for that directory. The tolerance is set by dividing the largest stiffness term by 1.E10. The THRESH keyword can be used to further control the amount of output (see Appendix A). Possible reasons for failure are also printed after the strain energies. Here is an example for the g-set that indicates the stiffness matrix passes the grounding check in all six rigid body directions:

```

RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET) FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 2.725275E-04
DIRECTION      STRAIN ENERGY      PASS/FAIL
-----
1              1.484295E-09          PASS
2              2.182787E-10          PASS
3              1.637090E-11          PASS
4              1.619810E-10          PASS
5              2.727802E-10          PASS
6              1.054841E-07          PASS
SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;
4. IMPROPERLY DEFINED DMIG MATRICES;

```

Here is an example for the f-set that indicates the stiffness matrix fails the grounding check in all six rigid body directions:

```

*** USER INFORMATION MESSAGE 7570 (GPWG1D)
RESULTS OF RIGID BODY CHECKS OF MATRIX KFF      (F-SET) FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 1.464858E-04
DIRECTION      STRAIN ENERGY      PASS/FAIL
-----
1              2.564102E+05          FAIL
2              7.326008E+05          FAIL
3              2.237437E+03          FAIL
4              6.057062E+02          FAIL
5              4.015165E+03          FAIL
6              8.747863E+04          FAIL
SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
1. CONSTRAINTS WHICH PREVENT RIGID-BODY MOTION.

```

If the DATAREC keyword is specified and the rigid body check fails then data recovery will be performed to compute and print the grounding forces to make it easy to locate the source of the failure. By default, only those grounding forces larger than 10 percent of the largest grounding force will be printed (see RTHRESH keyword). Here is an example of grounding forces created by moving the model in rigid body direction 1 associated with the f-set failure shown above:

DIRECTION		GROUND CHECK FORCES (F - SET)							
1									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
3	G	2.564102E+05	-2.884615E+05	.0	.0	.0	.0		
4	G	2.564102E+05	2.884615E+05	.0	.0	.0	.0		

Thermal Equilibrium Check

The following procedure can also be used to check for unconstrained thermal expansion if the structure is going to be subjected to thermal loads.

1. Remove the actual boundary conditions and apply a set of statically determinate constraints. Typically this procedure is done by constraining all six DOFs at a single grid point. If a single grid point is used, make sure this single grid point contains six degrees of freedom. A single grid point of a model consisting of all solid elements, for example, cannot satisfy this requirement since each grid point of a solid element contains only three degrees of freedom.
2. Change all the thermal coefficients of expansion to a single value.
3. Apply a uniform ΔT to the structure.

If the model is “clean,” then the structure should be strain free; in other words, there should be no reaction loads, element forces, or stresses. If this is not the case, then you may want to investigate around the vicinity where the element forces or stresses are nonzero. Incorrect modeling of rigid elements or offsets is a common cause of these types of errors. Once you are satisfied with your model, remember to change the boundary condition, thermal coefficients of expansion, and ΔT back to their original values.

Grid Point Force Balance

Frequently, there are several elements connected to a single grid point. When this happens, you may want to know how the loads are distributed among these elements at the common grid point. In other words, you want to understand the load path. Is the load path correct? These questions can be answered by using the GPFORCE (grid point force) option in MSC.Nastran.

Grid point force balance is computed only from linear stiffness elements, the sum of applied loads and thermal loads, and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), rigid elements and MPCs, general elements, DMIG entries, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. [Table 10-2](#) summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system.

Table 10-2 Contributions Used for the Grid Point Force Balance

Contributions Included	Contributions Ignored
Applied Loads	Boundary Loads from Upstream Superelements
SPC Forces	GENEL Forces
Element Elastic Forces	DMIG and DMI Forces
Thermal Loads	

The model in [Figure 10-22](#) illustrates this feature. This model consists of nine CQUAD4 and six CBAR elements and is subjected to vertical loads applied at the free end. The input file is shown in [Listing 10-6](#).

EXAMPLE

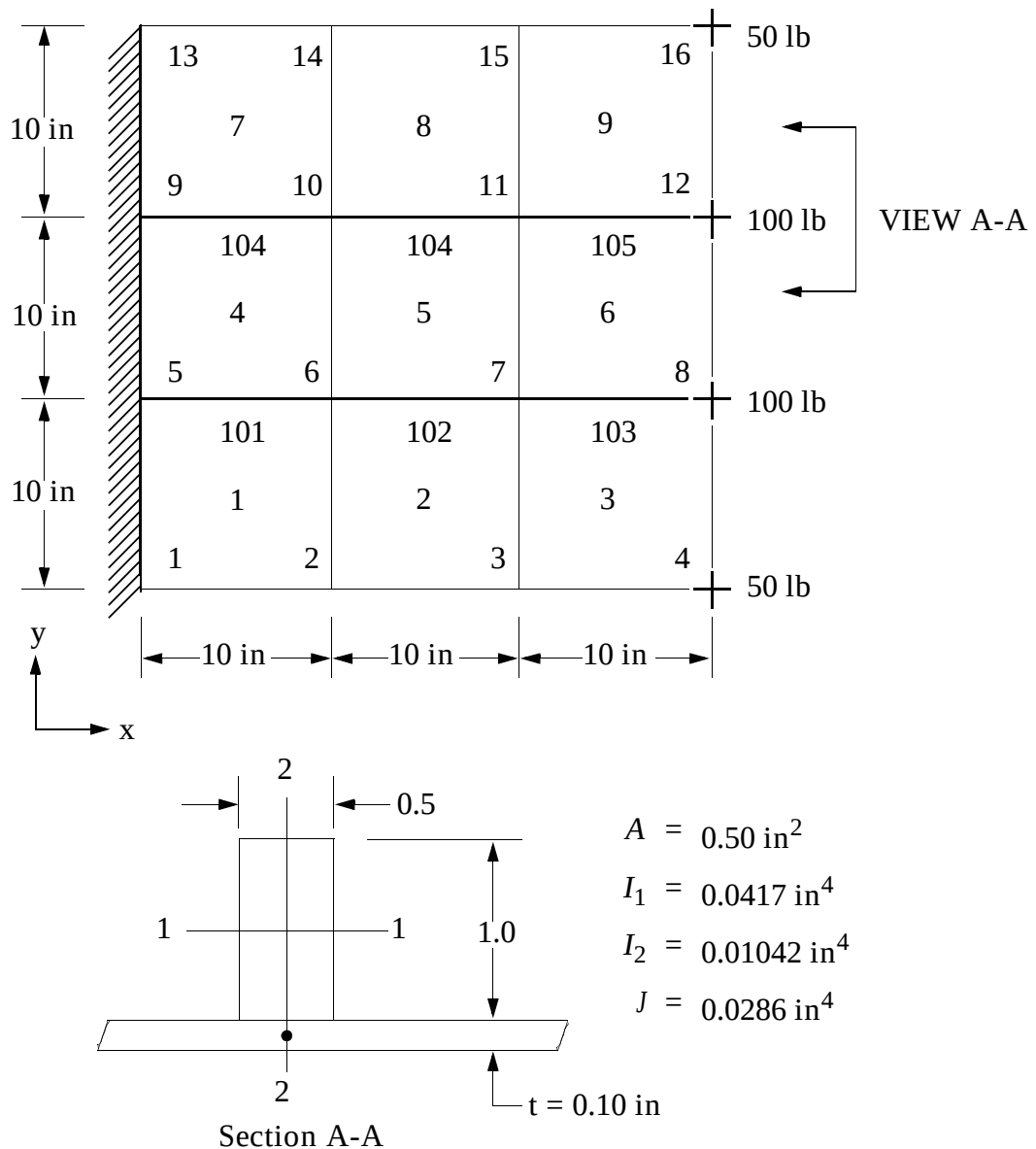


Figure 10-22 Clamped Plate with Stiffeners

Listing 10-6 Input File for the Demonstration of Grid Point Force Balance

```

$  FILENAME - q4bargpf.dat
$
SOL      101
TIME     5
CEND
TITLE = ILLUSTRATE USE OF GPFORCE
SUBTITLE =  INCORRECT PROPERTIES
DISP = ALL
GPFORCE = ALL
STRESS(CORNER) = ALL
SPC = 1
LOAD = 1
PARAM    AUTOSPC YES
PARAM    POST      0
$
$ THIS SECTION CONTAINS BULK DATA
$
GRID      1           0.0      0.0      0.0
GRID      2           10.      0.0      0.0
GRID      3           20.      0.0      0.0
GRID      4           30.      0.0      0.0
GRID      5           0.0      10.      0.0
GRID      6           10.      10.      0.0
GRID      7           20.      10.      0.0
GRID      8           30.      10.      0.0
GRID      9           0.0      20.      0.0
GRID     10           10.      20.      0.0
GRID     11           20.      20.      0.0
GRID     12           30.      20.      0.0
GRID     13           0.0      30.      0.0
GRID     14           10.      30.      0.0
GRID     15           20.      30.      0.0
GRID     16           30.      30.      0.0
$
CBAR     101      100      5           6           0.0      0.0      1.
+C101           0.      0.      .55           0.      0.      .55
CBAR     102      100      6           7           0.0      0.0      1.
+C102           0.      0.      .55           0.      0.      .55
CBAR     103      100      7           8           0.0      0.0
+C103           0.      0.      .55           0.      0.      .55
CBAR     104      200      9          10           0.0      0.0
+C104           0.      0.      .55           0.      0.      .55
CBAR     105      200     10          11           0.0      0.0
+C105           0.      0.      .55           0.      0.      .55
CBAR     106      200     11          12           0.0      0.0
+C106           0.      0.      .55           0.      0.      .55
$
CQUAD4   1         1         1         2         6         5
CQUAD4   2         1         2         3         7         6
CQUAD4   3         1         3         4         8         7
CQUAD4   4         1         5         6         10        9
CQUAD4   5         1         6         7         11        10
CQUAD4   6         1         7         8         12        11
CQUAD4   7         1         9         10        14        13
CQUAD4   8         1        10        11        15        14
CQUAD4   9         1        11        12        16        15
$

```

Listing 10-6 Input File for the Demonstration of Grid Point Force Balance (continued)

```

$ THIS SECTION CONTAINS THE LOADS and CONSTRAINTS
$
$
FORCE  1      4      0      50.      0.      0.      -1.
FORCE  1      8      0      100.     0.      0.      -1.
FORCE  1     12      0      100.     0.      0.      -1.
FORCE  1     16      0      50.      0.      0.      -1.
$
SPC1   1      123456  1      5      9      13
$
$ THIS SECTION CONTAINS THE PROPERTY AND MATERIAL BULK DATA ENTRIES
$
PSHELL 1      1      .1      1
$
PBAR   100     1      .50      .0417   .01042   .0286
+P100  .5     .25   -.5     .25     -.5     -.25   .5     -.25
PBAR   200     1      .50      .0417   .01042   .0286
+P200  .5     .25   -.5     .25     -.5     -.25   .5     -.25
$
$  CORRECT PBAR ENTRY WITH A AND I1
$
$PBAR   200     1      .50      .0417   .01042   .0286
$
$
MAT1    1      1.+7      .3
ENDDATA

```

If everything is done correctly, expect the stress contour plot to look something like [Figure 10-23](#) (a). However, say that you made a modeling mistake somewhere, and the stress contour plot is coming out as shown in [Figure 10-23](#)(b). You know that something is wrong because the stress contour plot is not symmetrical. Since the structure, boundary conditions, and applied loads are all symmetrical, you also expect the stress contour plot to be symmetrical. By inspecting the grid point force balance output ([Figure 10-24](#)), you notice that the stiffeners (CBAR 104, 105, and 106) are not picking up any axial, vertical, or out-of-plane bending loads at grid points 9, 10, 11, and 12.

A review of the input file explains the reason for this result. The area (A) and the out-of-plane bending moment of inertia (I1) were "accidentally" left out for CBAR elements 104, 105, and 106 (PBAR 200). Hence, all the axial and out-of-plane bending loads at grid points 9, 10, 11, and 12 were picked up completely by the surrounding CQUAD4 elements. By correcting the PBAR element properties, note that CBAR elements 104, 105, and 106 now pick up the correct loads as shown in [Figure 10-25](#).

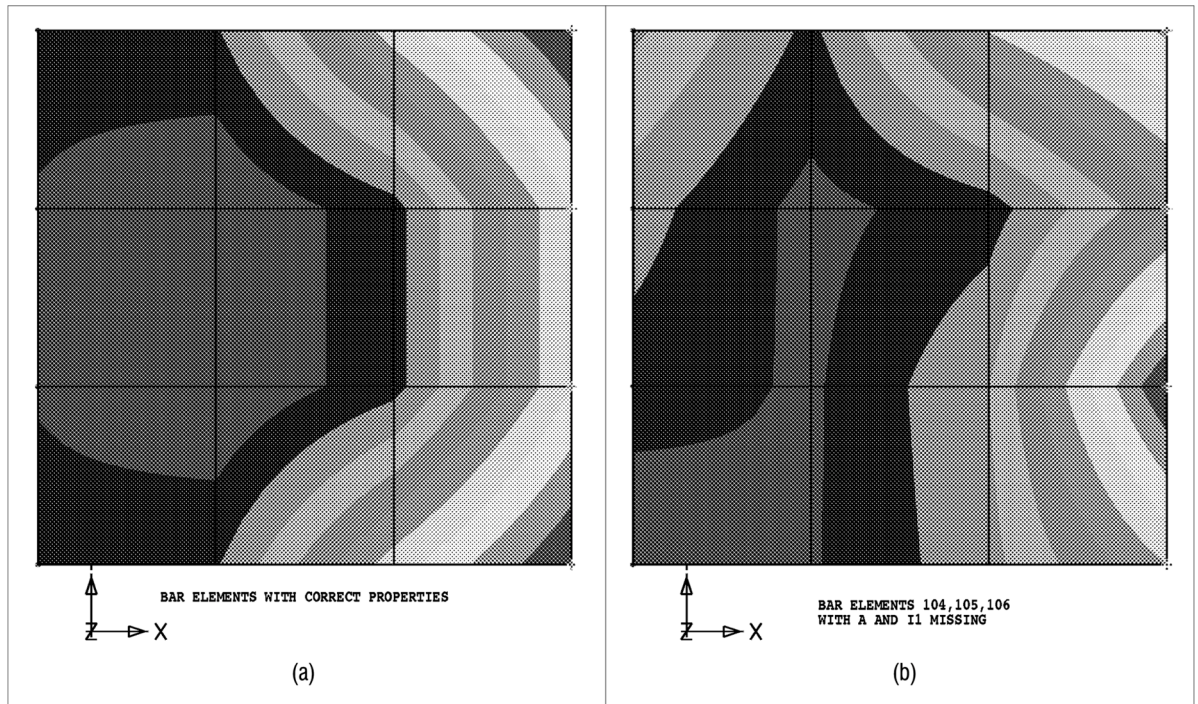


Figure 10-23 Stress Contour Plots of a Cantilever Plate

POINT-ID	ELEMENT-ID	SOURCE	GRID POINT FORCE			BALANCE		
			T1	T2	T3	R1	R2	R3
1	1	F-OF-SPC	2.974426E+03	9.828222E+02	1.589665E+01	1.144652E+02	1.100169E+01	.0
1		QUAD4	-2.974426E+03	-9.828222E+02	-1.589665E+01	-1.144652E+02	-1.100169E+01	.0
1		*TOTALS*	4.547474E-13	.0	-8.881784E-15	-4.263256E-14	-2.486900E-14	.0
9	104	F-OF-SPC	2.149695E+03	-7.652444E+02	4.273880E+01	5.070187E+02	-2.392951E+02	-1.157442E+02
9		BAR	.0	1.266284E+01	.0	-7.117958E+02	.0	1.157442E+02
9		QUAD4	-1.788234E+03	6.868671E+02	-5.524031E+01	1.670983E+02	1.526720E+02	.0
9	7	QUAD4	-3.614603E+02	6.571449E+01	1.250151E+01	3.767887E+01	8.662309E+01	.0
9		*TOTALS*	1.136868E-13	1.705303E-13	-1.136868E-13	-1.350031E-13	4.263256E-13	.0
10		BAR	.0	-1.266284E+01	.0	7.117958E+02	.0	1.088424E+01
10	105	BAR	.0	2.289362E-01	.0	-9.452994E+02	.0	-1.088424E+01
10	4	QUAD4	1.194813E+03	-1.018952E+02	6.047494E+01	-1.471362E+02	4.924568E+01	.0
10		QUAD4	-1.501499E+03	3.827450E+02	-6.910983E+01	3.978395E+02	-4.706585E+01	.0
10		QUAD4	7.029081E+02	-2.641614E+02	-1.464156E+01	-7.215542E+01	-5.501759E+01	.0
10	8	QUAD4	-3.962218E+02	-4.254534E+00	2.327645E+01	5.495581E+01	5.283775E+01	.0
10		*TOTALS*	8.526513E-13	-8.970602E-14	4.476419E-13	-1.627143E-12	-1.946887E-12	-5.329071E-15
11		BAR	.0	-2.289362E-01	.0	9.452994E+02	.0	1.317360E+01
11	106	BAR	.0	-1.317360E+00	.0	-7.749208E+02	.0	-1.317360E+01
11	5	QUAD4	5.103351E+02	-5.226174E+02	9.304677E+01	-4.448763E+02	-8.715293E+01	.0
11		QUAD4	-8.479946E+02	6.063411E+02	-8.722735E+01	4.217841E+02	3.954052E+01	.0
11		QUAD4	4.692538E+02	-8.115437E+01	-3.112581E+01	-1.103274E+02	2.823395E-01	.0
11	9	QUAD4	-1.315942E+02	-1.023029E+00	2.530639E+01	-3.695915E+01	4.733007E+01	.0
11		*TOTALS*	-3.439027E-12	2.285061E-12	-2.501110E-12	5.400125E-12	-7.815970E-13	6.572520E-14
12		APP-LOAD	.0	.0	-1.000000E+02	.0	.0	.0
12	106	BAR	.0	1.317360E+00	.0	7.749208E+02	.0	-4.178994E-14
12	6	QUAD4	-1.224487E+02	-1.046283E+01	1.404051E+02	-5.812056E+02	3.421728E+00	.0
12		QUAD4	1.224487E+02	9.145473E+00	-4.040514E+01	-1.937151E+02	-3.421728E+00	.0
12		*TOTALS*	7.389644E-13	-6.270540E-13	-1.371347E-12	1.023182E-12	-4.396483E-12	-4.178994E-14
16	9	APP-LOAD	.0	.0	-5.000000E+01	.0	.0	.0
16		QUAD4	-4.145872E-14	-8.016339E-13	5.000000E+01	-3.638509E-12	2.948251E-12	.0
16		*TOTALS*	-4.145872E-14	-8.016339E-13	1.691092E-12	-3.638509E-12	2.948251E-12	.0

Figure 10-24 Grid Point Force Balance Output for a Model with Incorrect CBAR Properties

GRID POINT FORCE BALANCE								
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	1.695155E+03	6.360602E+02	1.951741E+01	8.563783E+01	-2.406043E+01	.0
1	1	QUAD4	-1.695155E+03	-6.360602E+02	-1.951741E+01	-8.563783E+01	2.406043E+01	.0
1		*TOTALS*	.0	.0	3.552714E-15	1.421085E-14	-3.552714E-15	.0
9		F-OF-SPC	-1.695155E+03	-4.214514E+02	1.304826E+02	1.775948E+02	-4.475939E+03	-4.778485E+01
9	104	BAR	4.887901E+03	5.637726E+00	-1.451854E+02	-2.707557E+02	4.427706E+03	4.778485E+01
9	4	QUAD4	-1.885757E+03	3.980623E+02	-1.594168E+00	1.076575E+01	2.244092E+01	.0
9	7	QUAD4	-1.306990E+03	1.775138E+01	1.629696E+01	8.239518E+01	2.579244E+01	.0
9		*TOTALS*	-6.821210E-13	7.815970E-14	-1.101341E-13	5.684342E-14	-2.984279E-13	.0
10	104	BAR	-4.887901E+03	-5.637726E+00	1.451854E+02	2.707557E+02	-2.975852E+03	8.592416E+00
10	105	BAR	2.834930E+03	-2.358537E-02	-1.454878E+02	-2.992984E+02	2.890022E+03	-8.592416E+00
10	4	QUAD4	1.885757E+03	2.042302E+02	1.594168E+00	1.819319E+01	-6.499233E+00	.0
10	5	QUAD4	-1.123152E+03	-1.147349E+02	-1.597489E+00	3.257575E+01	2.503494E+01	.0
10	7	QUAD4	1.960801E+03	-4.728610E+02	-1.916611E+01	-9.600525E+01	-2.932959E+00	.0
10	8	QUAD4	-6.704347E+02	3.890270E+02	1.947184E+01	7.377895E+01	7.022765E+01	.0
10		*TOTALS*	-1.023182E-12	3.410605E-13	-3.623768E-13	-2.842171E-14	-4.263256E-13	1.598721E-14
11	105	BAR	-2.834930E+03	2.358537E-02	1.454878E+02	2.992984E+02	-1.435144E+03	8.356563E+00
11	106	BAR	8.836555E+02	-8.356562E-01	-1.431583E+02	-1.784329E+02	1.377845E+03	-8.356563E+00
11	5	QUAD4	1.123152E+03	-2.516147E+01	1.597489E+00	4.088165E+01	-9.060055E+00	.0
11	6	QUAD4	-3.665653E+02	1.245855E+02	-1.571911E+00	5.274417E+01	2.618646E+01	.0
11	8	QUAD4	1.240412E+03	-2.789030E+02	-2.937232E+01	-1.661472E+02	1.650724E+01	.0
11	9	QUAD4	-4.572457E+01	1.802911E+02	2.701724E+01	-4.834404E+01	2.366565E+01	.0
11		*TOTALS*	-6.849632E-12	-3.126388E-13	-1.712408E-12	2.771117E-13	3.197442E-12	5.151435E-14
12		APP-LOAD	.0	.0	-1.000000E+02	.0	.0	.0
12	106	BAR	-8.836555E+02	8.356562E-01	1.431583E+02	1.784329E+02	5.373812E+01	-3.213094E-14
12	6	QUAD4	3.665653E+02	4.705300E+02	1.571911E+00	5.770844E+01	-1.046736E+01	.0
12	9	QUAD4	5.170903E+02	-4.713657E+02	-4.473020E+01	-2.361413E+02	-4.327076E+01	.0
12		*TOTALS*	3.069545E-12	-8.526513E-13	-4.902745E-13	-8.810730E-13	1.975309E-12	-3.213094E-14
16		APP-LOAD	.0	.0	-5.000000E+01	.0	.0	.0
16	9	QUAD4	2.966625E-13	3.832166E-14	5.000000E+01	-1.397169E-12	1.610404E-12	.0
16		*TOTALS*	2.966625E-13	3.832166E-14	8.171241E-13	-1.397169E-12	1.610404E-12	.0

Figure 10-25 Grid Point Force Balance Output for a Model with Correct CBAR Properties

Both [Figure 10-24](#) and [Figure 10-25](#) are reduced versions of the full output. Only the grid point force balance output as related to grid points 1, 9, 10, 11, 12, and 16 are shown in these two figures for clarity. Notice that for grid points 1 and 9, in addition to the elastic element forces, the SPC forces are also printed. Similarly, for grid points 12 and 16, the applied loads are also printed. As you can see, the grid point force balance feature can provide you with a tremendous insight regarding the load path of your structure.

10.5 Stress Error Estimators

“**Grid Point Stresses**” on page 349 is devoted to a discussion of the averaging procedures utilized to provide meaningful stresses at the grid points of finite element models in MSC.Nastran. However, when you look at a stress distribution, frequently you may not be sure of whether further model refinement is necessary. This section focuses upon the description of error estimators that were first introduced in MSC.Nastran Version 67 to assist you in the identification of regions of your model that may require refinement.

The essence of the averaging procedures used to obtain grid point stress data are to:

1. Convert these local element stress components into a common coordinate system defined by you.
2. Average several values of each stress component to obtain a unique value of the stress component that is to be associated with the grid point in question.
3. Compute the stress invariants at the grid points from the stress components at the grid points.

In the general case, the stress components are σ_x , σ_y , σ_z , τ_{xy} , τ_{xz} , and τ_{yz} .

For discussion purposes, the averaging process used to compute the stress components at the grid points can be represented in the form

$$\sigma_g = \sum_{i=1}^{N_e} (W_i \sigma_{ei}) \quad \text{Eq. 10-3}$$

where:

σ_g = the weighted mean value of the stress component computed at the grid point.

σ_{ei} = the value of the stress component in the i -th element $i = 1, 2, \dots, N_e$ connected to the grid point. σ_{ei} is in the same coordinate system as σ_g .

W_i = the weighting factor assigned to the i -th element. The sum of the values of must equal 1. This requirement assures that all computed statistics are unbiased. The attribute of being unbiased implies that the variance is equal to the mean square error. Equal weighting, i.e., $W_i = 1/N_e$, is assumed in MSC.Nastran.

An estimate of the error in a particular component of stress at a grid point can then be computed by assuming that the values of the corresponding stress components computed by MSC.Nastran for the elements in the neighborhood of the grid point are data points with uncorrelated random errors. It then follows that an estimate of the probable error in the stress component δ_g at the grid point is

$$\delta_g = \sqrt{\sum_{i=1}^{N_e} (W_i \delta_{ei})^2} = \frac{1}{\sqrt{N_e}} \sqrt{\sum_{i=1}^{N_e} (\delta_{ei})^2} \quad \text{Eq. 10-4}$$

where $\delta_{ei} = \sigma_{ei} - \sigma_g$. Thus, the probable error δ_g is the root mean square error in δ_{ei} divided by

$$\sqrt{N_e}$$

It should be noted that the root mean square error is a reasonable measure of precision in many practical cases, but it is easy to provide examples in which this value is a poor measure of the concentration of the distribution about the mean. This subject is discussed further in the section "Discussion of Error Measures." [Eq. 10-4](#) is assumed to provide an approximate error estimator for the grid point stress data output by MSC.Nastran.

This stress error assessment is often referred to as stress discontinuity, and it is evaluated in two different ways in MSC.Nastran. The stress discontinuity is evaluated as grid point and element stress discontinuities.

Grid Point Stress Discontinuities

To illustrate the computations described in the previous section, consider the following portion of a finite element model consisting of CQUAD4 elements. For clarity, only a portion of the model is plotted. These elements lie on a surface (defined as surface 91 in this case).

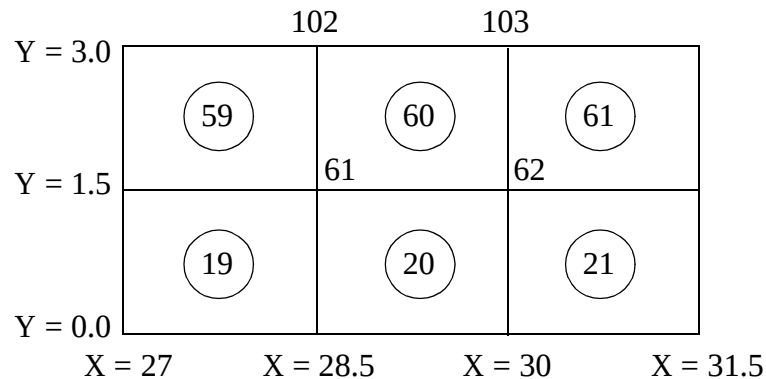


Figure 10-26 Partial Model for the Stress Discontinuity Calculations

The following data is extracted from the MSC.Nastran output. [Table 10-3](#) and [Table 10-4](#) contain partial listings of the element stresses and grid point stresses for the above model. The grid point stresses are obtained following the procedures described in "[Grid Point Stresses](#)" on page 349.

Table 10-3 Element Stresses for CQUAD4

Element ID	Stresses in the Element Coordinate System		
	NORMAL-X	NORMAL-Y	SHEAR-XY
19	-4.006485E+03	-8.949428E-09	-1.045506E+02
20	-3.820136E+03	2.255547E-09	-1.045506E+02
.			
.			
.			
59	-1.335495E+03	8.658390E-09	-2.287827E+02
60	-1.273379E+03	-1.818989E-09	-2.287827E+02

Table 10-4 Grid Point Stresses for Surface 91

Grid ID	Stresses in the Element System		
	NORMAL-X	NORMAL-Y	SHEAR-XY
20	-5.218E+03	-6.730E-09	-4.243E+01
21	-4.969E+03	1.692E-09	-4.243E+01
.			
.			
.			
60	-2.733E+03	-3.274E-10	-1.667E+02
61	-2.609E+03	-3.638E-11	-1.667E+02
62	-2.485E+03	1.273E-10	-1.667E+02
63	-2.360E+03	-5.457E-11	-1.667E+02
.			
.			
.			
102	-6.104E-05	6.803E-09	-2.909E+02
103	-9.155E-05	-1.437E-09	-2.909E+02

The probable error for stresses at both grid points 20 and 61 can be computed using [Eq. 10-4](#). The probable error δ_g is expected to provide some measure of the error in the magnitude of the grid point stress values. You must first compute the values $\delta_{ei} = \sigma_{ei} - \sigma_g$ for each stress component and then compute the probable error, δ_g . This calculation is done for

both grid points 20 and 61. The symbols “GID” and “EID”, which represent grid point ID and element ID, respectively, are used extensively throughout this section. Consider an example calculation using the NORMAL-X stress component.

NORMAL-X Stress

GID = 20			
EID	σ_{ei}	σ_g	δ_{ei}
19	-4.006485E+03	-5.218E+03	1.212E+03
20	-3.820136E+03	-5.218E+03	1.398E+03

Using [Eq. 10-4](#),

$$\delta_g = \frac{1}{\sqrt{2}} \sqrt{\frac{(1212)^2 + (1398)^2}{2}} = 924.9$$

GID = 61			
EID	σ_{ei}	σ_g	δ_{ei}
19	-4.006485E+03	-2.609E+03	-1.397E+03
20	-3.820136E+03	-2.609E+03	-1.211E+03
59	-1.335495E+03	-2.609E+03	1.274E+03
60	-1.273379E+03	-2.609E+03	1.336E+03

Again, using [Eq. 10-4](#),

$$\delta_g = \frac{1}{\sqrt{4}} \sqrt{\frac{(-1397)^2 + (-1211)^2 + (1274)^2 + (1336)^2}{4}} = 653.1$$

The other stress components (σ_y and τ_{xy}) can then be obtained in a similar manner.

This same finite element model is then remodeled using different element types-- CQUAD8, CHEXA(8) as well as CQUAD4 elements with the corner output option (see Reference 8.) The values of δ_g for each of the three stress components are then calculated at two locations: $x = 28.5$ and $y = 0.0$, as well as $x = 28.5$ and $y = 1.5$. Note that these two locations correspond to GID 20 and GID 61, respectively, for the above CQUAD4 model. This process is then repeated for each of the above element types. The results (δ_g and ERROR ESTIMATE) from MSC.Nastran are summarized in [Table 10-5](#). Note that these values may be slightly different than the hand-calculated values since the computer provides more precision than the hand calculation.

The probable error δ_g provides an estimated error for each of the three stress components at each of the two grid point locations considered. It is probably more useful and more desirable to combine these three estimated errors into a single representative error measure at each grid point. An approximate root mean square value of the three estimated errors for each of the three stress components offers such a representative error measure. That is,

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{gi})^2}{N_c}} \quad \text{Eq. 10-5}$$

where N_c , which is the number of stress components, is equal to three for plate elements and six for solid elements.

Table 10-5 The Values of δ_g and ERROR ESTIMATE for Different Element Types (X = 28.5, Y = 0.0)

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL-Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	9.247E+02	4.628E-09	4.392E+01	5.345E+02
CQUAD8	0.000E+00	1.204E-09	0.000E+00	9.840E-10
CHEXA (8)	8.839E+01	2.841E+01	0.000E+00	5.729E+01
CQUAD4 (CORNER)	9.653E+01	2.896E+01	3.548E+02	2.129E+02

Table 10-6 The Values of δ_g and ERROR ESTIMATE for Different Element Types (X = 28.5, Y = 1.5)

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL-Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	6.531E+02	3.196E-09	3.106E+01	3.775E+02
CQUAD8	0.000E+00	3.634E-09	0.000E+00	2.098E-09
CHEXA (8)	3.125E+01	1.497E+01	3.125E+01	2.148E+01
CQUAD4 (CORNER)	7.298E+01	2.153E+02	2.532E+02	1.965E+02

The following remarks should be noted in regard to [Table 10-5](#) and [Table 10-6](#).

1. By default, the stress components for the CQUAD4 elements are only output at the element centroids, while the stress components for the CQUAD8, and CHEXA(8) elements are output at the element centroids as well as the element vertices.

2. If vertex stresses are also desired for the CQUAD4 elements, then use the STRESS(CORNER) = x option in the Case Control Section. This output corresponds to the last rows of [Table 10-5](#) and [Table 10-6](#).
3. The calculation of δ_g for the CQUAD8 and CHEXA(8) elements involves grid point and element vertex stresses that are defined at the same points.
4. The calculation of δ_g for the CQUAD4 elements involve grid point stresses and element stresses provided at a different geometric location if the default center option is requested. They are evaluated at the same location when using the corner option.
5. The ERROR ESTIMATE for the CHEXA(8) also includes the contribution of σ_z , σ_{xz} , and σ_{yz} . These three values are not listed in [Table 10-5](#) and [Table 10-6](#). The procedure is identical to that of the plate element with the exception that it includes six instead of three stress components.

[Table 10-5](#) and [Table 10-6](#) indicate that the smaller values of δ_g and the ERROR ESTIMATE are obtained from the models composed of elements for which vertex stresses are computed by MSC.Nastran. A detailed discussion of the error estimator used by MSC.Nastran is presented later in the section “Discussion of Error Measures.”

In addition to values of δ_g for each stress component and the ERROR ESTIMATE of [Eq. 10-5](#) MSC.Nastran output also contains values of δ_g for each of the stress invariants. These quantities are output for each grid point in the defined SURFACE (plate elements) or VOLUME (solid elements).

Element Stress Discontinuities

Other statistical error estimates may be generated by associating the estimates with the elements rather than with the grid points. For example, root mean square errors for each stress component of an element may be computed from the values of δ_{ei} that are computed for each of the N_g connected vertex grid points where $\delta_{ei} = (\sigma_e - \sigma_{gi})$. This computation is done for each stress component for all of the elements of interest.

Next is an example calculating the NORMAL-X stress components for elements 20 and 60 in [Figure 10-26](#).

NORMAL-X Stress

EID=20			
GID	σ_e	σ_{gi}	δ_{ei}
20	-3.820136E+03	-5.218E+03	1.398E+03
21	-3.820136E+03	-4.969E+03	1.149E+03
62	-3.820136E+03	-2.485E+03	-1.335E+03
61	-3.820136E+03	-2.609E+03	-1.211E+03

$$\delta_e = \sqrt{\frac{\sum_{i=1}^N (\delta_{ei})^2}{N_g}}$$

$$\delta_g = \sqrt{\frac{(1398)^2 + (1149)^2 + (-1335)^2 + (-1211)^2}{4}} = 1277 \quad \text{Eq. 10-6}$$

EID = 60			
GID	σ_e	σ_{gi}	δ_{ei}
61	-1.273379E+03	-2.609E+03	1.336E+03
62	-1.273379E+03	-2.485E+03	1.212E+03
103	-1.273379E+03	-9.155E-05	-1.273E+03
102	-1.273379E+03	-6.104E-05	-1.273E+03

$$\delta_e = \sqrt{\frac{(1336)^2 + (1212)^2 + (-1273)^2 + (-1273)^2}{4}} = 1274$$

The other two components (σ_y and τ_{xy}) can be calculated in a similar manner.

The above data provides an estimated error for each of the three stress components for each of the two elements considered. As discussed in the previous section, it is desirable to combine these three estimated errors into a single representative error estimate for each element. The root mean square value of the three error estimates for each of the three stress components offers such a representative error measure. In other words,

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{ei})^2}{N_c}} \quad \text{Eq. 10-7}$$

(The results (δ_e and ERROR ESTIMATE) from MSC.Nastran for this model, using CQUAD4, CQUAD8, and CQUAD4 with the corner option, are summarized in [Table 10-7](#) and [Table 10-8](#).

Table 10-7 The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types

Element Type	Location at EID = 20			
	NORMAL-X (δ_e)	NORMAL-Y (δ_e)	SHEAR-XY (δ_e)	ERROR ESTIMATE

Table 10-7 The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types

CQUAD4	1.277E+03	4.757E-09	6.212E+01	7.382E+02
CQUAD8	0.000E+00	4.014E-09	0.000E+00	2.318E-09
CQUAD4 (CORNER)	1.407E+02	2.990E+02	4.932E+02	3.428E+02

Table 10-8 The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types

Element Type	Location at EID = 60			
	NORMAL-X (δ_e)	NORMAL-Y (δ_e)	SHEAR-XY (δ_e)	ERROR ESTIMATE
CQUAD4	1.274E+03	4.520E-09	6.212E+01	7.365E+02
CQUAD8	5.038E-10	3.362E-09	0.000E+00	2.020E-09
CQUAD4 (CORNER)	1.003E+02	2.969E+02	4.915E+02	3.366E+02

The CHEXA(8) element is not included in the above tables; however, the calculations are similar with the exception that six rather than three components of stress need to be considered. In addition to values of δ_e for each stress component and the ERROR ESTIMATE of [Eq. 10-7](#), MSC.Nastran output also contains values of σ for each of the stress invariants. For plate elements, all of these quantities are output at the neutral plane and at fiber distances Z1 and Z2 for each element in the defined SURFACE.

Discussion of Error Measures

An examination of [Table 10-5](#) through [Table 10-8](#) clearly shows that the error estimates for CQUAD4 elements without vertex stress data are substantially larger than the corresponding error estimates for those elements that provide vertex stress data as a default. In view of the fact that the finite model under discussion was constructed with a rather high mesh density to produce results that agreed very closely with theory, you may perhaps be surprised by the large values of the error estimates for those elements that only provide stress data at the element centroid.

These seemingly large values of error estimators for CQUAD4 elements are not uncommon occurrences in statistical error measures. For example, the mean and variance have direct analogies in engineering mechanics. The mean is analogous to the centroid of a body and is thus a measure of where the mass is centered. The variance is the second moment about the mean, and it tends to be small if the majority of the mass is concentrated about the centroid. As the mass is dispersed further from the centroid, the moment of inertia tends to increase. If the mass is concentrated at the centroid (as it might be in some idealized lumped mass models), the moment of inertia becomes zero.

Thus, the error estimates for CQUAD4 elements tend toward larger values because the data used to compute the estimators is dispersed relatively far from the mean. For those elements that provide stress data at vertices, the estimators tend toward smaller values because the stress data tends to be concentrated about the mean in well-designed finite element models.

Generally, you should design a finite element mesh for static analysis so that all important stress gradients are adequately represented. If the mesh is not of sufficient detail, the stress data whether available at element vertices or at element centroids results in the relatively large values of the error estimators discussed here.

You are cautioned that very inaccurate values of these error estimators may occur at the edges of defined SURFACES and on the faces of defined VOLUMES. In summary, the error estimators under discussion can, in some cases, be highly inaccurate. Nevertheless, these data are quite useful when interpreted properly.

User Interface

The element and grid point stress discontinuity output can only be obtained if the grid point stress output is requested via the STRFIELD Case Control Command. The STRFIELD Case Control command is also used for graphical postprocessing of grid point stress, element stress discontinuity, and grid point stress discontinuity. The STRFIELD command, however, does not provide printed output for the grid point stress. The GPSTRESS Case Control command is needed if printed output for the grid point stress is desired. Similar to the GPSTRESS command, the STRFIELD command also requires that you define all applicable SURFACES and VOLUMES in the OUTPUT(POST) portion of the Case Control Section. The element stresses must also be requested for those elements that lie on the SURFACES and /or VOLUMES. For a more detailed description of GPSTRESS, refer to “[Grid Point Stresses](#)” on page 349.

Two other Case Control commands are needed--GPSDCON for grid point stress discontinuity and ELSDCON for element stress discontinuity. Note that you can also request grid point stress discontinuity without requesting element stress discontinuity, and vice versa, by omitting the appropriate Case Control command (ELSDON or GPSDCON).

The following remarks should be noted relative to the use of the GPSDCON and /or ELSDCON commands.

1. The GPSDCON and ELSDCON commands are supported only in Solution Sequence101.
2. The GPSDCON and ELSDCON commands may be placed above the SUBCASE level or in individual SUBCASEs and /or SUBCOMs. The use of these commands above the SUBCASE level causes the stress discontinuity data to be output for all SUBCASEs in the MSC.Nastran input file. If you wish to restrict stress discontinuity output to specific SUBCASEs, GPSDCON and /or ELSDCON commands should appear only under these specific SUBCASEs.

3. You are cautioned that these commands can produce a substantial amount of data. Therefore, you should be judicious in their use.
4. Stress continuity data is not provided when both plate and solid elements are connected to a grid point that is involved in stress discontinuity calculations.

The following is a listing of the input file used to generate the results for the CQUAD4 element model used in [Table 10-5](#) through [Table 10-8](#). The example problem in [Listing 10-7](#) contains the full list of Case Control commands required to obtain Grid Point / Element Stress Discontinuity output.

Listing 10-7 Input File for the Error Estimators for CQUAD4

```
$
$ FILENAME - q4sdcon.dat
$
ID STRESS DISCON
TIME 25
SOL 101
CEND
TITLE = CANTILEVER BEAM WITH PLATES
$
SPC=1
LOAD=1000
$
$
$ ELEMENT OUTPUT REQUIRED
$
SET 22 = 19,20,21,59,60,61
$
ELSTRESS = 22
$
$ FOLLOWING REFERENCES SURFACE 91
$
SET 21 = 91
GPSTRESS=21
STRFIELD=21
$
$ FOLLOWING ARE FOR STRESS DISCONTINUITIES
$
GPSDCON=21 $ FOR GRID POINT DISCONTINUITIES
ELSDCON=21 $ FOR ELEMENT DISCONTINUITIES
$
$ FOLLOWING REQUIRED FOR GRID POINT STRESS
$
OUTPUT(POST)
SET 95 = 19,20,21,59,60,61
$
SURFACE 91 SET 95 NORMAL Z SYSTEM ELEMENT
$
BEGIN BULK
$
$ BRING IN THE REST OF THE BULK DATA FILE
$
INCLUDE 'q4sdcon.blk'
$
$
ENDDATA
```

10.6 Postprocessor Checks

Once you have completed your MSC.Nastran job, you are now faced with perhaps the most important task--interpreting the results. After all, the whole purpose of performing an analysis is the ability to reap rewards from the results of your analysis. For a large model, it becomes impractical to go through every page of your output to look at the results. A postprocessor is a graphical tool that helps you siphon and interpret a voluminous amount of output data. This section covers some of the basic output quantities that you may want to consider when using a postprocessor.

Displacement

Prior to your analysis, you should have an approximate idea of how your structure is going to behave. A displacement plot should be created for each of your loading conditions. You should be convinced that the results are reasonable; otherwise, further investigation is warranted.

If there are abrupt changes in the displacements at a certain region, you may want to zoom in to inspect that local region and see if they are justifiable. One possible cause can be improper modeling at that location or certain elements not being connected.

Stress Contours

Stress contour plots are calculated using a stress averaging technique of neighboring elements. Once these average stresses are calculated, the postprocessor assigns different colors to these stresses based on the stress ranges. These stresses can then be plotted as stress contour plots with various colors designating different stress ranges. This feature is probably the most frequently used postprocessing option. When used properly, it can help you quickly identify high stress regions and provide you with insight regarding your model.

When you create stress contour plots of component stresses, you should ensure that the component that you are plotting is in a consistent direction for all the elements that are included in your plot. The default component direction may be a function of element type. Some postprocessing packages may also transform them into a common coordinate system. The following example problem in [Figure 10-27](#) illustrates the importance of consistent direction. The stresses in a CQUAD4 element are in terms of the element coordinate system (see “[MSC.Nastran Elements](#)” on page 65 and the *MSC.Nastran Quick Reference Guide*).

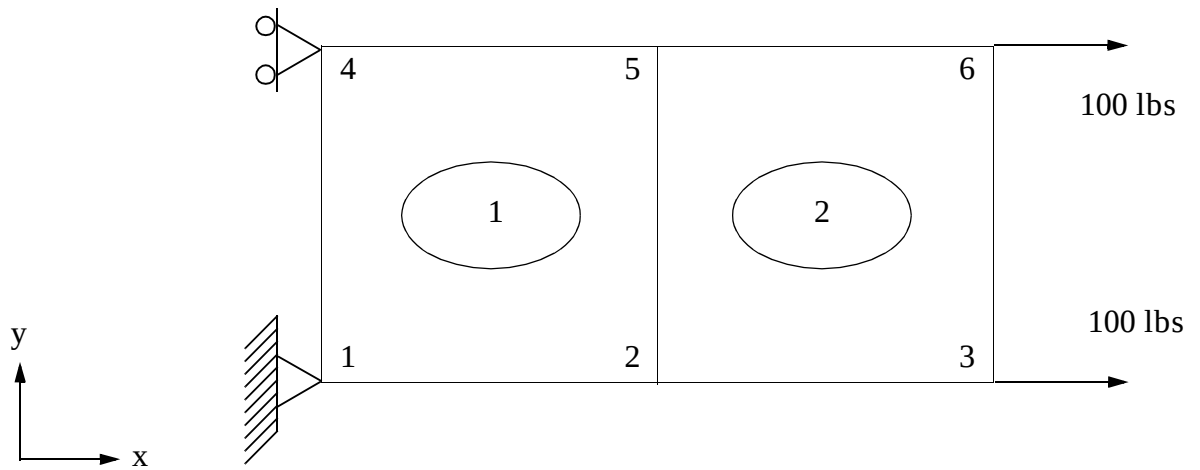


Figure 10-27 CQUAD4 Model

The complete input file used for this problem is shown in [Listing 10-8](#).

**Listing 10-8 Input File for CQUAD4 Stress Contour Plots
with Consistent Connectivity**

```
$
$ FILENAME - (q4consc.dat)
$
SOL      101
TIME     5
CEND
TITLE = CONSISTENT CQUAD4 CONNECTIVITIES
SPC =1
LOAD =1
STRESS = ALL
BEGIN BULK
$
$  FOR CREATING MSC/XL PLOTS
$
PARAM, POST, 0
$
$ THIS SECTION CONTAINS GRID POINT LOCATIONS
$
GRID     1           0.0       0.0       0.0
GRID     2          10.       0.0       0.0
GRID     3          20.       0.0       0.0
GRID     4           0.0      10.       0.0
GRID     5          10.      10.       0.0
GRID     6          20.      10.       0.0
$
```

Listing 10-8 Input File for CQUAD4 Stress Contour Plots with Consistent Connectivity (continued)

```

$ THIS SECTION CONTAINS ELEMENT CONNECTIVITIES
$
CQUAD4  1      10      1      2      5      4
CQUAD4  2      10      2      3      6      5
$
$CQUAD4  2      10      3      6      5      2
$
$ LOADS
$
FORCE   1      3      100.    1.
FORCE   1      6      100.    1.
$
$ CONSTRAINTS
$
SPC      1      1      12345    0.0
SPC      1      4      1345     0.0
$
$ SECTION AND MATERIAL PROPERTIES
$
PSHELL  10      10      .1      10
MAT1     10      1.+7      .32
$
ENDDATA

```

In this example, the connectivities for both CQUAD4 elements are described in a consistent manner as shown in [Figure 10-28](#). Due to this consistency, the corresponding component stresses, e.g., point in the same direction for both elements. Note that for this model is equal to zero for both elements.

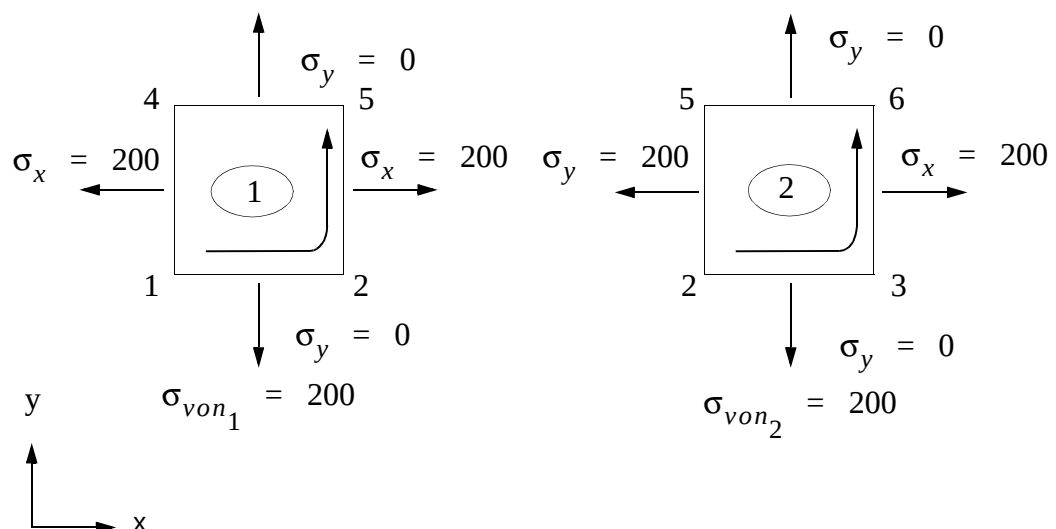


Figure 10-28 Consistent Connectivity

If you perform a stress contour on this model using σ_x , the contour plot for σ_x across the two elements can be calculated as follows:

$$\sigma_x = \frac{(\sigma_{x1} + \sigma_{x2})}{2} = \frac{(200 + 200)}{2} = 200.$$

The contour plot for σ_{von} across the two elements can be calculated as follows:

$$\sigma_{von} = \frac{(\sigma_{von1} + \sigma_{von2})}{2} = \frac{(200 + 200)}{2} = 200.$$

On the other hand, if the two elements are not connected in a consistent matter as shown in **Figure 10-29**, then the corresponding component stresses (e.g. σ_x ,) do not necessarily point in the same directions for both elements. As shown in **Figure 10-29**, the σ_x component stresses point in the directions of the x- and y-axis for elements 1 and 2, respectively.

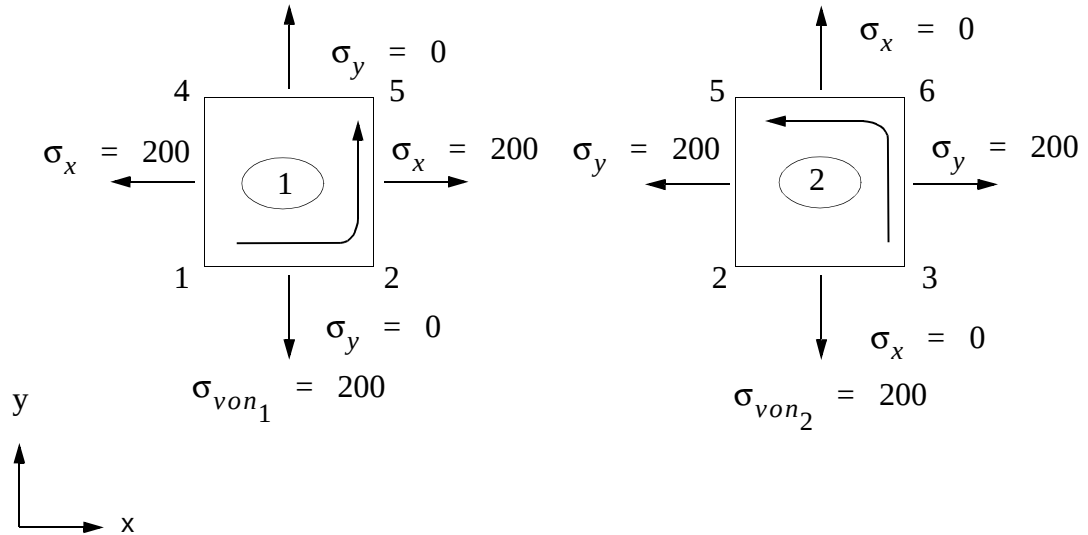


Figure 10-29 Inconsistent Connectivity

Once again, if you perform a stress contour on this model using , the value used in the contour plot for across the two elements in this case is as follows:

$$\sigma_x = \frac{(\sigma_{x1} + \sigma_{x2})}{2} = \frac{(200 + 0)}{2} = 100.$$

As you can see, the σ_x value used in the contour plot for this case is basically meaningless and could lead you to an erroneous conclusion.

The von Mises stresses used for the contour plot can be calculated as follows:

$$\sigma_{von} = \frac{(\sigma_{von1} + \sigma_{von2})}{2} = \frac{(200 + 200)}{2} = 200.$$

As you can see, σ_{von} remains the same since it is an invariant quantity and is independent of the order of connectivity. Note that the input file used for **Figure 10-29** is identical to **Listing 10-8** with the exception of the shaded CQUAD4 entry. In other words, replace the following CQUAD4 entry:

1	2	3	4	5	6	7	8	9	10
CQUAD 4	2	10	2	3	6	5			

with the following:

1	2	3	4	5	6	7	8	9	10
CQUAD 4	2	10	3	6	5	2			

and the two input files are then identical.

As you have probably observed by now, using contour plots of component stresses can lead to misleading results unless they are transformed into a common coordinate system. Therefore, in most instances, it is generally easier and more meaningful to look at the invariant stress quantities, such as von Mises stresses, instead of the component stresses. However, there are situations where you may want to look at component stresses. An example is a cylindrical model whereby you are interested in looking at the hoop stress. In this case, you should be aware of the consequences if your model is not created in a consistent manner. Alternate invariant stress quantities are the principal stresses.

Grid Point Stress and Stress Discontinuity Plots

The grid point stress feature enables you to obtain stress output at the grid points instead of the elements. This feature is described in detail in “**Grid Point Stresses**” on page 349. If the contour plot using grid point stress is substantially different from the contour plot using element stress, it is probably an indication that the model needs to be refined or that you may have inconsistent element orientations as described in the previous section.

Stress discontinuity is a measure of the probable error of the stresses across the grid points or elements. This type of plot assists you in the identification of regions of your model that may require refinement. This feature is discussed in detail in “**Stress Error Estimators**” on page 418.

Element Strain Energy Plots

The element strain energy plot is an excellent tool for identifying areas that are most effective in influencing your design. For a large model, this output request can generate a large amount of printed output. The plot option can be used to generate plots without generating printed output. This feature is discussed in detail in “**Strain Energy Output**” on page 379.

Summary

Many model verification tools have been presented in this chapter. We are not suggesting that you have to perform all these checks for all your models nor that they are the only ones to perform. The less familiar you are with your model, the more checks you probably should perform. But by performing these checks, you will notice errors that might otherwise have gone undetected. You will benefit by producing a much higher quality production finite element model.

Inertia Relief

- Introduction to Inertia Relief
- Description of Inertia Relief Using PARAM,INREL,-1
- Implementation of Inertia Relief in MSC.Nastran Using PARAM,INREL,-1
- Automatic Inertia Relief

11.1 Introduction to Inertia Relief

Inertia relief is an advanced option in MSC.Nastran that allows you to simulate unconstrained structures in a static analysis. Typical applications of inertia relief include modeling an aircraft in flight, an automobile on a test track, or a satellite in space. The inertia relief method is a tool that you may not be familiar with. To help understand the method, the first section provides a brief description of inertia relief. The last section shows its implementation within MSC.Nastran.

Although the internal processing is somewhat complex, the interface to the inertia relief option is quite straightforward. There are two ways to invoke the inertia relief. The first method is to specify the SUPORT entry explicitly by including “PARAM,INREL,-1” in the Bulk Data Section. The second method (the recommended method) is to let MSC.Nastran select the SUPORT degrees of freedom automatically by including “PARAM,INREL,-2” in the Bulk Data Section. An optional “PARAM,GRDPNT,x”--where x is a grid point ID--can be specified in the model. If “PARAM,GRDPNT,x” is used, the loads and accelerations will be summed about this point. If “PARAM,GRDPNT,x” is not specified, then the loads and accelerations will be summed about the origin of the basic coordinate system.

11.2 Description of Inertia Relief Using PARAM,INREL,-1

As discussed previously, static analysis by the finite element method assumes that the model contains no mechanisms and may not move as a rigid body (strain free). If either of these conditions exists in a conventional finite element analysis, the stiffness matrix for the model becomes singular. When MSC.Nastran attempts to decompose a singular matrix, a fatal message or unreasonable answers result.

Consequently, conventional finite element static analysis cannot be performed on unconstrained structures. However, a method called inertia relief is provided in MSC.Nastran for analyzing these conditions. A simple description of inertia relief is that the inertia (mass) of the structure is used to resist the applied loadings, that is, an assumption is made that the structure is in a state of static equilibrium even though it is not constrained. Two examples are a spacecraft in orbit or an aircraft in flight. In these cases, the structure is in state of static equilibrium, although it is capable of unconstrained motion.

To invoke inertia relief, you must provide a SUPORT Bulk Data entry with a list of up to six non-redundant degrees of freedom that describe the possible unconstrained motion. The easiest way to describe how to use the SUPORT entry in static analysis is if you hold the SUPORT degrees of freedom constrained, there is no possible rigid body motion. If all possible rigid body motion is not described on the SUPORT entry, then the stiffness matrix is singular, and the problem either fails in decomposition or gives unreasonable answers.

When inertia relief is specified, MSC.Nastran calculates the forces that result from a rigid body acceleration about the point specified on the “PARAM,GRDPNT,x” in the specified directions. MSC.Nastran then calculates the summation of all applied loadings in the same directions. Accelerations are applied to the structure in the appropriate directions to “balance” the applied loadings. The structure is now in a state of static equilibrium, i.e., the summation of all applied loads is 0.0. Since the problem is not constrained, rigid body displacement is still possible.

The next step performed by MSC.Nastran is to constrain the SUPORT degrees of freedom to a displacement of 0.0 and provide the relative motion of all other grid points with respect to that reference point. Hence, the term “reference” degree of freedom is used to describe the SUPORT degrees of freedom in MSC.Nastran. The set of degrees of freedom described on the SUPORT entry belong to the *r*-set or reference set for the solution. The computed solution is the correct one, and it is relative to any rigid body motion that is occurring. A simple way to think of this is that the solution coming from MSC.Nastran represents the deformation of the structure you would see if you were standing at the SUPORT degrees of freedom.

11.3 Implementation of Inertia Relief in MSC.Nastran Using PARAM,INREL,-1

Solution 101 requires that the model does not contain mechanisms. The reference degrees of freedom are specified using the the SUPORT entry as follows:

1	2	3	4	5	6	7	8	9	10
SUPPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Field	Contents
IDi	Grid or scalar point identification number.
Ci	Component numbers.

In addition to the PARAM,INREL,-1 entry, a SUPORT entry is also needed. If PARAM,GRDPNT,x is specified, the rigid body mass matrix is calculated about grid point x; otherwise, the rigid body mass is calculated about the origin of the basic coordinate system. The inertia relief method requires that a realistic mass distribution exists, and the degrees of freedom listed on the SUPORT entry must be connected elastically to the model in all specified degrees of freedom. (For example, degrees of freedom 4, 5, and 6 on a grid point with only solid elements attached cannot be used since solid elements have no stiffness in these degrees of freedom.)

Whenever a SUPORT entry is used in static analysis, the epsilon and strain energy printed in the table from UIM 3035 should be checked. The values printed for epsilon and strain energy in the UIM 3035 table should all be numeric zero. The strain energy printed in this table for matrix KLR represents the strain energy in the model when the one SUPORT degree of freedom is moved 1.0 unit, while all other SUPORT degrees of freedom are constrained. If the SUPORT degrees of freedom are properly specified, the model should be capable of rigid body motion (strain-free) with no strain energy.

The values printed for the strain energy indicate the ability of the model to move as a rigid body. These values should always be checked. If the structure is not constrained, the values should be numeric zero, but roundoff almost always results in a small nonzero value. Acceptable values are a function of the units, size of the structure, and precision of the hardware; therefore, a recommended value is not provided in this user’s guide.

An additional feature allowed in Solution 101 is the solution of a problem under uniform acceleration. This problem is posed using the DMIG,UACCEL entry in addition to the previously mentioned requirements. In this case, uniform accelerations are applied to the model, and the solution is found. Uniform accelerations are useful for situations, such as spacecraft liftoff and landing loadings, which are often specified as static accelerations.

As an example of inertia relief, consider the three CBEAM model shown in [Figure 11-1](#).

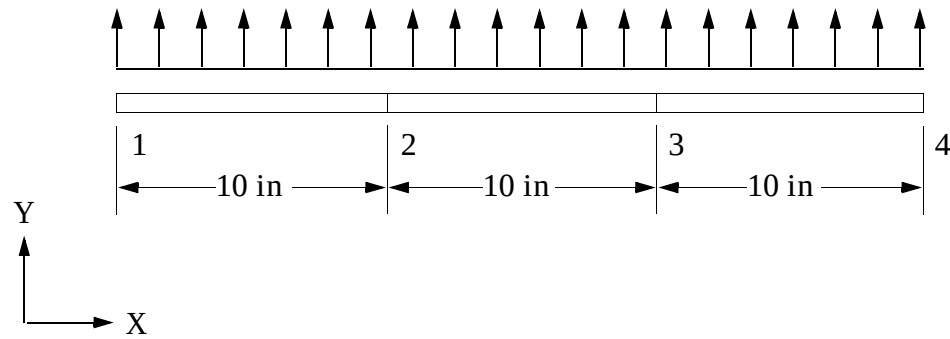


Figure 11-1 Inertia Relief Analysis of a CBEAM Model

The three CBEAM structure is to be analyzed as a free-free structure with a line load acting in the Y-direction as shown and using inertia relief. To show the effect of the SUPORT point, two runs are made, each with a different SUPORT grid point. The input file is given in [Listing 11-1](#). Note that the SUPORT information is shown for both runs; however, one set is commented out. For the first run, “inertia1.dat”, the SUPORT point is located at grid point 1. All six degrees of freedoms for the SUPORT point are placed on one grid point as required for Solution 101. The parameter INREL is set to -1, and the parameter GRDPNT is set to the SUPORT point, which is grid point 1 in this case.

Listing 11-1 Three CBEAM Model with Inertia Relief

```

$
$ FILENAME - INERTIA1.DAT
$
ID      LINEAR,INERTIA
SOL      101
TIME     5
CEND
TITLE = BAR WITH SUPORT ENTRY
LOAD = 1
DISP = ALL
SPCF = ALL
STRESS = ALL
BEGIN BULK
PARAM    POST      0
$
$
$ SUPORT ENTRY FOR INERTIA1.DAT
$
SUPORT   1          123456
PARAM    GRDPNT    1
PARAM    INREL     -1
$
$ SUPORT ENTRY FOR INERTIA2.DAT
$
$SUPORT   3          123456
$PARAM    GRDPNT    3
$PARAM    INREL     -1
$
GRID     1          0.0      0.0      0.0
GRID     2          10.      0.0      0.0
GRID     3          20.      0.0      0.0
GRID     4          30.      0.0      0.0
$
CBEAM    1          1          1          2          1.          1.          0.0
CBEAM    2          1          2          3          1.          1.          0.0
CBEAM    3          1          3          4          1.          1.          0.0
$
.5        .5          .5          .5
PBEAM    1          1          1.          .667      .167          .1
.5        .1          1.5        -.1          -.5          .1          -.5          -.1
PLOAD1   1          1          FY          FR          0.          1000.      1.          1000.
PLOAD1   1          2          FY          FR          0.          1000.      1.          1000.
PLOAD1   1          3          FY          FR          0.          1000.      1.          1000.
$
MAT1     1          1.+7          .3          7.43E-3
ENDDATA

```

A partial listing of the output is shown in [Figure 11-2](#).

1	BAR WITH SUPORT ENTRY				OCTOBER 16, 2001		MSC.NASTRAN 4/ 9/01		PAGE 7
0									
0	O U T P U T F R O M G R I D P O I N T W E I G H T G E N E R A T O R								
	REFERENCE POINT = 1								
	M O								
	*	2.229000E-01	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	*	
	*	0.000000E+00	2.229000E-01	0.000000E+00	0.000000E+00	0.000000E+00	3.343500E+00	*	
	*	0.000000E+00	0.000000E+00	2.229000E-01	0.000000E+00	-3.343500E+00	0.000000E+00	*	
	*	0.000000E+00	0.000000E+00	0.000000E+00	1.858986E-01	0.000000E+00	0.000000E+00	*	
	*	0.000000E+00	0.000000E+00	-3.343500E+00	0.000000E+00	7.058500E+01	0.000000E+00	*	
	*	0.000000E+00	3.343500E+00	0.000000E+00	0.000000E+00	0.000000E+00	7.058500E+01	*	
	S								
		*	1.000000E+00	0.000000E+00	0.000000E+00	*			
		*	0.000000E+00	1.000000E+00	0.000000E+00	*			
		*	0.000000E+00	0.000000E+00	1.000000E+00	*			
	DIRECTION								
	MASS AXIS SYSTEM (S)	MASS		X-C.G.		Y-C.G.		Z-C.G.	
	X	2.229000E-01		0.000000E+00		0.000000E+00		0.000000E+00	
	Y	2.229000E-01		1.500000E+01		0.000000E+00		0.000000E+00	
	Z	2.229000E-01		1.500000E+01		0.000000E+00		0.000000E+00	
	I(S)								
		*	1.858986E-01	0.000000E+00	0.000000E+00	*			
		*	0.000000E+00	2.043250E+01	0.000000E+00	*			
		*	0.000000E+00	0.000000E+00	2.043250E+01	*			
	I(Q)								
		*	1.858986E-01			*			
		*		2.043250E+01		*			
		*			2.043250E+01	*			
	Q								
		*	1.000000E+00	0.000000E+00	0.000000E+00	*			
		*	0.000000E+00	1.000000E+00	0.000000E+00	*			
		*	0.000000E+00	0.000000E+00	1.000000E+00	*			
1	BAR WITH SUPORT ENTRY				OCTOBER 16, 2001		MSC.NASTRAN 4/ 9/01		PAGE 9
0									
0	RESULTANTS ABOUT 1 IN SUPERELEMENT BASIC SYSTEM COORDINATES.								
0									
	SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3	
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00	
		FY	----	3.000000E+04	----	0.000000E+00	----	4.500000E+05	
		FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----	
		MX	----	----	----	0.000000E+00	----	----	
		MY	----	----	----	----	0.000000E+00	----	
		MZ	----	----	----	----	----	5.456968E-11	
	TOTALS		0.000000E+00	3.000000E+04	0.000000E+00	0.000000E+00	0.000000E+00	4.500000E+05	
0	2	FX	-2.229000E-01	----	----	----	0.000000E+00	0.000000E+00	
		FY	----	0.000000E+00	----	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	----	----	----	----	----	0.000000E+00	
	TOTALS		-2.229000E-01	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
0	3	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00	
		FY	----	-2.229000E-01	----	0.000000E+00	----	-3.343500E+00	
		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	-2.229000E-01	0.000000E+00	0.000000E+00	0.000000E+00	-3.343500E+00	
0	4	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00	
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00	
		FZ	----	----	-2.229000E-01	0.000000E+00	3.343500E+00	----	
		MX	----	----	----	0.000000E+00	----	----	
		MY	----	----	----	----	0.000000E+00	----	
		MZ	----	----	----	----	----	0.000000E+00	

Figure 11-2 Inertia Relief Output When Grid Point 1 Is the SUPORT Point


```

TOTALS 0.000000E+00 0.000000E+00 -2.229000E-01 0.000000E+00 3.343500E+00 0.000000E+00
0 5 FX 0.000000E+00 ---- ---- 0.000000E+00 0.000000E+00
FY ---- 0.000000E+00 ---- 0.000000E+00 ---- 0.000000E+00
FZ ---- ---- 0.000000E+00 0.000000E+00 0.000000E+00 ----
MX ---- ---- ---- -1.858986E-01 ----
MY ---- ---- ---- 0.000000E+00 ----
MZ ---- ---- ---- ---- 0.000000E+00
TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00
0 6 FX 0.000000E+00 ---- ---- 0.000000E+00 0.000000E+00
FY ---- 0.000000E+00 ---- 0.000000E+00 ---- 0.000000E+00
FZ ---- ---- 3.343500E+00 0.000000E+00 -7.058500E+01 ----
MX ---- ---- ---- 0.000000E+00 ----
MY ---- ---- ---- 0.000000E+00 ----
MZ ---- ---- ---- ---- 0.000000E+00
TOTALS 0.000000E+00 0.000000E+00 3.343500E+00 0.000000E+00 -7.058500E+01 0.000000E+00
0 7 FX 0.000000E+00 ---- ---- 0.000000E+00 0.000000E+00
FY ---- -3.343500E+00 ---- 0.000000E+00 ---- -7.058500E+01
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 -3.343500E+00 0.000000E+00 0.000000E+00 0.000000E+00 -7.058500E+01
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 10

0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
FOR DATA BLOCK KLR
SUPPORT PT.NO. EPSILON STRAIN ENERGY EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1 2.3344220E-15 0.0000000E+00
2 2.3344220E-15 7.2759576E-12
3 2.3344220E-15 0.0000000E+00
4 2.3344220E-15 0.0000000E+00
5 2.3344220E-15 0.0000000E+00
6 2.3344220E-15 5.3551048E-09
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 11

0
INTERMEDIATE MATRIX ... QRR

1 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 6
1 0.000000E+00 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 3.343500E+00 6
1 0.000000E+00 0.000000E+00 2.229000E-01 0.000000E+00 -3.343500E+00 0.000000E+00 6
1 0.000000E+00 0.000000E+00 0.000000E+00 1.858986E-01 0.000000E+00 0.000000E+00 6
1 0.000000E+00 0.000000E+00 -3.343500E+00 0.000000E+00 7.058500E+01 0.000000E+00 6
1 0.000000E+00 3.343500E+00 0.000000E+00 0.000000E+00 0.000000E+00 7.058500E+01 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 12

0
INTERMEDIATE MATRIX ... QRL

1 0.000000E+00 -3.000000E+04 0.000000E+00 0.000000E+00 0.000000E+00 -4.500000E+05 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 13

```

Figure 11-2 Inertia Relief Output When Grid Point 1 Is the SUPORT Point (continued)

0	INTERMEDIATE MATRIX ... URA									
				COLUMN	1					
1	0.000000E+00	1.345895E+05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	8.246463E-13		6	
*** 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	-1.6152625E-14		1.5617192E+02						
1	BAR WITH SUPORT ENTRY					OCTOBER 16, 2001	MSC.NASTRAN 4/ 9/01	PAGE	15	
0	D I S P L A C E M E N T V E C T O R									
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
	1	G	0.0	0.0	0.0	0.0	0.0	0.0		
	2	G	0.0	-6.246877E-02	0.0	0.0	0.0	-1.249375E-02		
	3	G	0.0	-2.498751E-01	0.0	0.0	0.0	-2.498751E-02		
	4	G	0.0	-5.622189E-01	0.0	0.0	0.0	-3.748126E-02		
1	BAR WITH SUPORT ENTRY					OCTOBER 16, 2001	MSC.NASTRAN 4/ 9/01	PAGE	16	
0	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	-8.807738E-13	0.0	0.0	0.0	-6.462349E-27		
1	BAR WITH SUPORT ENTRY					OCTOBER 16, 2001	MSC.NASTRAN 4/ 9/01	PAGE	17	
0										
1	BAR WITH SUPORT ENTRY					OCTOBER 16, 2001	MSC.NASTRAN 4/ 9/01	PAGE	18	
0										
	S T R E S S E S I N B E A M E L E M E N T S (C B E A M)									
	ELEMENT-ID	GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T M.S.-C
0	1									
		1	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		2	1.000	0.0	0.0	0.0	0.0	0.0	0.0	
0	2									
		2	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		3	1.000	0.0	0.0	0.0	0.0	0.0	0.0	
0	3									
		3	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		4	1.000	0.0	0.0	0.0	0.0	0.0	0.0	

Figure 11-2 Inertia Relief Output When Grid Point 1 Is the SUPORT Point (continued)

The GPWG (Grid Point Weight Generator) output is calculated with respect to the grid point specified on the parameter GRDPNT. The total mass of the structure in this model is 0.229, which is used by MSC.Nastran to develop the inertia loads. Inspection of the GPWG output should be part of your routine model checkout.

The OLOAD RESULTANT contains seven sections--consisting of seven lines per section. The first section is the resultant of the applied loads about the GRDPNT, which is the basic coordinate system in this case. The remaining six sections are the loads necessary to impose unit acceleration about the SUPORT point. Subcase 2 corresponds to the X-direction; Subcase 3 corresponds to Y-direction, etc. The first six lines of each section represent the detailed contributions, whereas the seventh line represents the total.

Following the OLOAD output is User Information Message 3035 showing the strain energy and epsilon due to the imposed unit accelerations about the SUPORT point. As discussed earlier, the epsilons and strain energies should be small, which is the case for this example. Small epsilons and strain energy tell you that you do not have any unwanted constraints or poorly defined MPCs causing a constraint in your model. Always inspect User Information Message 3035 to make sure that the epsilons and strain energies are small.

The intermediate matrix QRR is printed following the UIM 3035. The QRR matrix is the total rigid body mass of the total structure. It is a 6 x 6 matrix measured about the PARAM,GRDPNT point in the global coordinate system. Masses on the scalar points are not included. Following the QRR matrix is the QRL matrix. The QRL matrix is the resultant of the “apparent reaction loads,” measured at the SUPORT point. This resultant is equal and opposite to the OLOAD resultant shown earlier in the output, if the SUPORT point is the same as the PARAM,GRDPNT point. There is one column for each loading condition. The last matrix output is the URA matrix, which is the rigid body acceleration matrix that is computed from the applied loads.

The displacement and stress output shown is the standard output as requested through the Case Control Section. Note that the displacement at the SUPORT point is exactly 0.0. This SUPORT point should be 0.0 since this point is constrained for the solution, and the forces of constraint should be numeric zero. The SPC forces are shown in the output to confirm that the loads are balanced at the SUPORT point. The displacement of all of the other points in the model are relative to the SUPORT point.

An interesting effect occurs when the SUPORT point is changed in the previous example. Suppose you change the location of the SUPORT point from 1 to 3 (see the commented SUPORT entries in [Listing 11-1](#)).

The resulting displacement and stress output is shown in [Figure 11-3](#).

```

1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 7

0
0      OUTPUT FROM GRID POINT WEIGHT GENERATOR
0      REFERENCE POINT = 3
      M 0
      * 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 -1.114500E+00 *
      * 0.000000E+00 0.000000E+00 2.229000E-01 0.000000E+00 1.114500E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 0.000000E+00 1.858986E-01 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.114500E+00 0.000000E+00 2.600500E+01 0.000000E+00 *
      * 0.000000E+00 -1.114500E+00 0.000000E+00 0.000000E+00 0.000000E+00 2.600500E+01 *
      S
      * 1.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 1.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.000000E+00 *
      DIRECTION
      MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
      X      2.229000E-01      0.000000E+00 0.000000E+00 0.000000E+00
      Y      2.229000E-01      -5.000000E+00 0.000000E+00 0.000000E+00
      Z      2.229000E-01      -5.000000E+00 0.000000E+00 0.000000E+00
      I(S)
      * 1.858986E-01 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 2.043250E+01 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 2.043250E+01 *
      I(Q)
      * 1.858986E-01 *
      * 2.043250E+01 *
      * 2.043250E+01 *
      Q
      * 1.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 1.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.000000E+00 *

0
0      OLOAD      RESULTANT
0      SUBCASE/  LOAD
      DAREA ID  TYPE      T1      T2      T3      R1      R2      R3
0      1      FX      0.000000E+00      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      3.000000E+04      ----      0.000000E+00 ----      -1.500000E+05
      FZ      ----      ----      0.000000E+00 0.000000E+00 0.000000E+00 ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      5.456968E-11
      TOTALS  0.000000E+00 3.000000E+04 0.000000E+00 0.000000E+00 0.000000E+00 -1.500000E+05
0      2      FX      -2.229000E-01      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00      ----      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      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS  -2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0      3      FX      0.000000E+00      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      -2.229000E-01      ----      0.000000E+00 ----      1.114500E+00
      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 -2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 1.114500E+00
0      4      FX      0.000000E+00      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00      ----      0.000000E+00 ----      0.000000E+00
      FZ      ----      ----      -2.229000E-01 0.000000E+00 -1.114500E+00 ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS  0.000000E+00 0.000000E+00 -2.229000E-01 0.000000E+00 -1.114500E+00 0.000000E+00
0      5      FX      0.000000E+00      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00      ----      0.000000E+00 ----      0.000000E+00
      FZ      ----      ----      0.000000E+00 0.000000E+00 0.000000E+00 ----
      MX      ----      ----      ----      -1.858986E-01 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00

```

Figure 11-3 Inertia Relief Output When Grid Point 3 Is the SUPORT Point

```

TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00
0      6  FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY  ----  0.000000E+00  ----  0.000000E+00  ----  0.000000E+00
      FZ  ----  ----  -1.114500E+00  0.000000E+00  -2.600500E+01  ----
      MX  ----  ----  ----  0.000000E+00  ----  ----
      MY  ----  ----  ----  ----  0.000000E+00  ----
      MZ  ----  ----  ----  ----  ----  0.000000E+00
TOTALS 0.000000E+00 0.000000E+00 -1.114500E+00 0.000000E+00 -2.600500E+01 0.000000E+00
0      7  FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY  ----  1.114500E+00  ----  0.000000E+00  ----  -2.600500E+01
      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 1.114500E+00 0.000000E+00 0.000000E+00 0.000000E+00 -2.600500E+01
1  BAR WITH SUPORT ENTRY  OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 10

0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
      FOR DATA BLOCK KLR
SUPPORT PT.NO.  EPSILON  STRAIN  ENERGY  EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
      1  8.8574680E-16  0.0000000E+00
      2  8.8574680E-16  1.4551915E-11
      3  8.8574680E-16  0.0000000E+00
      4  8.8574680E-16  0.0000000E+00
      5  8.8574680E-16  1.1641532E-10
      6  8.8574680E-16  3.2596290E-09
1  BAR WITH SUPORT ENTRY  OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 11

0
INTERMEDIATE MATRIX ... QRR

      COLUMN 1
1  2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  6
      COLUMN 2
1  0.000000E+00  2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  -1.114500E+00  6
      COLUMN 3
1  0.000000E+00  0.000000E+00  2.229000E-01  0.000000E+00  1.114500E+00  0.000000E+00  6
      COLUMN 4
1  0.000000E+00  0.000000E+00  0.000000E+00  1.858986E-01  0.000000E+00  0.000000E+00  6
      COLUMN 5
1  0.000000E+00  0.000000E+00  1.114500E+00  0.000000E+00  2.600500E+01  0.000000E+00  6
      COLUMN 6
1  0.000000E+00  -1.114500E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.600500E+01  6
1  BAR WITH SUPORT ENTRY  OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 12

0
INTERMEDIATE MATRIX ... QRL

      COLUMN 1
1  0.000000E+00  -3.000000E+04  0.000000E+00  0.000000E+00  0.000000E+00  1.500000E+05  6
1  BAR WITH SUPORT ENTRY  OCTOBER 16, 2001 MSC.NASTRAN 4/ 9/01 PAGE 13

0
INTERMEDIATE MATRIX ... URA

      COLUMN 1
1  0.000000E+00  1.345895E+05  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  6

*** 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  -2.6191993E-15  1.5617192E+02

```

Figure 11-3 Inertia Relief Output When Grid Point 3 Is the SUPORT Point(continued)

1	BAR WITH SUPORT ENTRY					OCTOBER	16, 2001	MSC.NASTRAN	4/ 9/01	PAGE	14
DISPLACEMENT VECTOR											
POINT ID.		TYPE	T1	T2	T3	R1	R2	R3			
1		G	0.0	-2.498751E-01	0.0	0.0	0.0	2.498751E-02			
2		G	0.0	-6.246877E-02	0.0	0.0	0.0	1.249375E-02			
3		G	0.0	0.0	0.0	0.0	0.0	0.0			
4		G	0.0	-6.246877E-02	0.0	0.0	0.0	-1.249375E-02			
1	BAR WITH SUPORT ENTRY					OCTOBER	16, 2001	MSC.NASTRAN	4/ 9/01	PAGE	16
0											
1	BAR WITH SUPORT ENTRY					OCTOBER	16, 2001	MSC.NASTRAN	4/ 9/01	PAGE	17
0											
STRESSES IN BEAM ELEMENTS (CBEAM)											
ELEMENT-ID		GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
0	1										
		1	0.000	0.0	0.0	0.0	0.0	0.0	0.0		
		2	1.000	0.0	0.0	0.0	0.0	0.0	0.0		
0	2										
		2	0.000	0.0	0.0	0.0	0.0	0.0	0.0		
		3	1.000	0.0	0.0	0.0	0.0	0.0	0.0		
0	3										
		3	0.000	0.0	0.0	0.0	0.0	0.0	0.0		
		4	1.000	0.0	0.0	0.0	0.0	0.0	0.0		

Figure 11-3 Inertia Relief Output When Grid Point 3 Is the SUPORT Point(continued)

As can be seen, the displacement vector changes as the displacements are now relative to grid point 3, which is the new SUPORT point. However, the stresses in the elements are the same because the stresses are based on the relative displacement between the grid points, which is independent of the grid point used for the SUPORT point.

A final comment on this simple beam model: suppose you used CBAR elements instead of CBEAM elements for this model. The run would fail because the CBAR element does not have any torsional inertia mass, whereas the CBEAM element does. This model is basically a one-dimensional structure so that the rotational inertia about the X-axis is the mass inertia of the elements. When the inertia relief method is used, the structure must have mass in all six directions. In general, however, there are no restrictions on using a CBAR element with inertia relief. Most structures include CBARs that are not colinear. The example above was introduced merely to emphasize the need to have mass in all six degrees of freedom.

Suppose you wish to impose a 10 g acceleration at grid point 2 of the beam structure shown in [Figure 11-1](#). This acceleration can be applied using the DMIG,UACCEL Bulk Data entry as shown below.

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	"0"	"9"	TIN					
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

Field	Contents
TIN	Type of matrix being input.
L	Load sequence number.
Gi	Grid point identification number of a single reference point.
Ci	Component number for Gi in the basic coordinate system.
Xi	Value of enforced acceleration term in the basic coordinate system.

The input file for this example is shown in [Listing 11-2](#).

Listing 11-2 Imposing a 10 g Acceleration Using the PARAM,UACCEL

```

$
$ FILENAME - UACCEL
$
ID      LINEAR,UACCEL
SOL      101
TIME      5
CEND
TITLE = BAR WITH UACCEL INPUT
OLOAD = ALL
DISP = ALL
BEGIN BULK
PARAM    POST      0
$
$
$ SUPORT INFORMATION
$
SUPORT   2          123456
PARAM    GRDPNT    2
PARAM    INREL     -1
DMIG     UACCEL    0          9          1
DMIG     UACCEL    1          2          2          3864.0
$
GRID     1          0.0        0.0        0.0
GRID     2          10.        0.0        0.0
GRID     3          20.        0.0        0.0
GRID     4          30.        0.0        0.0
$
CBEAM    1          1          1          2          1.          1.          0.0
CBEAM    2          1          2          3          1.          1.          0.0
CBEAM    3          1          3          4          1.          1.          0.0
$
PBEAM    1          1          1.          .667          .167          .1
          .1          .1          .1          -.1          -.1          .1          -.1          -.1
$
MAT1     1          1.+7          .3          7.43E-3
ENDDATA

```

The acceleration of 10 g is entered on the DMIG,UACCEL entry. When a DMIG (Direct Matrix Input) entry is used, UACCEL, the first entry is the header entry, which is denoted by the 0 in field 3. Field 4 must be a 9, and field 5 is either a 1 for single precision input (the most common) or a 2 for double precision input. For this example, the input is in single precision.

The second entry is where the applied acceleration is entered. Field 3 must be a 1, indicating that this is the first load case (and the only load case in this model). The grid point component and value of the enforced acceleration are entered in fields 6, 7, and 8. If acceleration is to be enforced in more than one component, then the continuation entries are used. For this example, the acceleration of 3864. in / sec² is entered in field 8. The 3864.0 is 10 g in the English system-you must ensure that the units are consistent.

When using the DMIG,UACCEL, the intermediate matrix URACCEL is output as shown in [Figure 11-4](#). This matrix is the rigid body acceleration that you input using the DMIG,UACCEL entry. Also shown is the OLOAD output and the displacement vector. As can be seen, the SUPORT point is constrained, which is what you would expect, and the total load applied to the structure corresponds to the mass of the structure (see the GPWG output in [Figure 11-4](#)) times the enforced acceleration.

Although not commonly done, you can perform multiple inertia relief analyses in a single run. This feature can be activated with the SUPORT1 entries instead of the SUPORT entry. Unlike the SUPORT entry, which is automatically activated, the SUPORT1 entry must be called out by the SUPORT1 Case Control command in order for it to be applied.


```

O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
                                REFERENCE POINT =      2
                                M O
* 2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.114500E+00 *
* 0.000000E+00  0.000000E+00  2.229000E-01  0.000000E+00 -1.114500E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  0.000000E+00  1.858986E-01  0.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00 -1.114500E+00  0.000000E+00  2.600500E+01  0.000000E+00 *
* 0.000000E+00  1.114500E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.600500E+01 *
                                S
* 1.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  1.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
X      2.229000E-01      0.000000E+00  0.000000E+00  0.000000E+00
Y      2.229000E-01      5.000000E+00  0.000000E+00  0.000000E+00
Z      2.229000E-01      5.000000E+00  0.000000E+00  0.000000E+00
                                I(S)
* 1.858986E-01  0.000000E+00  0.000000E+00 *
* 0.000000E+00  2.043250E+01  0.000000E+00 *
* 0.000000E+00  0.000000E+00  2.043250E+01 *
                                I(Q)
* 1.858986E-01 *
* 2.043250E+01 *
* 2.043250E+01 *
                                Q
* 1.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  1.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  1.000000E+00 *
                                INTERMEDIATE MATRIX ... URA
                                COLUMN      1
1      0.000000E+00      3.864000E+03      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      6
DISPLACEMENT VECTOR
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      G      .0      -7.547017E-03      .0      .0      .0      1.076069E-03
2      G      .0      .0      .0      .0      .0      .0
3      G      .0      -3.340174E-02      .0      .0      .0      -5.380345E-03
4      G      .0      -9.475221E-02      .0      .0      .0      -6.456414E-03
LOAD VECTOR
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      G      .0      -1.435476E+02      .0      .0      .0      .0
2      G      .0      -2.870952E+02      .0      .0      .0      .0
3      G      .0      -2.870952E+02      .0      .0      .0      .0
4      G      .0      -1.435476E+02      .0      .0      .0      .0

```

Figure 11-4 Partial Output of the UACCEL Example

11.4 Automatic Inertia Relief

In the previous sections, manual inertia relief is discussed where the SUPORT entry must be specified explicitly along with PARAM,INREL,-1. The SUPORT entry lists the DOFs in the r-set, the reference set. Some skill is required to use this entry correctly. When the r-set is chosen such that structure is not constrained in a statically determinate manner, or if the stiffness attached to these points is inadequate, the rigid body mode shapes may be poor, leading to low accuracy in inertial load calculations. A poor static analysis solution may also result from the poor constraints, independent of the loading problems. Both effects may result in poor overall accuracy. Poor results can occur with little warning when the SUPORT entry is misused.

This section discusses the automatic inertia relief, the recommended inertia relief method. With the automatic method, the specification of the SUPORT entry is no longer needed. To turn it on, simply add PARAM,INREL,-2 to the input file. The reference frame is selected automatically, in a manner that poor solutions are unlikely because of the choice of reference frame variables.

The constraints associated with the reference frame are distributed to all points with mass. This means that structures with modeling errors, such as a region of elements left out through oversight, will still give reasonable results that can aid in diagnosing the modeling errors. A model in development may contain many disjoint parts, inadvertently. Any part with three non-collinear points with mass will be adequately constrained for solution.

In the manual support option, the free stiffness matrix K_{aa} is constrained by removing the r-set DOFs from the a-set and imposing zero motion on them,

$$[K_{aa}] \cdot [u_a] = \{P_a\} \quad \text{Eq. 11-1}$$

$$[u_r] = \{0\} \quad \text{Eq. 11-2}$$

With the automatic inertia relief, the u_r variables are no longer a subset of the u_a variables. The constraint equation (Eq. 11-2) is replaced by a more general constraint equation,

$$[M_{aa} \cdot D_{a6}]^T \cdot \{u_a\} = \{u_r\} = \{0\} \quad \text{Eq. 11-3}$$

The variables u_r are a set of generalized coordinates that represent the average motion of the u_a variables, as weighted by the mass matrix. The zero value for this equation implies that the average motion of all a-set points is zero, although the displacement variables are free to move relative to this least-squares fit reference frame.

The manual style of inertia relief allows use of PARAM,GRDPNT to define the origin used in determining the shape functions D_{a6} . The presence of this parameter causes the output of the Grid Point Weight Generator (GPWG) table, which lists the c.g. location of the model, its moments of inertia, and other related data. This point in space is used as the reference point when computing rigid body mass and moments of inertia, and rigid body

accelerations. If the parameter is not present, the basic origin is used, in the basic coordinate system. It is good practice (but not required by MSC.Nastran) to use the same grid point on PARAM,GRDPNT and on the SUPORT entry.

Unlike the manual option, PARAM,GRDPNT is not used as the inertia relief reference point for the automatic support option. The basic origin is used. If the PARAM,GRDPNT is present, it causes the generation of the GPWG table, but does not influence the inertia relief calculations in any way.

The constraint forces caused by the distributed constraints are printed with the SPCFORCE case control command. They are computational zeros for a well-defined model.

PARAM,TINY (default value is 1.E-3) is traditionally used to discard element strain energies with values less than “tiny”. The same parameter requests that small SPC forces of the reference frame constraint type be replaced with binary zeros, for the automatic support option only. This filtering is not used on conventional SPC forces from selected SPC entries or AUTOSPC, and manual support DOFs.

Limitations for the Automatic Inertia Relief Method

The following features are not supported for the Automatic Inertia Relief Method:

- The DMIG,UACCEL option
- The Iterative Solver

EXAMPLE

Let’s revisit the CBEAM model in [Figure 11-1](#) using the automatic inertia relief method. This is accomplished by removing the SUPORT entry and replacing PARAM,INREL,-1 with PARAM,INREL,-2. [Figure 11-5](#) summarizes the displacements, spcforces, and element stresses output.

The spcforces and stresses are both zeros, similar to the other two runs. As for the displacements, the mass-weighted average motion of all a-set points must be zero. Since the mass is evenly distributed, u_r can be calculated for the T2 components as follows:

$$U_r = \{0.5 \cdot (-8.329169\text{E-}2) + 1.0 \cdot (4.164584\text{E-}2) + 1.0 \cdot (4.164584\text{E-}2) + 0.5 \cdot (-8.329169\text{E-}2)\} \quad \text{Eq. 11-4}$$

See TAN 4002 for further details regarding the Automatic Inertia Relief Method.

```

0
                                D I S P L A C E M E N T   V E C T O R
                                T1      T2      T3      R1      R2      R3
POINT ID.  TYPE      0.0      -8.329169E-02  0.0      0.0      0.0      1.874063E-02
1          2      G      0.0      4.164584E-02  0.0      0.0      0.0      6.246876E-03
          3      G      0.0      4.164584E-02  0.0      0.0      0.0      -6.246876E-03
          4      G      0.0      -8.329169E-02  0.0      0.0      0.0      -1.874063E-02
1 BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 17
0
                                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
                                T1      T2      T3      R1      R2      R3
POINT ID.  TYPE      0.0      6.251190E-13  0.0      0.0      0.0      0.0
1          2      G      0.0      8.820207E-13  0.0      0.0      0.0      0.0
          3      G      0.0      5.138035E-13  0.0      0.0      0.0      0.0
          4      G      0.0      7.279315E-14  0.0      0.0      0.0      0.0
1 BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 18
0
1 BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 19
0
                                S T R E S S E S   I N   B E A M   E L E M E N T S      ( C B E A M )
                                STAT DIST/
ELEMENT-ID  GRID  LENGTH  SXC      SXD      SXE      SXF      S-MAX      S-MIN      M.S.-T      M.S.-C
0          1      1  0.000  0.0      0.0      0.0      0.0      0.0      0.0
          2      2  1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          3      3  0.000  0.0      0.0      0.0      0.0      0.0      0.0
          4      4  1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          5      5  0.000  0.0      0.0      0.0      0.0      0.0      0.0
          6      6  1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          7      7  0.000  0.0      0.0      0.0      0.0      0.0      0.0
          8      8  1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          9      9  0.000  0.0      0.0      0.0      0.0      0.0      0.0
          10     10 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          11     11 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          12     12 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          13     13 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          14     14 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          15     15 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          16     16 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          17     17 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          18     18 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          19     19 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          20     20 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          21     21 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          22     22 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          23     23 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          24     24 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          25     25 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          26     26 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          27     27 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          28     28 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          29     29 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          30     30 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          31     31 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          32     32 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          33     33 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          34     34 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          35     35 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          36     36 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          37     37 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          38     38 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          39     39 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          40     40 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          41     41 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          42     42 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          43     43 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          44     44 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          45     45 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          46     46 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          47     47 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          48     48 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          49     49 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          50     50 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          51     51 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          52     52 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          53     53 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          54     54 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          55     55 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          56     56 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          57     57 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          58     58 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          59     59 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          60     60 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          61     61 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          62     62 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          63     63 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          64     64 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          65     65 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          66     66 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          67     67 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          68     68 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          69     69 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          70     70 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          71     71 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          72     72 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          73     73 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          74     74 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          75     75 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          76     76 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          77     77 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          78     78 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          79     79 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          80     80 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          81     81 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          82     82 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          83     83 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          84     84 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          85     85 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          86     86 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          87     87 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          88     88 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          89     89 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          90     90 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          91     91 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          92     92 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          93     93 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          94     94 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          95     95 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          96     96 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          97     97 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          98     98 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0          99     99 0.000  0.0      0.0      0.0      0.0      0.0      0.0
          100    100 1.000  0.0      0.0      0.0      0.0      0.0      0.0
0

```

Figure 11-5 Abridged Output Using the Automatic Inertia Relief Method

CHAPTER
12

Matrix Operations

- Introduction to Matrix Operations
- Set Definition
- Static Condensation (Guyan Reduction)
- Direct Matrix Input

12.1 Introduction to Matrix Operations

When creating a model in MSC.Nastran, equations are written to simulate the necessary structural properties. These equations are rows and columns in the matrix equations that describe the structural behavior. Six equations are created per grid point; one equation is created per scalar point (SPOINT). The model definition (elements, properties, loads, etc.) is used to generate appropriate row and column entries in the matrices (e.g., stiffness coefficients). Certain data entries (i.e., SPCs, MPCs, ASETs, etc.) cause matrix operations to be performed in the various stages of the solution process. To organize the matrix operations, MSC.Nastran assigns each degree of freedom (DOF) to a displacement set.

Most matrix operations used in a structural analysis involve addition, multiplication, decomposition, forward / backward substitution, partitioning, merging, appending, and / or transforming matrix arrays from one subset to another. All components of motion of a given type form a vector set that is distinguished from other vector sets. A given component of motion can belong to many combined sets but can belong to only one exclusive set.

A thorough understanding of matrix operations in MSC.Nastran is not necessary to perform static analysis; however, some knowledge of it gives you a better appreciation of the solution process that MSC.Nastran undergoes.

The following matrix operations are covered in this chapter:

- Set definition
- Static condensation
- Direct matrix input

12.2 Set Definition

In MSC.Nastran, there are two basic types of sets:

- Combined sets.
- Mutually exclusive sets

Combined sets are formed through the union (combination) of two or more sets. Mutually exclusive sets are important in the solution process because if a DOF is a member of a mutually exclusive set, it cannot be a member of any other mutually exclusive set. The importance of this rule is demonstrated as the sets are described in the rest of this chapter.

The Global Displacement Set

The degrees of freedom of the finite element model include all of the components of displacement at the grid points and scalar points. The DOFs for the grid points are expressed in the local coordinate systems identified in field 7 of the GRID entry. The union of all of the local displacement components at all of the grid points and scalar points is called the global displacement set or g -set. The g -set is the top level set.

The g -set displacement is represented as either u_g or $\{u_g\}$. Its significance is that the mathematical expression for the static equilibrium of the finite element model can be expressed in matrix form as

$$[K_{gg}]\{u_g\} = \{P_g\} \quad \text{Eq. 12-1}$$

where $[K_{gg}]$ is the global stiffness matrix and $\{P_g\}$ is the vector of explicit loads (e.g., loads generated from FORCE entries) or implicit loads (e.g., loads generated from GRAV entries) applied to all of the grid points and scalar points.

The stiffness matrix $[K_{gg}]$ is formed by generating and assembling the stiffness matrices for all of the elements. The load vector is formed by assembling all of the implicitly and explicitly defined load components within a given loading condition. Loads defined within elements are automatically transferred to the appropriate grid points in a consistent manner.

The number of terms in the $\{u_g\}$ and $\{P_g\}$ vectors is equal to six times the number of grid points plus the number of scalar points. $[K_{gg}]$ is a real and symmetric matrix.

Subsets of the Global Displacement Set

The g -set represents an unconstrained set of structural equations. $[K_{gg}]$ is usually singular; therefore, the equations of equilibrium cannot be solved in the form shown by [Eq. 12-1](#).

The next step in the solution process is to partition the g -set into two subsets-the m -set and the n -set. The dependent DOFs of all multipoint constraint relations (MPCs, RBEs, etc.) define the m -set. During the solution process, the contributions of these m -set DOFs are condensed into the set of independent DOFs, the n -set. The n -set represents all of the

independent DOFs that remain after the dependent DOFs are removed from the active set of equations. The *m*-set represents the DOFs that are linearly dependent upon the *n*-set DOFs.

Using the *n*-set, SPCs are then applied to the independent equations to further partition the equations. Degrees of freedom identified by PARAM,AUTOSPC and / or SPCi entries are assigned to the *sb*-set. See “**Constraints**” on page 257 for more details regarding PARAM,AUTOSPC. Permanent constraints, those DOFs specified in field 8 of the GRID entry, are assigned to the *sg*-set. The union of the *sb*- and *sg*-sets compose the *s*-set. After reducing the *n*-set by applying the *s*-set constraints, the *f*-set remains. The *f*-set represents the “free” DOFs of the structure. If applied constraints are properly specified, then the *f*-set equations represent a statically stable solution (i.e, a nonsingular stiffness matrix).

The *f*-set can be further reduced by partitioning it into the *o*- and *a*-set. This operation is known as static condensation (Guyan reduction) and is discussed in detail in “**Static Condensation (Guyan Reduction)**” on page 462. By default, all DOFs in the *f*-set are assigned to the *a*-set in other words, the *o*-set is null, by default.

In the case where you want to analyze an unsupported structure (such as an aircraft in flight), you can put some of the *a*-set degrees of freedom into a special set--the reference set or *r*-set--that is used to aid in the calculation of inertia forces and to balance the applied loads. The SUPORT Bulk Data entry is used for this purpose. In this case, the remaining degrees of freedom in the *a*-set are placed into the *l*-set. The use of the SUPORT entry for the automatic calculation of inertia loads in static analysis is described in “**Inertia Relief**” on page 435.

In order for the SUPORT entry to work, the finite element model must have a mass matrix. The SUPORT entry must be specified for all six DOFs on a single grid point. (Note that the word “SUPORT” is spelled with one “P”.)

Another important use of the SUPORT entry is to provide a reference point in the computation of rigid body modes in dynamic analysis (see the **MSC.Nastran Basic Dynamics User’s Guide**). **Table 12-2** illustrates the basic partitioning operations. It is a representation of the basic sets, set partitions, and set combinations used in MSC.Nastran static analysis.

Table 12-1 Basic Set Partitioning Operations

$g\text{-set} - m\text{-set} = n\text{-set}$
$n\text{-set} - s\text{-set} = f\text{-set}$
$f\text{-set} - o\text{-set} = a\text{-set}$
$a\text{-set} - r\text{-set} = l\text{-set}$

In the above table, the *m*-, *s*-, *o*-, and *r*-sets form the mutually exclusive sets. Physically, the mutually exclusive set partitioning ensures that operations cannot be performed on a DOF that is no longer active. For example, if you apply an SPC to a DOF that is a dependent degree of freedom on an RBAR, a User Fatal Message (UFM) 2101A is issued. Using an SPC entry moves a DOF to the *sb*-set, but this cannot occur if the same DOF is already a member

of the m -set. Both the m -set and sb -set are mutually exclusive. **Table 12-2** represents the commonly used mutually exclusive sets in MSC.Nastran. **Table 12-3** and **Figure 12-1** represent the combined sets used in MSC.Nastran.

Table 12-2 Mutually Exclusive Sets in MSC.Nastran

Set Name	Description
m	Degrees of freedom eliminated by multipoint constraints.
sb^*	Degrees of freedom eliminated by single-point constraints that are included in a boundary condition changes (SPC and SPC1 entries) and by the automatic SPC feature (PARAM,AUTOSPC).
sg^*	Degrees of freedom eliminated by permanent single-point constraints that are listed on the grid point Bulk Data entries.
o	Degrees of freedom omitted by structural matrix partitioning (see “ Static Condensation (Guyan Reduction) ” on page 462).
r	Reference DOFs used to determine free body motion. Used for inertia relief in static and rigid body modes calculations in dynamic analysis.
l^{**}	Degrees of freedom in which static analysis is performed.

*Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees of freedom may exist in both sets simultaneously. These sets are exclusive, however, from the other mutually exclusive sets.

**The l -set is not truly a mutually exclusive set; although it is conceptually viewed as a mutually exclusive set in static analysis. The definition of the l -set is different in dynamic analysis--it is a combination of the b - and c -set. The b - and c -set are not used in static analysis.

Table 12-3 Combined Sets in MSC.Nastran (+ Indicates the Union of Two Sets)

$f = a + o$	Unconstrained (free) structural DOFs.
$s = sb + sg$	All DOFs eliminated by single-point constraints.
$n = f + s$	All DOFs not constrained by multipoint constraints.
$g = n + m$	All structural DOFs including scalar points.

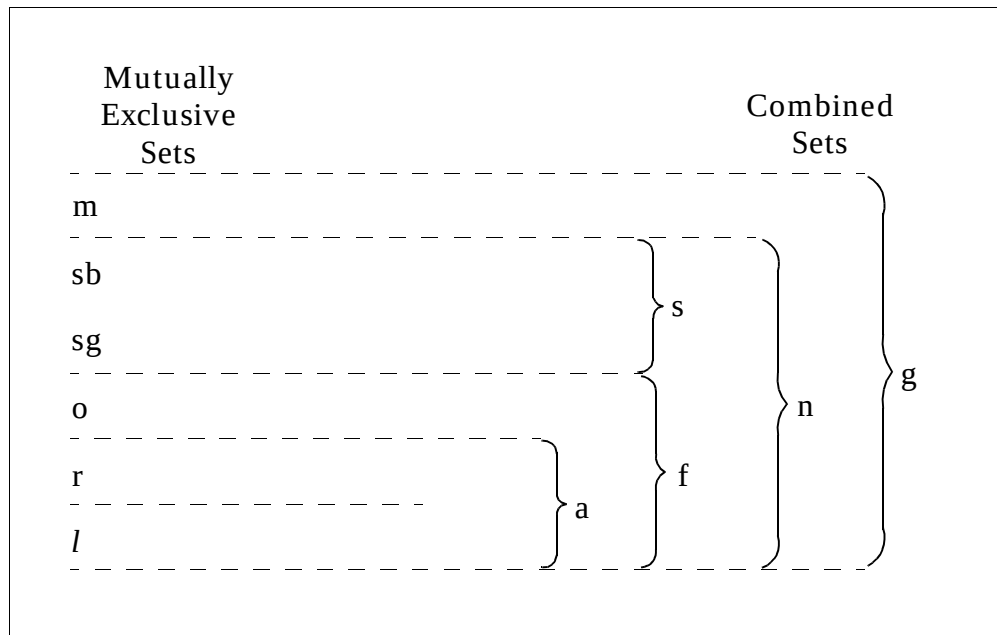


Figure 12-1 Combined Sets Formed from Mutually Exclusive Sets

When a particular set above has no DOFs associated with it, it is a null set. If this set exists, a partition is then applied, and the DOFs are moved to the subsequent partition. **Table 12-4** provides a list of possible Bulk Data entries, which generate a corresponding subset.

Table 12-4 Subsets Versus Bulk Data Entries

Set	Associated Bulk Data Entries
u_m	MPC, MPCADD, RBAR, RBE1, RBE2, RBE3, RROD, RSPLINE, RTRPLT
u_s	GRID, GRDSET, SPC, SPC1, SCPADD, (PARAM,AUTOSPC,YES) *
u_o	OMIT, OMIT1
u_r	SUPPORT
u_a	ASET, ASET1

*The PARAM,AUTOSPC,YES Bulk Data entry enables MSC.Nastran to change the membership of a degree of freedom in the f -set to membership in the s -set, based on the criteria described in “**Constraints**” on page 257.

The final set of degrees of freedom remaining after the above exclusive sets are eliminated is called the l -set (degrees of freedom left over). The static solution is performed on the u_l set. The reduced equations of equilibrium are expressed in the matrix form

$$[K_{ll}]\{u_l\} = \{P_l\} \quad \text{Eq. 12-2}$$

The reduced stiffness matrix $[K_{ll}]$ is symmetric and is also nonsingular if all mechanisms and rigid body motion are removed by the partitioning of the m -set, s -set, and r -set. **Eq. 12-2** is solved for $\{u_l\}$ by Gaussian elimination (decomposition and forward-backward

substitution). All of the other subsets of vector--namely u_r , u_o , u_s , and u_m --can be obtained subsequently by successive substitution into their defining equations. By default, all f -set degrees of freedom are assigned to the a -set. If you do not have a rigid body support (r -set) in your model, then the a -set is identical to your l -set.

The set membership of all of the degrees of freedom can be output by means of the PARAM,USETPRT,x Bulk Data entry. This entry should be used with caution since a large model may produce a substantial amount of output using this parameter. See the [*MSC.Nastran Quick Reference Guide*](#) for a complete description of all the options available for PARAM,USETPRT,x.

12.3 Static Condensation (Guyan Reduction)

The terms static condensation and Guyan reduction represent the same matrix operations and are used interchangeably. Guyan Reduction involves a reformulation of the stiffness and mass matrices based on partitions of the stiffness matrix. If you start with the stiffness equation in terms of the set of the unconstrained (free) structural coordinates, you have

$$[K_{ff}]\{u_f\} = \{P_f\} \quad \text{Eq. 12-3}$$

Partitioning the free degrees of freedom into two subsets of the f -set, you obtain

$$\{u_f\} = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} \quad \text{Eq. 12-4}$$

where:

u_a = the a -set variables

u_o = the omitted set of variables, which are removed by static condensation

Rewriting the static equation for u_f in terms of o -set and a -set partitions, you obtain

$$[K_{ff}][u_f] = \begin{bmatrix} \overline{K_{aa}} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} \overline{P_a} \\ P_o \end{Bmatrix} \quad \text{Eq. 12-5}$$

Expanding the bottom equation, you arrive at

$$\{u_o\} = [G_{oa}]\{u_a\} + \{u_{oo}\} \quad \text{Eq. 12-6}$$

where:

$$[G_{oa}] = -[K_{oo}]^{-1}[K_{oa}]$$

$$\{u_{oo}\} = [K_{oo}]^{-1}[P_o]$$

Expanding the top part of [Eq. 12-5](#) and substituting u_o with [Eq. 12-6](#), you arrive at the equation

$$\{u_a\} = [K_{aa}]^{-1}\{P_a\} \quad \text{Eq. 12-7}$$

where:

$$[K_{aa}] = [\overline{K_{aa}} + K_{ao}G_{oa}]$$

$$\{P_a\} = [\overline{P_a} + G_{oa}^T P_o]$$

The solution to the f -set degrees of freedom is obtained in three phases. The first phase is to reduce the f -set to the a -set using Guyan reduction. The second phase is to solve for $\{u_a\}$ in Eq. 12-7, which is the analysis set. Once this is done, $\{u_o\}$ can then be solved by Eq. 12-6, which is the omitted set. The conventional method involves solving Eq. 12-3 in a single step.

In static analysis, the results using static condensation are numerically exact. The partitioned solution merely changes the order of the operations of the unpartitioned solution. Guyan reduction is, however, not generally recommended for static analysis. This recommendation stems from the cost of the Guyan reduction process since it creates reduced matrices $[K_{aa}]$ and $[M_{aa}]$, which are typically smaller but denser (i.e., matrix bandwidth is destroyed). Additional decomposition of the matrix $[K_{oo}]$ is also required. In other words, even though the solution size is smaller, the solution time may take longer and the disk space requirement may be larger due to a denser matrix. Furthermore, it requires additional interaction on your part. However, some understanding of the concept of Guyan reduction in static analysis is quite helpful if you ever decide to apply this feature in dynamic, cyclic symmetry, or superelement analyses.

Guyan reduction has special applications in dynamic analysis. In dynamic analysis, the reduction is approximate; the term $[u_{oo}]$ is ignored in this case. The reduction is based solely on static transformation and is exact provided that no loads are applied to the o -set degrees of freedom.

User Interface

To utilize static condensation, you can choose either the ASET/ASET1 or the OMIT/OMIT1 entries. With these entries you should specify only the a -set (with ASET/ASET1 entries) or o -set (with OMIT/OMIT1 entries) degrees of freedom. The unspecified remaining f -set DOFs are automatically placed in the complementary set. However, if you specify both the a -set and o -set DOFs, then the unspecified remaining f -set DOFs are automatically placed in the o -set. The same DOF cannot be specified on both the a -set and o -set; otherwise, the job fails with UFM 2101A.

Since the reduction process is performed on an individual degree of freedom, it is possible to have some of the degrees of freedom at a grid point in the a -set and other degrees of freedom at a grid point in one of the other mutually exclusive sets. No additional user input is required.

Format for ASET

ASET can be used in conjunction with ASET1. The formats for the ASET/ASET1 are as follows:

1	2	3	4	5	6	7	8	9	10
ASET	G1	C1	G2	C2	G3	C3	G4	C4	

Field	Contents
Gi	The i-th grid or scalar point identification number
Ci	Component number(s) for the i-th grid or scalar point

Example:

ASET	5	3	30	345	8	12345			

For the above example, component 3 of grid point 5, components 3, 4, and 5 of grid point 30, and components 1,2,3,4, and 5 of grid point 8 are assigned to the *a*-set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (*o*-set).

Format for ASET1

	1	2	3	4	5	6	7	8	9	10
ASET1	C	G1	G2	G3	G4	G5	G6	G7		
	G8	G9	G10	G11	-etc.-					

Field	Contents
C	Component number(s).
Gi	The i-th grid or scalar point identification number.

Example:

ASET 1	345	3	4	1	16	8	9	5	
	13	20	25						

For the above example, components 3,4, and 5 of grid points 1, 3, 4, 5, 8, 9, 13, 16, 20, and 25 are assigned to the *a*-set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (*o*-set).

Alternate Format for ASET1

	1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	"THRU"	ID2						

Field	Contents
C	Component number(s).
ID1	The first grid or scalar point identification number in the range.
ID2	The last grid or scalar point identification number in the range.

Example:

ASET1	123456	10	THRU	909					

For this example, components 1,2,3,4,5, and 6 of grid point 10 through 909 are assigned to the *a*-set. All grid points between grid points 10 and 909 need not exist as long as at least one of them exists. All other degrees of freedom that are outside of grid points 10 through 909 and not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (*o*-set).

Format for OMIT

OMIT can be used in conjunction with OMIT1. The formats for the OMIT / OMIT1 are as follows:

1	2	3	4	5	6	7	8	9	10
OMIT	G1	C1	G2	C2	G3	C3	G4	C4	

Field**Contents**

G_i The *i*-th grid or scalar point identification number.

C_i Component number(s) for the *i*-th grid or scalar point.

Example:

OMIT	5	3	30	345	8	123456			

For the above example, component 3 of grid point 5, components 3, 4, and 5 of grid point 30, and components 1,2,3,4,5, and 6 of grid point 8 are assigned to the *o*-set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (*a*-set).

Format for OMIT1

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	G11	-etc.-				

Field**Contents**

C Component number(s).

G_i The *i*-th grid or scalar point identification number.

Example:

OMIT1	345	3	4	1	16	8	9	5	
	13	20	25						

For the above example, components 3, 4, and 5 of grid points 1, 3, 4, 5, 8, 9, 13, 16, 20, and 25 are assigned to the *o*-set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (*a*-set).

Alternate Format for OMIT1

	1	2	3	4	5	6	7	8	9	10
OMIT1	C	ID1	"THRU"	ID2						

Field	Contents
C	Component number(s).
ID1	The first grid or scalar point identification number in the range.
ID2	The last grid or scalar point identification number in the range.

Example:

OMIT1	123456	10	THRU	909					

For this example, components 1,2,3,4,5, and 6 of grid points 10 through 909 are assigned to the *o*-set. All grid points between grid points 10 and 909 need not exist as long as at least one of them exists. All other degrees of freedom that are outside of grid points 10 through 909 and not otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (*a*-set).

The choice of whether to use the ASET / ASET1 or OMIT / OMIT1 is really a matter of convenience.

Consider a simple example using Guyan reduction. The cantilever beam model in [Figure 12-2](#) is used for this purpose. The common Bulk Data entries for this model are contained in [Listing 12-1](#).

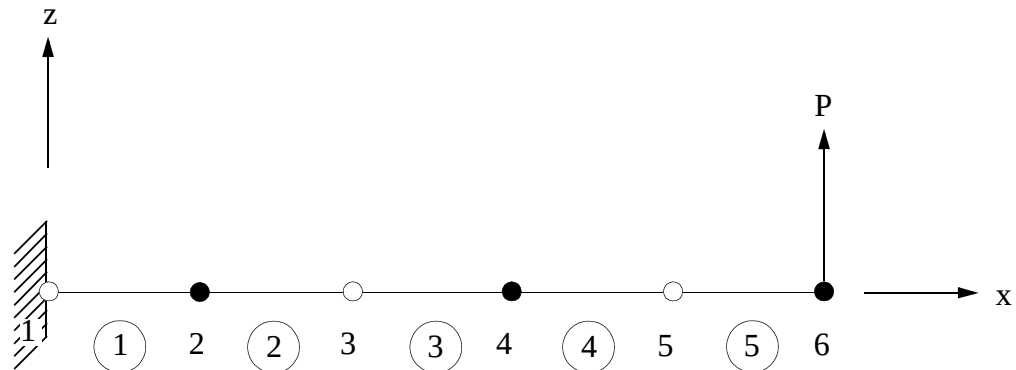


Figure 12-2 Cantilever Beam Using Guyan Reduction

Listing 12-1 Input File for Common Bulk Data Entries

```

$
$  filename - bulkm.dat
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
CBAR    5      1      5      6      10
FORCE  10      6              100.    0.      0.      1.
$
GRID    1              0.      0.      0.
GRID    2              1.      0.      0.
GRID    3              2.      0.      0.
GRID    4              3.      0.      0.
GRID    5              4.      0.      0.
GRID    6              5.      0.      0.
GRID   10              0.      0.     10.
MAT1    1      7.1+10      .33     2700.
PBAR    1      1      2.654-3  5.869-7  5.869-7  9.9-7
SPC1    10      123456  1
$

```

The first run is a conventional run without any reduction ([Listing 12-2](#)). Since Guyan reduction is not used, then, by default, all degrees of freedom that are not otherwise constrained are assigned to the *a*-set. In other words, degrees of freedom 1 through 6 for grid points 2 through 6 are assigned to the *a*-set. Components 1 through 6 of grid point 1 are assigned to the *s*-set due to the SPC Case Control command and SPC1 Bulk Data entry.

Listing 12-2 Input File for Using Conventional Analysis

```

$
$  filename - full.dat
$
id bar aset
sol 101
time 10
cend
$
spc = 10
disp = all
$
subcase 1
load = 10
$
begin bulk
$
include 'bulkm.dat'
$
enddata

```

The second run uses Guyan reduction by specifying degrees of freedom 1 through 6 at grid points 2, 4, and 6 to the *a*-set by using the ASET1 entries ([Listing 12-3](#)). Due to this action, the remaining unconstrained degrees of freedom (DOF 1 through 6 for grid points 3 and 5) are assigned to the *o*-set. Components 1 through 6 of grid point 1 are assigned to the *s*-set due to the SPC Case Control command and SPC1 Bulk Data entry.

Listing 12-3 Input File Specifying A-Set DOFs

```
$
$  filename - aset.dat
$
id bar aset
sol 101
time 10
cend
$
title =
subtitle =
spc = 10
disp = all
$
subcase 1
load = 10
$
begin bulk
$
aset1,123456,2,4,6
$
include 'bulkm.dat'
$
enddata
```

The third run is similar to the second run except the omitted degrees of freedom are specified rather than the analysis degrees of freedom ([Listing 12-4](#)). Degrees of freedom 1 through 6 for grid points 3 and 5 are assigned to the *o*-set by using the OMIT1 entries. Due to this action, degrees of freedom 1 through 6 for grid points 2, 4, and 6 are automatically assigned to the *a*-set. Components 1 through 6 of grid point 1 are assigned to the *s*-set due to the SPC Case Control command and SPC1 Bulk Data entry.

Listing 12-4 Input File Specifying O-Set DOFs

```
$  
$  filename - omit.dat  
$  
id bar omit  
sol 101  
time 10  
cend  
$  
title =  
subtitle =  
spc = 10  
disp = all  
$  
subcase 1  
load = 10  
$  
begin bulk  
$  
omit1,123456,3,5  
$  
include 'bulkm.dat'  
$  
enddata
```

Since Guyan reduction is numerically exact in static analysis, all three runs should yield the same results. **Figure 12-3** summarizes the results from the three runs—they are identical, as expected.

***** PARTIAL OUTPUT FROM MODEL WITHOUT USING GUYAN REDUCTION *****

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
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0

***** PARTIAL OUTPUT FROM MODEL WITH GUYAN REDUCTION USING ASET ENTRIES *****

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
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0

***** PARTIAL OUTPUT FROM MODEL WITH GUYAN REDUCTION USING OMIT ENTRIES *****

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
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0

Figure 12-3 Results Comparison With and Without Guyan Reduction

Note that the only differences in the three input files are the ASET1/OMIT1 entries. The common Bulk Data entries are inserted in the run stream by using the INCLUDE statement. Refer to “[Database Management](#)” on page 541 for further details regarding the use of the INCLUDE statement.

12.4 Direct Matrix Input

The finite element approach simulates the structural properties with mathematical equations written in matrix format. Once you provide the grid point locations, element connectivities, cross-sectional properties, material properties, applied loads, and boundary conditions, MSC.Nastran then automatically generates the appropriate structural matrices. The structural behavior is then obtained by solving these equations.

If these structural matrices are available externally, you can input these matrices directly without providing all the modeling information. Normally this is not a recommended procedure since it requires additional effort on your part. However, there are occasions where the availability of this feature is very useful and in some cases is extremely crucial. Some possible applications are listed below:

1. Suppose you are a subcontractor to a classified project. The substructure that you are analyzing is attached to the main structure built by the primary contractor. The flexibility of this main structure is crucial to the response of your component, but the geometry of the main structure is classified. The main contractor, however, can provide you with the stiffness matrix of the classified structure. By reading in this stiffness matrix and adding it to your MSC.Nastran model, you can account for the flexibility of the attached structure without compromising the security. The stiffness matrix is the inverse of the flexibility matrix.
2. Suppose you are investigating a series of design options on a component attached to an aircraft bulkhead. Your component consists of 500 DOFs and the aircraft model consists of 100,000 DOFs. The flexibility of the backup structure is somewhat important. You can certainly analyze your component by including the full aircraft model (100,500 DOFs). On the other hand, if the flexibility at the attachment points on the aircraft can be measured experimentally, then you can add the experimental backup structure stiffness to your component without including the whole aircraft model. The experimental backup structure stiffness matrix is the inverse of the measured flexibility matrix. This way your model size remains at 500 DOFs, and you still have a good approximation of the backup structure stiffness.
3. The same concept can be applied to a component attached to a test fixture. The stiffness of the fixture at the attachment locations can be read in as a stiffness matrix. Once again, the experimental test fixture stiffness matrix at the attachment points is the inverse of the measured flexibility at these points.

The above are just a few of the many possible applications. You can probably think of other usages. There are several ways that these matrices can be read in; some of these are DMIG, GENEL, and INPUTT4. Only DMIG is covered in this section. GENEL is covered in detail in “[MSC.Nastran Elements](#)” on page 65. The DMIG and the INPUTT4 options offer alternate methods for inputting large matrices. Note that INPUTT4 provides more precision than the DMIG input; the DMIG yields more precision than the GENEL on a shortword machine. See the [MSC.Nastran 2005 DMAP Programmer's Guide](#) for more details on INPUTT4.

DMIG

The DMIG feature can be used to directly input a stiffness, mass, or loading matrix to the grid and / or scalar points in static analysis. These matrices are referenced in terms of their external grid point IDs and DOFs. The symbols for these *g*-type matrices in standard mathematical format are

$$[K_{gg}^2], [M_{gg}^2], \text{ and } [P_g^2]$$

All three matrices must be real. Furthermore,

$$[K_{gg}^2] \text{ and } [M_{gg}^2]$$

must also be symmetric. These matrices are implemented on the *g*-set level. In other words, these terms are added to the finite element model at the specified DOFs prior to the application of constraints. This section only includes features of the DMIG that are used in static analysis. The DMIG has additional features that can be used in dynamic analysis, but they are not discussed in this user's guide.

DMIG Bulk Data User Interface

In the Bulk Data Section, the DMIG matrix is defined by a single DMIG header entry followed by a series of DMIG data entries. Each of these DMIG data entries contains a column containing the nonzero elements for the particular matrix that you are entering.

Header Entry Format

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT			NCOL	

Field	Contents
NAME	Name of the matrix. It consists of one to eight alphanumeric characters, the first of which must be alphabetic. (Required)
IFO	Form of matrix input: Integer, required 1 = square 9 or 2 = rectangle 6 = symmetric
TIN	Type of matrix that is input: Integer, required 1 = real, single precision 2 = real, double precision
TOUT	Type of matrix that is created: Integer 0 = set by your machine precision (default) 1 = real, single precision

2 = real, double precision

NCOL Number of columns in a rectangular matrix. Used only for IFO = 9.
(Integer > 0).

Remarks:

1. One header entry is required for each input matrix.
2. The matrix name used in field 2 must be unique.
3. Field 3 of the header entry must contain an integer 0.
4. The default value of 0 for TOUT is recommended.
5. If the input matrix is rectangular, the use of IFO = 9 is recommended instead of IFO = 2.

Column Data Entry Format

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	GJ	CJ		G1	C1	A1		
	G2	C2	A2		-etc.-				

Field	Contents
NAME	This is the same name that is used with its corresponding header entry as described above.
GJ	The external grid or scalar point identification number for the column. (Integer > 0).
CJ	The component number for the GJ grid point. ($0 < \text{Integer} \leq 6$ if GJ is a grid point; blank or zero if GJ is a scalar point).
Gi	The external grid or scalar point identification number for the row. (Integer > 0).
Ci	The component number for the Gi grid point. ($0 < \text{Integer} \leq 6$ if Gi is a grid point; blank or zero if Gi is a scalar point).
Ai	The value of the matrix term.

Remarks:

1. Each non-null column (a column with at least one nonzero term) starts with a GJ, CJ pair. The entries (Gi, Ci pairs) for each row of that column follow. The number of Gi, Ci pairs equals the number of nonzero terms in that column. The terms may be input in arbitrary order. High precision is required when using DMIG; therefore, in general, the use of large field input is recommended. Each nonnull column of your matrix makes up a DMIG column data entry. Therefore, the total number of DMIG entries is equal to the number of nonnull columns plus one. The extra entry is the DMIG header entry.

2. For symmetric matrices (IFO = 6), a given off-diagonal term may be input either below or above the diagonal. While the upper and lower triangle terms may be mixed, a fatal error results if the corresponding symmetric element is input both below and above the diagonal. However, to keep it simple for yourself, it is recommended that for a given matrix, you should decide to input the terms either completely below the diagonal or above the diagonal, but not both. The symmetric option is generally used for the mass and stiffness matrices.
3. Rectangular matrices can be input using the option IFO = 9. This option is generally used to input load matrices.

DMIG Case Control User Interface

In order to include these matrices, the Case Control Section must contain the appropriate K2GG, M2GG, or P2G command.

Examples

K2GG = mystif

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “mystif”, to the g -set stiffness matrix.

M2GG = yourmass

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “yourmass”, to the g -set mass matrix.

P2G = hisload

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “hisload”, to the g -set load matrix.

DMIG Examples

Consider a few examples using the DMIG features. The cantilever beam model shown in [Figure 12-4](#) is used for this purpose.

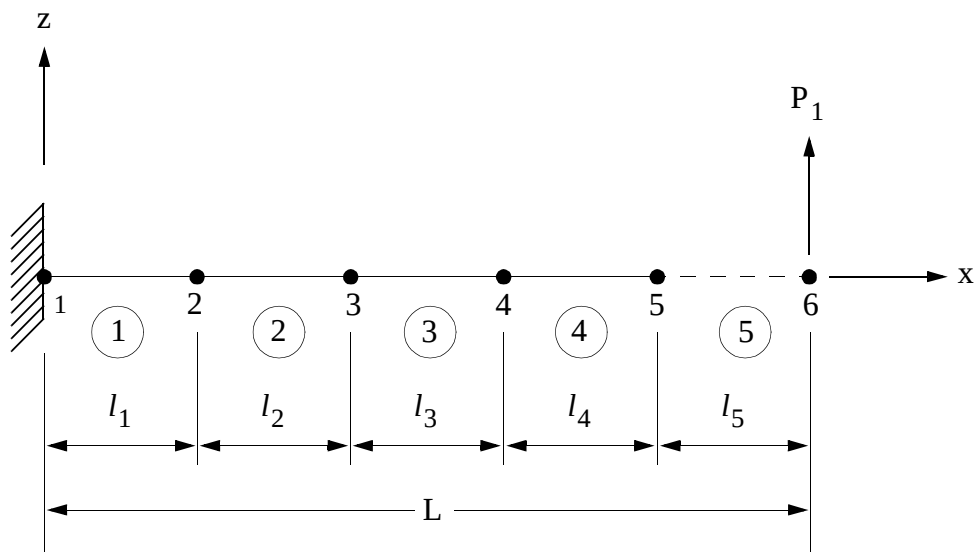


Figure 12-4 Planer Cantilever Beam

Stiffness Matrix Using DMIG

The first step is to analyze this problem using the conventional method with two loading conditions--a vertical tip load and a gravity load. [Listing 12-5](#) contains the input file for this run. Note that for illustrative purposes, consider a planar problem (xz plane). However, this same concept is applicable to a general three-dimensional structure.

Listing 12-5 Input File for Conventional Analysis

```

$
$  FILENAME - DMIGFULL.DAT
$
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = PLANAR CANTILEVER BEAM
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
SUBCASE 2
LABEL = GRAVITY LOAD
LOAD = 20
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
CBAR    5      1      5      6      10
FORCE   10      6      100.    0.    0.    1.
GRAV    20      9.8    0.    0.    1.
GRID    1      0.    0.    0.    246
GRID    2      1.    0.    0.    246
GRID    3      2.    0.    0.    246
GRID    4      3.    0.    0.    246
GRID    5      4.    0.    0.    246
GRID    6      5.    0.    0.    246
GRID   10      0.    0.    10.    123456
MAT1     1      7.1+10    .33    2700.
PBAR     1      1      2.654-3  5.869-7
SPC1    10      123456  1
$
ENDDATA

```

The next run removes CBAR element number 5 from the input file and uses the DMIG feature to directly read in the bar element stiffness matrix. The bar element stiffness matrix for a planar element and ignoring the axial stiffness is as follows:

$$[K]_e = \frac{2EI}{l^3} \begin{bmatrix} 6 & -3l & -6 & -3l \\ -3l & 2l^2 & 3l & l^2 \\ -6 & 3l & 6 & 3l \\ -3l & l^2 & 3l & 2l^2 \end{bmatrix}$$

Since the matrix is symmetric, either the lower or upper triangular portion of the matrix must therefore be provided. For $E = 7.1 \cdot 10^{10} \text{ N/m}^2$, $I = 5.869 \cdot 10^{-7} \text{ m}^4$, and $l_5 = 1 \text{ m}$, the stiffness matrix for CBAR element number 5 is as follows:

$$[K]_e = \begin{matrix} & \begin{matrix} 5, 3 & 5, 5 & 6, 3 & 6, 5 \end{matrix} \\ \begin{matrix} 5, 3 \\ 5, 5 \\ 6, 3 \\ 6, 5 \end{matrix} & \begin{bmatrix} 500038.3 & & & \\ -250019.4 & 166679.6 & & \text{SYM} \\ -500038.8 & 250019.4 & 500038.8 & \\ -250019.4 & 83339.8 & 250019.4 & 166679.6 \end{bmatrix} \end{matrix} \quad \text{Eq. 12-8}$$

In this case, you chose UGSTIF as the name of the input stiffness for bar element number 5. Therefore, in order to bring in this stiffness matrix and add it to the global stiffness matrix, the Case Control command K2GG = UGSTIF is required. Note these stiffness terms are additions to the global stiffness matrix at that location and are not a replacement of the stiffness terms at that location. In the Bulk Data Section, five DMIG entries are required—one for the header entry and four for the data column entries—since there are four non-null columns in the above matrix.

For the header entry, the same name UGSTIF must be used to match the name called out in the Case Control Section. The third field is 0, which must be the value used for the header entry. The fourth field (IFO) is set to 6 to denote a symmetric matrix input. The fifth field (TIN) is set to 1 to denote that the matrix is provided as real, single precision.

Once again, the terms in the matrix are referenced in terms of their external grid IDs when using the DMIG feature. Physically, each term in a particular column of the stiffness matrix (k_{ij}) represents the induced reactive load in the i-th degree of freedom due to a unit displacement in the j-th direction with all other displacement degrees of freedom held to zero. Since the matrix is symmetric, only the lower triangular portion of the matrix is read.

The first DMIG data column entry reads the first column of the above matrix. Field 2 of this DMIG entry must have the same name UGSTIF as referenced by the Case Control K2GG = UGSTIF command. Fields 3 and 4 of this entry identify this column in terms of its external grid ID and corresponding degree of freedom, respectively. In this case, it is grid point number 5, degree of freedom 3 (z-translation at grid number 5).

Once this column is defined and by following the format description as described in the section for column data entry format, you can then input the four terms in this column row by row. These four terms are defined by sets of three fields. They are the external grid point ID number, corresponding degree of freedom, and the actual matrix term, respectively. The first row of column one is defined by external grid point ID 5, degree of freedom 3 (z-translation at grid point number 5) with a stiffness value of 500039. The

second row of column one is defined by external grid point ID 5, degree of freedom 5 (y-rotation at grid point number 5) with a stiffness value of -250019. The third row of column one is defined by external grid point ID 6, degree of freedom 3 (z-translation at grid point number 6) with a stiffness value of -500039. The fourth row of column one is defined by external grid point ID 6, degree of freedom 5 (y-rotation at grid point number 6) with a stiffness value of -250019.

The next DMIG entry reads the second column of the above matrix starting with the diagonal term. Fields 3 and 4 of this entry identify this column in terms of its external grid point ID and corresponding degree of freedom, respectively. In this case, it is grid number 5, degree of freedom 5 (y-rotation at grid point number 5). The rest of the procedure is similar to that of column one with the exception that only three rows need to be read due to symmetry (rows two through four).

The next two DMIG entries read columns three and four of the stiffness matrix, respectively. Note that due to symmetry, one less row needs to be read for each additional column.

The input file using DMIG is contained in [Listing 12-6](#). The results as compared with the conventional analysis are tabulated in the first two sections of [Figure 12-5](#). The comparison is performed for Subcase 1 with a vertical load applied to the tip of the cantilever beam. In this case, the results are accurate to within three to four digits.

As mentioned previously, a high degree of precision should be retained when inputting a matrix with the DMIG method. In this case, since the small-field input format is used, the maximum number of characters that can be used are eight (including sign and decimal place). The same job was then rerun for the DMIG input with additional significant digits added using double-precision and large-field format ([Listing 12-7](#)). In this case, the results are identical by comparing the results of the first and last sections of [Figure 12-5](#). Note that even the double-precision input on short-word machines does not contain all of the significant digits of the machine.

Listing 12-6 Input File for Reading a Stiffness Matrix Using DMIG with Small-Field Format

```

$
$  FILENAME - DMIGSTFS.DAT
$
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = PLANER CANTILEVER BEAM
SUBTITLE = USE DMIG TO BRING IN STIFFNESS MATRIX OF ELEMENT # 5 USING SMALL FIELD
K2GG = UGSTIF
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
$
$  DMIG HEADER ENTRY
$
DMIG    UGSTIF  0      6      1
$
$  DMIG DATA COLUMN ENTRIES
$
DMIG    UGSTIF  5      3      5      3      500039.      +000001
++0000015      5      -250019.      6      3      -500039.      +000002
++0000026      5      -250019.      +000003
$
DMIG    UGSTIF  5      5      5      5      166680.      +000004
++0000046      3      250019.      6      5      83340.      +000005
$
DMIG    UGSTIF  6      3      6      3      500039.      +000006
++0000066      5      250019.      +000007
$
DMIG    UGSTIF  6      5      6      5      166680.
$
FORCE   10      6      100.      0.      0.      1.
GRAV    20      9.8      0.      0.      1.
GRID    1      0.      0.      0.      246
GRID    2      1.      0.      0.      246
GRID    3      2.      0.      0.      246
GRID    4      3.      0.      0.      246
GRID    5      4.      0.      0.      246
GRID    6      5.      0.      0.      246
GRID    10     0.      0.      10.      123456
MAT1    1      7.1+10      .33      2700.
PBAR    1      1      2.654-3  5.869-7
SPC1    10     123456  1
$
ENDDATA

```

Listing 12-7 Input File for Reading a Stiffness Matrix with Large-Field Format

```

$
$   FILENAME - DMIGSTFL.DAT
$
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = PLANER CANTILEVER BEAM
SUBTITLE = DMIG TO BRING IN STIFFNESS MATRIX FOR ELEMENT # 5 USING LARGE FIELD
K2GG = UGSTIF
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
BEGIN BULK
$
CBAR   1      1      1      2      10
CBAR   2      1      2      3      10
CBAR   3      1      3      4      10
CBAR   4      1      4      5      10
$
$   DMIG HEADER ENTRY
$
DMIG   UGSTIF  0      6      2
$
$   DMIG DATA COLUMN ENTRIES
$
DMIG*   UGSTIF      5      3      *A
*A      5      3      5.000388 D+5      *B
*B      5      5      -2.500194 D+5      *C
*C      6      3      -5.000388 D+5      *D
*D      6      5      -2.500194 D+5
$
DMIG*   UGSTIF      5      5      *A2
*A2     5      5      1.666796 D+5      *B2
*B2     6      3      2.500194 D+5      *C2
*C2     6      5      8.33398 D+4
$
DMIG*   UGSTIF      6      3      *A3
*A3     6      3      5.000388 D+5      *B3
*B3     6      5      2.500194 D+5
$
DMIG*   UGSTIF      6      5      *A4
*A4     6      5      1.666796 D+5
$
FORCE  10      6      100.    0.    0.    1.
GRAV   20      9.8    0.    0.    1.
GRID   1      0.    0.    0.    246
GRID   2      1.    0.    0.    246
GRID   3      2.    0.    0.    246
GRID   4      3.    0.    0.    246
GRID   5      4.    0.    0.    246
GRID   6      5.    0.    0.    246
GRID  10      0.    0.    10.    123456
MAT1    1      7.1+10    .33    2700.
PBAR    1      1      2.654-3  5.869-7
SPC1    10      123456  1
$
ENDDATA

```

***** PARTIAL OUTPUT FROM DMIGFULL.F06 - FULL MODEL *****								
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	
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

***** PARTIAL OUTPUT FROM DMIGSTIFS.F06 - DMIG TO INPUT STIFFNESS FOR ELEM # 5 USING SMALL FIELD								
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	
2	G	.0	.0	5.598505E-03	.0	-1.079704E-02	.0	
3	G	.0	.0	2.079415E-02	.0	-1.919427E-02	.0	
4	G	.0	.0	4.318711E-02	.0	-2.519168E-02	.0	
5	G	.0	.0	7.037757E-02	.0	-2.878928E-02	.0	
6	G	.0	.0	9.996621E-02	.0	-2.998813E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

***** PARTIAL OUTPUT FROM DMIGSTIFL.F06 - DMIG TO INPUT STIFFNESS FOR ELEM # 5 USING LARGE FIELD								
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	
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

Figure 12-5 Results Comparison Between the Conventional Method Versus DMIG

Mass Matrix Using DMIG

So far the discussion has focused on reading a stiffness matrix using DMIG. The mass matrix can be included in a similar manner as the stiffness matrix with the following exceptions:

1. The command M2GG = “name”, instead of K2GG = “name”, is used in the Case Control section. The “name” is the name of the mass matrix referenced in field two of the DMIG Bulk Data entries.
2. The matrix defined in the DMIG entries represents the mass matrix terms instead of the stiffness matrix terms.

Load Matrix Using DMIG

The load vectors can also be read using the DMIG feature if desired, although this is not a commonly used feature. It is similar to the format for reading in the stiffness or mass matrix, with the following exceptions:

1. The command P2G = “name”, instead of K2GG = “name” or M2GG = “name”, is used in the Case Control Section.

2. The matrix defined in the DMIG entries are columns of the load vectors instead of the stiffness or mass matrix terms.
3. The load matrix, in general, is rectangular and unsymmetric (i.e., IFO = 9 on the header entry).
4. The column number (GJ) specified on the DMIG input must match the load sequence number. The GJ field is the load sequence number, and the CJ field is not used at all for a rectangular matrix. For example, if there is a total of ten load conditions in your model and you want to add load vectors to the third and seventh subcases using DMIG, then the NCOL field should be assigned as 10 on the header entry. Two additional DMIG Data entries, representing the two load vectors added to the third and seventh subcases, are needed. The GJ field for the first of these two data entries must have a value of 3 to denote the third load sequence. The remaining data for this entry are the nonzero rows of this load vector. Their input format is identical to that of the stiffness or mass matrix. The GJ field for the second of these two data entries must have a value of 7 to denote the seventh load sequence number. Again, the remaining data for this entry are the nonzero rows of this load vector.

Again the problem in [Figure 12-4](#) is used to illustrate this feature. The input file is shown in [Listing 12-8](#). In this example, the DMIG feature is used to read in the load for the vertical tip load in Subcase 10. In the Case Control Section, the P2G = UGLOAD command is used to activate the DMIG entries with the name of UGLOAD. In the Bulk Data Section, the DMIG load header entry is similar to that of the stiffness and mass header entry with the exception of the IFO and NCOL fields. In general, the matrix is rectangular; therefore, IFO is defined as 9 instead of 6. NCOL is the total number of load cases for the problem.

Since there are two load cases, NCOL must be defined as 2. The number of DMIG load data entries is equal to the number of external load columns added. In this case, you only need one DMIG load data entry since only one load condition is added to the first subcase. The GJ field of 1 denotes that you are adding the load to the first subcase (first load sequence). Note that GJ references the load sequence (1) rather than the SUBCASE ID (10). The CJ field should be blank when using the DMIG load feature. The rest of the fields are read in the same manner as the stiffness and mass matrices--only nonzero entries need to be defined. Note that if you have reversed your subcase setup--that is, if you apply your gravity load as the first subcase and bring in the tip load as the second subcase--then field three of the second DMIG entry must then be modified to 2 instead of 1.

Listing 12-8 Input File for Reading in a Load Vector Using DMIG

```

$
$  FILENAME - dmigload.dat
$
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = USE DMIG TO BRING IN LOAD VECTOR FOR SUBCASE 1
P2G = UGLOAD
SPC = 10
DISP = ALL
$
SUBCASE 10
LABEL = TIP LOAD AT END
$
SUBCASE 20
LABEL = GRAVITY LOAD
LOAD = 20
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
CBAR    5      1      5      6      10
$
DMIG    UGLOAD  0      9      1      2
DMIG    UGLOAD  1      6      3      100.
$
GRAV    20      9.81    0.    0.    1.
GRID    1      0.    0.    0.    246
GRID    2      1.    0.    0.    246
GRID    3      2.    0.    0.    246
GRID    4      3.    0.    0.    246
GRID    5      4.    0.    0.    246
GRID    6      5.    0.    0.    246
GRID    10     0.    0.    10.    123456
MAT1    1      7.1+10  .33    2700.
PBAR    1      1      2.654-3  5.869-7
SPC1    10     123456  1
$
ENDDATA

```

***** PARTIAL OUTPUT FROM DMIGFULL.F06 - FULL MODEL *****

TIP LOAD AT END

SUBCASE 1

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

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0

GRAVITY LOAD

SUBCASE 2

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

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	9.278418E-03	.0	-1.729160E-02	.0
3	G	.0	.0	3.233388E-02	.0	-2.783526E-02	.0
4	G	.0	.0	6.326194E-02	.0	-3.331796E-02	.0
5	G	.0	.0	9.784514E-02	.0	-3.542669E-02	.0
6	G	.0	.0	1.335530E-01	.0	-3.584844E-02	.0
10	G	.0	.0	.0	.0	.0	.0

***** PARTIAL OUTPUT FROM DMIGLOAD.F06 - DMIG TO INPUT LOAD CONDITION # 1 *****
LOAD CONDITION # 2 IS APPLIED BY THE CONVENTIONAL MANNER

TIP LOAD AT END

SUBCASE 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
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0

GRAVITY LOAD

SUBCASE 20

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
2	G	.0	.0	9.278418E-03	.0	-1.729160E-02	.0
3	G	.0	.0	3.233388E-02	.0	-2.783526E-02	.0
4	G	.0	.0	6.326194E-02	.0	-3.331796E-02	.0
5	G	.0	.0	9.784514E-02	.0	-3.542669E-02	.0
6	G	.0	.0	1.335530E-01	.0	-3.584844E-02	.0
10	G	.0	.0	.0	.0	.0	.0

Figure 12-6 Results Comparison Between Conventional Load Input Versus DMIG

As you may notice by now, inputting large matrices by hand using the DMIG method can be time consuming, error prone, and quite tedious. However, in most instances, the DMIG input may be automatically generated by a DMAP module when using MSC.Nastran or other external programs. PARAM,EXTOUT,DMIGPCH can also be used to generate these DMIG entries. See the [MSC.Nastran Superelement User's Guide](#) for a further description of this capability.

Linear Buckling

- Introduction to Linear Buckling
- Finite Element Approach
- Eigenvalue Extraction Method
- Assumptions and Limitations of Linear Buckling Analysis
- Examples

13.1 Introduction to Linear Buckling

In linear static analysis, a structure is normally considered to be in a state of stable equilibrium. As the applied load is removed, the structure is assumed to return to its original position. However, under certain combinations of loadings, the structure may become unstable. When this loading is reached, the structure continues to deflect without an increase in the magnitude of the loading. In this case, the structure has actually buckled or has become unstable; hence, the term “instability” is often used interchangeably with the term “buckling.”

Only linear buckling or elastic stability is considered in this user’s guide; in other words, assume there is no yielding of the structure and the direction of the forces do not change (i.e., follower force effects are ignored). Other assumptions of elastic stability are discussed in “[Assumptions and Limitations of Linear Buckling Analysis](#)” on page 497.

This chapter is organized into the following sections:

- Finite element approach
- Eigenvalue extraction method
- Assumptions and limitations of linear buckling analysis
- Examples

13.2 Finite Element Approach

In finite element analysis, the problem of linear buckling is addressed by including the effect of the differential stiffness to the linear stiffness matrix. The differential stiffness results from including the higher-order terms of the strain-displacement relationships. From a physical standpoint, the differential stiffness matrix represents the linear approximation of softening (reducing) the linear stiffness matrix in the case of a compressive axial load, and stiffening (increasing) the linear stiffness matrix in the case of a tensile axial load. Since the differential stiffness either adds or subtracts from the linear stiffness, often it is also called the incremental stiffness. The element linear and differential stiffness matrices are denoted as $[k_a]_i$ and $[k_d]_i$, respectively.

The differential stiffness matrix is a function of the geometry, element type, and applied loads. A look at the differential stiffness matrix of a single planar bar element as shown in **Figure 13-1** shows how a linear buckling analysis is handled using a finite element approach. For clarity, only the y and θ_z degrees of freedom at each end are retained for this example.

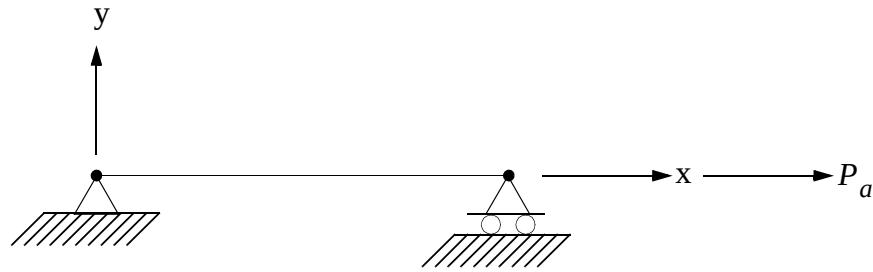


Figure 13-1 Single Planar Bar Element

From **The Nastran Theoretical Manual**, the differential stiffness for this planar bar element can be represented as

$$[k_d]_i = \begin{bmatrix} \frac{6F_{x_i}}{5l_i} & -\frac{F_{x_i}}{10} & -\frac{6F_{x_i}}{5l_i} & -\frac{F_{x_i}}{10} \\ -\frac{F_{x_i}}{10} & \frac{2l_i F_{x_i}}{15} & \frac{F_{x_i}}{10} & -\frac{l_i F_{x_i}}{30} \\ -\frac{6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} & \frac{6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} \\ -\frac{F_{x_i}}{10} & -\frac{l_i F_{x_i}}{30} & \frac{F_{x_i}}{10} & \frac{2l_i F_{x_i}}{15} \end{bmatrix} \quad \text{Eq. 13-1}$$

where F_{x_i} is the axial force in the CBAR element. In this case, P_a (applied load) = F_{x_i} since there is only one element in the model and the applied load is in line with the element axis. In the general case, F_{x_i} is proportional to P_a as long as the structure remains linear; in other words, if P_a is increased by scale factor a_i , then F_{x_i} also increases by the same scale factor. The value “i” stands for the i-th element. Note also that the differential stiffness matrix is

dependent only on the element type, the applied forces, and the geometry of the structure. This is the reason why the differential stiffness is also often called the geometric stiffness matrix. **Eq. 13-1** can, therefore, be rewritten as follows:

$$[k_d]_i = P_a \begin{bmatrix} \frac{6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} & \frac{-6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{2l_i\alpha_i}{15} & \frac{\alpha_i}{10} & \frac{-l_i\alpha_i}{30} \\ \frac{-6\alpha_i}{5l_i} & \frac{\alpha_i}{10} & \frac{6\alpha_i}{5l_i} & \frac{\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{-l_i\alpha_i}{30} & \frac{\alpha_i}{10} & \frac{2l_i\alpha_i}{15} \end{bmatrix} = P_a [\bar{k}_d]_i \quad \text{Eq. 13-2}$$

One can view α_i as the distribution factor of the applied load to the i-th element. Each element in the structure that supports differential stiffness (see “**Assumptions and Limitations of Linear Buckling Analysis**” on page 497) has an element differential stiffness matrix associated with it. Each of these differential stiffness matrices has a scale factor similar to **Eq. 13-2**. In general, each α_i is different for each element in the structure. The value of each α_i depends on the element type, the orientation of the element relative to the overall structure, and the applied load.

The system linear stiffness matrix can then be represented as

$$[K_a] = \sum_i^n k_{a_i}$$

Similarly, the system differential stiffness matrix can be represented as

$$[K_d] = \sum_i^n k_{d_i}$$

In general, the individual $[k_d]_i$ is more complicated than **Eq. 13-1**; however, the concept is the same. The overall system stiffness matrix is represented by **Eq. 13-3**.

$$[K] = [K_a] + [K_d] \quad \text{Eq. 13-3}$$

The total potential energy is equal to

$$[U] = 0.5 \{u\}^T [K_a] \{u\} + 0.5 \{u\}^T [K_d] \{u\} \quad \text{Eq. 13-4}$$

In order for the system to achieve static equilibrium, the total potential must have a stationary value; in other words, the relationship in **Eq. 13-5** must be satisfied

$$\frac{\partial [U]}{\partial u_i} = [K_a]\{u\} + [K_d]\{u\} = \{0\} \quad \text{Eq. 13-5}$$

where u_i is the displacement of the i -th degree of freedom.

It is convenient to rewrite **Eq. 13-5** as

$$[[K_a] + P_a[\overline{K_d}]]\{u\} = \{0\} \quad \text{Eq. 13-6}$$

where $[K_d] = P_a[\overline{K_d}]$ and P_a is the applied load. In order for **Eq. 13-6** to have a non-trivial solution, the following relationship must be true:

$$|[K_a] + P_a[\overline{K_d}]| = \{0\} \quad \text{Eq. 13-7}$$

where $| |$ stands for the determinant of the matrix. **Eq. 13-7** is only satisfied for certain values of P_a . These values of are the critical buckling loads.

A real structure has an infinite number of degrees of freedom. The finite element model approximates the behavior of the structure with a finite number of degrees of freedom. The number of buckling loads obtainable for your finite element model is equal to the number of degrees of freedom of your model. In other words,

$$P_{cr_i} = \lambda_i \cdot P_a \quad \text{Eq. 13-8}$$

Eq. 13-7 can, therefore, be rewritten as

$$|[K_a] + \lambda_i[K_d]| = [0] \quad \text{Eq. 13-9}$$

Eq. 13-9 is in the form of an eigenvalue problem. Once you obtain the eigenvalues λ_i , the buckling loads can then be obtained using **Eq. 13-8**. The values λ_i are the scale factors by which the applied load P_a is multiplied to produce the critical buckling loads

$$P_{cr_i}$$

As you can see from **Eq. 13-8**, the magnitude of the applied load P_a is arbitrary for arriving at the correct

$$P_{cr_i}$$

As an example, if P_a is increased by a factor of 10, then the calculated λ_i values in **Eq. 13-9** are reduced by a factor of 10; in other words, their resulting products

$$P_{cr_i}$$

remain the same.

In general, only the lowest buckling load is of any practical interest. The structure will fail prior to reaching any of the higher buckling loads.

Solution 105. In MSC.Nastran you can solve a linear buckling problem by using Solution 105 and following the procedure listed below.

1. Apply the static loads to the first n subcases (n is usually equal to one) and treat them as static analysis. The distribution of element forces due to these applied loads is generated internally. The actual magnitude of these applied loads is not critical.
2. You can perform buckling analysis on any or all of the loading conditions used in Step 1. One additional subcase is needed for each buckling analysis.
3. The $n+1$ to the $n+m$ subcases must each request an eigenvalue method from the Bulk Data Section to solve the eigenvalue problem shown in [Eq. 13-9](#). In this case, m is equal to the number of buckling analyses that you want to perform. Each buckling subcase may call out a unique eigenvalue solution.
4. The differential stiffness matrix is automatically generated for each element that supports differential stiffness. See “[Assumptions and Limitations of Linear Buckling Analysis](#)” on page 497 for a list of elements that support differential stiffness.
5. You must then multiply the eigenvalues obtained in Step 3 by the appropriate applied loads to obtain the buckling loads ([Eq. 13-8](#)) for each buckling analysis.
6. Each subcase may have a different boundary condition.

A typical input file used to calculate the buckling loads is shown in [Listing 13-1](#). In most applications, only one static and one buckling analysis is performed per run. Example 6 in “[Examples](#)” on page 498 contains an application of multiple static and buckling analyses.

Listing 13-1 Input File for a Typical Buckling Analysis

```

$
SOL 105
TIME 10
CEND
TITLE = SAMPLE INPUT FOR BUCKLING ANALYSIS
SPC = 10
DISP = ALL
$
$ STATIC SUBCASE TO GENERATE INTERNAL ELEMENT FORCES
$
SUBCASE 1
LOAD = 10
$
$ EIGENVALUE CALCULATION TO OBTAIN SCALE FACTORS BY WHICH
$ THE APPLIED LOAD(S) IS MULTIPLIED BY
$
SUBCASE 2
METHOD = 20
$
BEGIN BULK
$
$ APPLIED LOAD IN THE STATIC SUBCASE - SUBCASE ONE
$
FORCE,10,100,, -100.,1.0,0.,0.
$
$ LANCZOS EIGENVALUE METHOD REQUESTING THE LOWEST EIGENVALUE
$
EIGRL,20,,,1
$
$ BRING IN THE REST OF THE BULK DATA ENTRIES
$
include 'bulk.dat'
$
ENDDATA

```

You should also adhere to the following guidelines when solving a linear buckling problem in MSC.Nastran.

1. The Case Control Section must contain at least two subcases.
2. A METHOD command must appear in each buckling subcase to select the appropriate eigenvalue extraction method (EIGRL or EIGB entry) from the Bulk Data Section. If there are two or more buckling subcases, then a STATSUB = x command must be placed in each buckling subcase to select the appropriate static subcase. If there is only one buckling subcase, then the STATSUB = x command is optional if x references the first static subcase ID.
3. A static loading condition must be defined with a LOAD, TEMP(LOAD), or DEFORM selection unless all loading is specified by grid point displacements on SPC entries. All static subcases must be placed before the first buckling subcase.
4. Correct SPC sets must be selected for all subcases.
5. Output requests that apply to only a particular subcase must be placed inside that subcase.

6. Output requests that apply to all subcases may be placed above the subcase level.
7. Some type of output request (e.g., `disp(plot) = all`) must be requested for at least one subcase.

“**Examples**” on page 498 contains six example problems that illustrate some of these guidelines.

13.3 Eigenvalue Extraction Method

Seven methods of real eigenvalue extraction are available in MSC.Nastran:

- Givens
- Modified Givens
- Householder
- Modified Householder
- Inverse power
- Enhanced inverse power
- Lanczos

Three out of these seven methods can be used for linear buckling analysis:

- Inverse power
- Enhanced inverse power
- Lanczos

A brief description of each eigenvalue method is presented in the following sections. For further details, see the [*MSC.Nastran Numerical Methods User's Guide*](#).

Inverse Power Method (INV)

The inverse power method is a tracking method since it attempts to extract the lowest eigenvalue and eigenvector in the desired range first. Their effects are then “swept” out. The next higher mode is then found, and its effects are “swept” out, and so on--hence, the term “tracking,” which means that one root at a time is found. Furthermore, each root is found via an iterative process. (The classical literature often refers to this method as the “inverse iteration method with sweeping.”)

Unfortunately, depending on your selection of the eigenvalue range, the inverse power method can miss modes, which makes it unreliable. For this reason, the inverse method is not recommended. The enhanced inverse power method is, in general, a more reliable tracking method.

Enhanced Inverse Power Method (SINV)

This method is similar to the inverse power method except that it uses Sturm sequence logic to ensure that all modes are found within the specified eigenvalue range. The Sturm sequence informs you of the number of modes below each trial eigenvalue. See [*MSC.Nastran Numerical Methods User's Guide*](#) and the [*MSC.Nastran Basic Dynamics User's Guide*](#) for further details regarding the Sturm sequence check.

Lanczos Method

The Lanczos method overcomes the limitations and combines the best features of the other methods. It is efficient, and if an eigenvalue cannot be extracted within the range that you specify, a diagnostic message is issued. This method computes accurate eigenvalues and eigenvectors. Furthermore, it prints meaningful user diagnostics and supports parallel processing computers.

Comparison of Methods

The best method for a particular model depends on four factors:

- The size of the model (the total number of degrees of freedom)

- The number of eigenvalues desired

- The available real memory on your computer

- How well you can estimate the range of your eigenvalues

The enhanced inverse power method (SINV) can be a good choice if the model is too large to fit into memory, only a few modes are needed, and you have a reasonable idea of your eigenvalue range of interest. It is useful for models in which only the lowest few modes are desired. This method is also useful as a backup method to verify the accuracy of other methods.

For medium to large models, the Lanczos method is the recommended method.

Furthermore, the Lanczos method takes full advantage of sparse matrix methods that can substantially increase computational speed and reduce disk space usage. For overall robustness, the Lanczos method is the recommended method.

User Interface

A METHOD command is required in a subcase of the Case Control Section to select the appropriate eigenvalue extraction method in the Bulk Data Section. The Bulk Data entry is different depending on whether you are using the inverse power (INV), enhanced inverse power (SINV), or Lanczos method. The EIGRL entry is used for the Lanczos method, and the EIGB entry is used for the INV and SINV methods.

EIGRL Format

The EIGRL entry has the following format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND					

The SID field is the set identification number, which is referenced by the METHOD command in the Case Control Section. The V1 field defines the lower eigenvalue bound λ_l , and the V2 field defines the upper eigenvalue bound λ_u . The ND field specifies the number of roots desired. [Table 13-1](#) summarizes the action MSC.Nastran takes depending on the values specified for V1, V2, and ND.

Table 13-1 Number and Type of Roots Found with EIGRL Entry

Case	V1	V2	ND	Number and Type of Roots Found
1	x	x	x	Lowest ND or all in range, whichever is smaller
2	x	x		All in range
3	x		x	Lowest ND in range $[V1, +\infty]$
4	x			Lowest root in range $[V1, +\infty]$
5			x	Lowest ND roots in $[-\infty, +\infty]$
6				Lowest root
7		x	x	Lowest ND roots below V2
8		x		All below V2

EIGRL Example

1	2	3	4	5	6	7	8	9	10
EIGRL	10			1					

The above example selects the Lanczos method requesting the lowest root for your model.

In order for this entry to be used, the **METHOD = 10** command must be specified in the Case Control Section.

EIGB Format

The EIGB entry has the following format:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP				

The SID field is the set identification number, which is referenced by the **METHOD** command in the Case Control Section. The **METHOD** field selects the desired eigenvalue extraction method (SINV or INV). The **L1** field defines the lower eigenvalue bound λ_l , and the **L2** field defines the upper eigenvalue bound λ_u . The **NEP** field estimates the number of positive roots in the range and is used only for the INV method.

EIGB Example

1	2	3	4	5	6	7	8	9	10
EIGB	10	SINV	0.5	1.2					

The above example selects the enhanced inverse power method. All eigenvalues between 0.5 and 1.2 are desired. In order for this entry be used, the METHOD = 10 command must be specified in the Case Control Section.

EIGB Example

EIGB	20	INV	0.5	1.2	2				

The above example selects the inverse power method. There are two estimated roots in this eigenvalue range. In order for this entry be used, the METHOD = 20 command must be specified in the Case Control Section.

13.4 Assumptions and Limitations of Linear Buckling Analysis

The following assumptions and limitations apply to linear buckling analysis:

1. The deflections must be small.
2. The element stresses must be elastic.
3. The differential stiffness is supported for the following elements: CONROD, CROD, CTUBE, CBAR, CBEAM, CBEND, CQUAD4, CQUAD8, CTRIA3, CTRIA6, CSHEAR, CHEXA, CPENTA, and CTETRA.
4. A minimum of five grid points per half sine wave (buckled shape) is recommended.
5. The distribution of the internal element forces due to the applied loads remains constant.
6. Offsets should not be used in beam, plate, or shell elements for buckling analysis.
7. For 3-D buckling problems, the use of PARAM,K6ROT is recommended for CQUAD4 and CTRIA3 elements. A value of 100 is recommended.
8. For structures that exhibit nonlinear material or large deflection deformations, the linear buckling load obtained from Solution 105 may be different than the actual buckling load. For structures with significant nonlinearities, it is recommended that you perform a nonlinear buckling analysis using Solution 106. The discussion of nonlinear analysis is beyond the scope of this user's guide. See Reference [19](#) for further details regarding nonlinear buckling analysis.

13.5 Examples

This section contains six example problems pertaining to buckling. Each problem is chosen to illustrate either a particular type of failure mode, the effect of an element type, the effect of a structural configuration, or the method of performing multiple buckling analyses in a single run. The examples are as follows:

1. Classical Euler beam buckling.
2. Lateral buckling.
3. Planar frame buckling.
4. Buckling of a stiffened panel with transverse shear flexibility.
5. Buckling of a cylinder under uniform axial load.
6. Multiple buckling analyses in a single run.

In each example, the results produced by MSC.Nastran are compared with known theoretical solutions.

Example 1 -- Classical Euler Beam Buckling

Consider a classical Euler buckling problem. This problem is shown in [Figure 13-2](#).

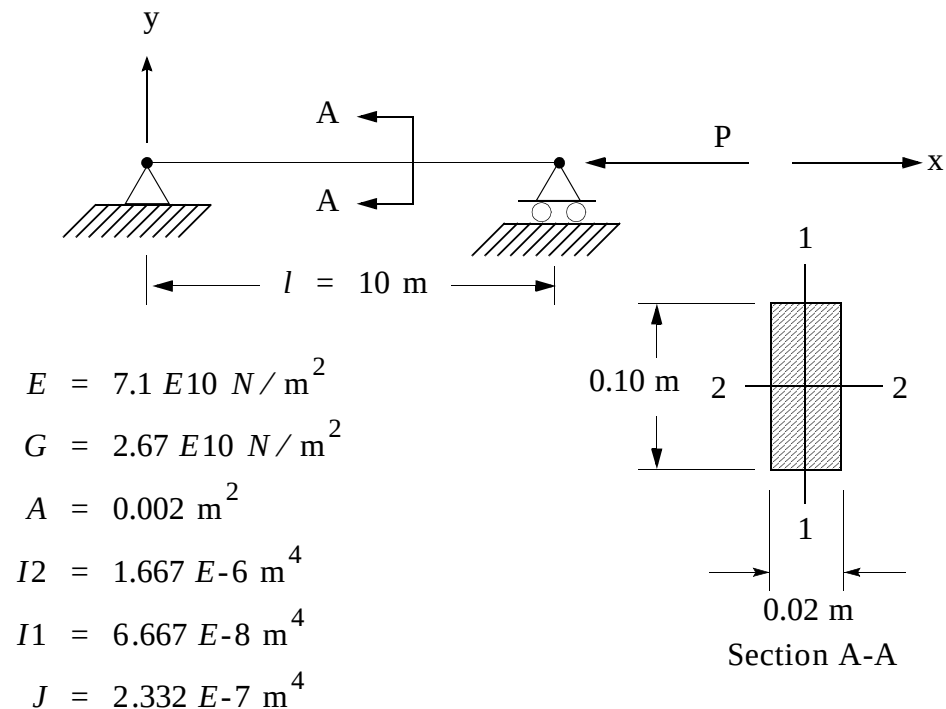


Figure 13-2 Euler Beam

The initial buckling occurs in the xz plane since I_1 is smaller than I_2 .

MSC.Nastran Results for Example 1

The MSC.Nastran input file used for this problem is shown in [Listing 13-2](#). In this case, a 1000 N compressive load is applied at the roller end to generate internal loads for the structure. The Lanczos eigenvalue extraction method is used to request the first five modes. Either the CBAR or CBEAM elements can be chosen for this problem; the CBAR elements are used in this case.

Listing 13-2 Input File for Euler Beam

```

$
$   FILENAME - bukeuler.dat
$
ID EULER BEAM
SOL 105
TIME 10
CEND
$
TITLE = EULER BEAM
SUBTITLE = METRIC UNITS
SPC = 10
DISP = ALL
$
SUBCASE 1
LOAD = 10
$
SUBCASE 2
METHOD = 10
$
BEGIN BULK
$
PARAM   POST   0
EIGRL   10             5
FORCE   10      11      -1000.  1.      0.      0.
$
CBAR    1         1         1         2         100
CBAR    2         1         2         3         100
CBAR    3         1         3         4         100
CBAR    4         1         4         5         100
CBAR    5         1         5         6         100
CBAR    6         1         6         7         100
CBAR    7         1         7         8         100
CBAR    8         1         8         9         100
CBAR    9         1         9        10         100
CBAR   10         1        10        11         100
$
GRID    1             0.      0.      0.
GRID    2             1.      0.      0.
GRID    3             2.      0.      0.
GRID    4             3.      0.      0.
GRID    5             4.      0.      0.
GRID    6             5.      0.      0.
GRID    7             6.      0.      0.
GRID    8             7.      0.      0.
GRID    9             8.      0.      0.
GRID   10             9.      0.      0.
GRID   11            10.      0.      0.
GRID   100            0.      0.      10.
$
MAT1     1         7.1+10         .33      2700.
$
$   RECTANGULAR SECTION OF DIMENSION .1m x .02m
$
PBAR     1         1         .002      6.667-8 1.667-6 2.332-7
$
SPC1     10        123         1
SPC1     10        23         11
SPC      10         1         4
$
ENDDATA

```

As indicated on the EIGRL entry, the first five modes are requested for this problem, and the eigenvalue table is shown in **Figure 13-3**. The first eigenvalue λ_1 in this case is equal to 0.4672, while the applied load in SUBCASE 1 is equal to -1000 N. Therefore, the lowest buckling load is equal to

$$P_1 = (\lambda_1)(P_a) = (.46719)(-1000) = -467.2 \text{ N}$$

MODE NO.	EXTRACTION ORDER	EIGENVALUE	REAL EIGENVALUES		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	4.671909E-01	6.835136E-01	1.087846E-01	4.934669E+02	2.305432E+02
2	2	1.869135E+00	1.367163E+00	2.175907E-01	2.181382E+03	4.077298E+03
3	3	4.209068E+00	2.051601E+00	3.265225E-01	4.431894E+03	1.865414E+04
4	4	7.498919E+00	2.738415E+00	4.358323E-01	4.971707E+03	3.728243E+04
5	5	1.168153E+01	3.417825E+00	5.439637E-01	4.934669E+02	5.764446E+03

Figure 13-3 Eigenvalue Table for a Euler Beam

A partial output that contains the first, second, and fifth eigenvectors is shown in **Figure 13-4**. The mode shapes for the first five modes are plotted in **Figure 13-5**.

EIGENVALUE = 4.671909E-01									
POINT ID.	TYPE	T1	REAL EIGENVECTOR NO.			1			
			T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	-3.141592E-01	-8.436650E-13		
2	G	2.344155E-14	-3.535500E-13	3.090170E-01	3.617327E-13	-2.987832E-01	5.605364E-13		
3	G	2.295983E-14	1.997131E-13	5.877852E-01	-5.413820E-14	-2.541602E-01	-5.748283E-13		
4	G	3.762660E-14	-8.750774E-15	8.090170E-01	-8.143186E-13	-1.846582E-01	-5.513294E-13		
5	G	5.222389E-14	1.995187E-14	9.510565E-01	2.312171E-13	-9.708054E-02	1.283237E-14		
6	G	6.208775E-14	-6.846741E-14	1.000000E+00	-2.349688E-13	6.485303E-13	-2.091279E-13		
7	G	9.229606E-14	1.712600E-13	9.510565E-01	-1.026823E-12	9.708054E-02	1.438084E-12		
8	G	8.977146E-14	-7.130678E-14	8.090170E-01	7.968065E-14	1.846582E-01	-5.453334E-13		
9	G	6.058189E-14	1.523950E-14	5.877852E-01	1.248126E-13	2.541602E-01	1.957519E-12		
10	G	8.648088E-14	-1.169763E-13	3.090170E-01	5.284149E-13	2.987832E-01	-1.513853E-12		
11	G	9.207012E-14	.0	.0	-1.129664E-12	3.141592E-01	2.010604E-12		
100	G	.0	.0	.0	.0	.0	.0		
EIGENVALUE = 1.869135E+00									
POINT ID.	TYPE	T1	REAL EIGENVECTOR NO.			2			
			T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	-6.606488E-01	1.696698E-11		
2	G	-5.494490E-13	8.408953E-12	6.180340E-01	-8.098147E-12	-5.344761E-01	-1.038079E-11		
3	G	-5.359018E-13	-4.648322E-12	1.000000E+00	1.386206E-12	-2.041517E-01	1.248417E-11		
4	G	-8.844888E-13	-1.029370E-13	1.000000E+00	1.719914E-11	2.041517E-01	1.241602E-11		
5	G	-1.226278E-12	-3.770107E-13	6.180340E-01	-5.748548E-12	5.344761E-01	-2.901899E-12		
6	G	-1.454765E-12	1.844125E-12	-1.042989E-12	3.660258E-12	6.606488E-01	5.452428E-12		
7	G	-2.164086E-12	-4.046730E-12	-6.180340E-01	2.005433E-11	5.344761E-01	-3.277508E-11		
8	G	-2.103291E-12	1.503905E-12	-1.000000E+00	-2.061866E-12	2.041517E-01	1.278820E-11		
9	G	-1.422088E-12	-3.211627E-13	-1.000000E+00	-3.318442E-12	-2.041517E-01	-4.639094E-11		
10	G	-2.025011E-12	2.810148E-12	-6.180340E-01	-1.061518E-11	-5.344761E-01	-3.454609E-11		
11	G	-2.158100E-12	.0	.0	2.222183E-11	-6.606488E-01	-4.772200E-11		
100	G	.0	.0	.0	.0	.0	.0		
EIGENVALUE = 1.168152E+01									
POINT ID.	TYPE	T1	REAL EIGENVECTOR NO.			5			
			T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	2.954037E-12	3.141592E-01		
2	G	-1.530020E-14	3.090170E-01	-4.216155E-12	-8.584071E-11	6.817444E-12	2.987832E-01		
3	G	-3.407930E-14	5.877852E-01	-1.068567E-11	8.455609E-11	4.070235E-12	2.541602E-01		
4	G	-4.763809E-14	8.090170E-01	-7.933438E-12	-6.013640E-11	-8.832388E-12	1.846582E-01		
5	G	-5.124878E-14	9.510565E-01	3.385932E-12	-8.368475E-11	-1.058233E-11	9.708054E-02		
6	G	-2.379340E-14	1.000000E+00	1.107325E-11	8.325122E-11	-5.506725E-12	2.671122E-13		
7	G	-4.143882E-15	9.510565E-01	1.763771E-11	-4.475530E-11	-8.492263E-12	-9.708054E-02		
8	G	-2.010117E-15	8.090170E-01	2.572503E-11	-1.136649E-11	-4.849835E-12	-1.846582E-01		
9	G	3.640430E-15	5.877852E-01	2.352012E-11	-2.112232E-10	9.559306E-12	-2.541602E-01		
10	G	-1.566881E-14	3.090170E-01	1.092055E-11	5.283983E-11	1.326611E-11	-2.987832E-01		
11	G	-2.601413E-14	.0	.0	1.037907E-10	9.724306E-12	-3.141592E-01		
100	G	.0	.0	.0	.0	.0	.0		

Figure 13-4 Eigenvectors for an Euler Beam

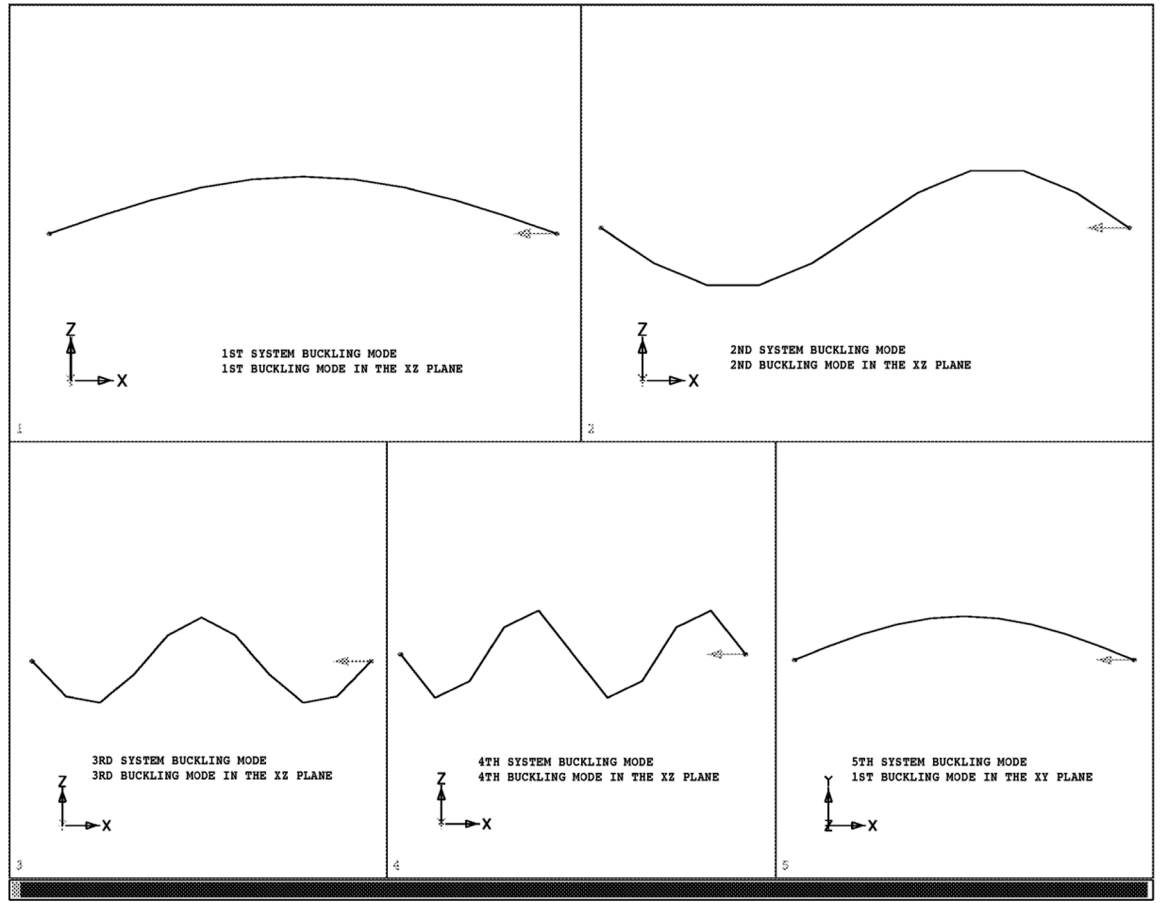


Figure 13-5 Mode Shapes for an Euler Beam

Note that the first four system buckling modes are in the xz plane and the first buckling mode in the xy plane is the fifth overall system mode. The lowest buckling load in the xy plane is, therefore, equal to

$$P_5 = (\lambda_5)(P_a) = (11.6815)(-1000) = -11681.5 \text{ N}$$

Theoretical Results for Example 1

From Reference 9., this Euler buckling load is calculated as follows:

$$P_1 = \frac{\pi^2 EI_2}{l^2} = \pi^2 (7.1 \cdot 10^{10}) (6.667 \cdot 10^{-8}) / 100 = 467.2 \text{ N}$$

For this particular problem, the buckling load in the xy plane is equal to

$$P_5 = \frac{\pi^2 EI_2}{l^2} = \pi^2 (7.1 \cdot 10^{10}) (1.667 \cdot 10^{-6}) / 100 = 11681.4 \text{ N}$$

A comparison between MSC.Nastran results and the theoretical results is summarized in [Table 13-2](#).

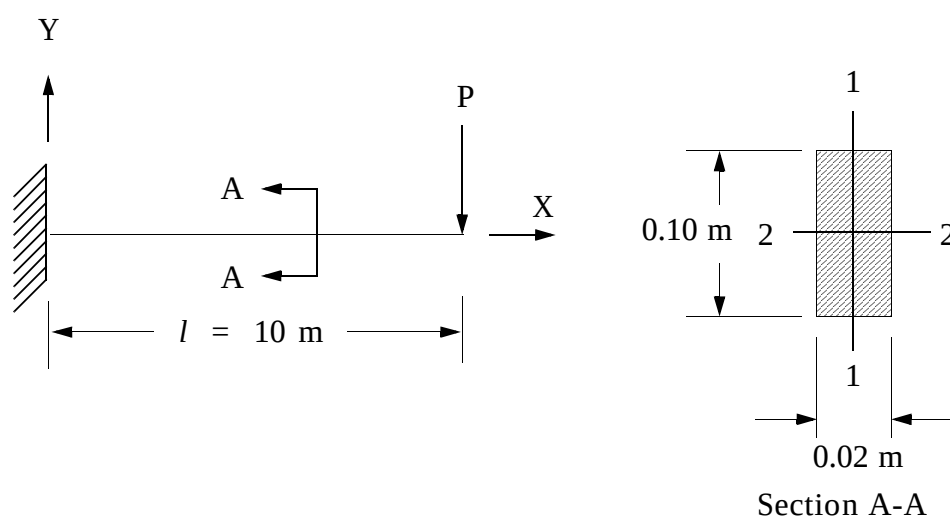
Table 13-2 Euler Beam Results Comparison

Mode Number	MSC.Nastran (N)	Theoretical (N)	% Difference
1 (XZ Plane)	467.2	467.2	0
5 (XY Plane)	11681.5	11681.4	0.001

As you approach the higher modes, the mode shapes become less smooth. This effect occurs because for the same mesh density, the higher modes have fewer grid points per buckled sine wave than the lower modes. If you are interested in these mode shapes, you should increase your model mesh density to follow the general guideline of maintaining a minimum of five grid points per half sine wave. However, these higher failure modes are seldom of interest in real-life applications.

Example 2 -- Lateral Buckling

The problem shown in **Figure 13-6** illustrates an interesting case of lateral buckling. The model consists of a cantilever beam with a vertical tip load applied at the free end. The dimensions of the beam are identical to Example 1. In this example, the structure buckles in the lateral direction (xz plane) perpendicular to the applied load rather than in the plane of the applied load (xy plane).

**Figure 13-6 Lateral Buckling**

MSC.Nastran Results for Example 2

Listing 13-3 contains the corresponding MSC.Nastran model for this lateral buckling problem. The CBEAM element is used in this case, although the CBAR element can also be used since this beam consists of a constant cross section. A vertical 1000 N load is applied in the negative y-direction to generate the element internal loads in Subcase 1. Once again, the Lanczos eigenvalue extraction method is employed requesting two modes. **Figure 13-8** contains the eigenvalue table and corresponding eigenvectors. The static deflection and buckling mode shapes are shown in **Figure 13-10**.

It is interesting to note that the static deflection is in the plane of the applied load (xy plane), while the buckling mode shape is in the plane perpendicular to the applied load (xz plane). Two modes were requested on the EIGRL entry to illustrate another subtle point for this problem. The two eigenvalues are identical in magnitude but different in sign. What is the significance of a negative eigenvalue in a buckling analysis? In buckling analysis, a negative eigenvalue implies that the load that causes the structure to buckle is opposite to the direction of the applied load.

Since both the structure and loading are symmetric, it does not matter whether the applied load is in the +y or -y direction. This fact brings up another subtle point. If you specify V2 only on the EIGRL entry, then MSC.Nastran attempts to extract all the roots below V2. In this case, these roots include all the negative eigenvalues. For a large model, calculating a large number of modes can be expensive especially when you are only interested in the lowest modes. Note that these negative roots are legitimate both from a physical and mathematical standpoint. In general, this situation only occurs if the applied buckling load is in the opposite direction of the buckling load. For this particular problem, there is no right or wrong direction. Therefore, in order to avoid this type of surprise, it is best to request the lowest number of desired roots (ND) on the EIGRL entry when you are not sure of the buckling load direction.

Listing 13-3 Input File for Lateral Buckling

```

$
$  FILENAME - bucklat.dat
$
SOL 105
TIME 10
CEND
$
TITLE = LATERAL BUCKLING OF CANTILEVER BEAM
SUBTITLE = METRIC UNITS
SPC = 10
DISP = ALL
$
SUBCASE 1
LOAD = 10
$
SUBCASE 2
METHOD = 10
$
BEGIN BULK
$
EIGRL  10 2
$
FORCE  10 11 -1000. 0. 1. 0.
$
CBEAM  1 1 1 2 100
CBEAM  2 1 2 3 100
CBEAM  3 1 3 4 100
CBEAM  4 1 4 5 100
CBEAM  5 1 5 6 100
CBEAM  6 1 6 7 100
CBEAM  7 1 7 8 100
CBEAM  8 1 8 9 100
CBEAM  9 1 9 10 100
CBEAM 10 1 10 11 100
$
GRID  1 0. 0. 0.
GRID  2 1. 0. 0.
GRID  3 2. 0. 0.
GRID  4 3. 0. 0.
GRID  5 4. 0. 0.
GRID  6 5. 0. 0.
GRID  7 6. 0. 0.
GRID  8 7. 0. 0.
GRID  9 8. 0. 0.
GRID 10 9. 0. 0.
GRID 11 10. 0. 0.
GRID 100 0. 0. 10.
$
MAT1  1 7.1+10 .33 2700.
$
$  RECTANGULAR SECTION OF DIMENSION .1m x .02m
$
PBEAM 1 1 .002 6.667-81.667-6 2.332-7
$
SPC1  10 123456 1
$
ENDDATA

```


R E A L E I G E N V A L U E S						
MODE	EXTRACTION	EIGENVALUE	RADIANS	CYCLES	GENERALIZED	GENERALIZED
NO.	ORDER				MASS	STIFFNESS
1	1	2.183456E-01	4.672747E-01	7.436907E-02	1.749877E+02	3.820780E+01
2	2	-2.183456E-01	4.672747E-01	7.436907E-02	-1.749877E+02	3.820780E+01

EIGENVALUE = 2.183456E-01

R E A L E I G E N V E C T O R N O . 1								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	2.537741E-11	-3.892257E-09	2.499774E-03	3.424457E-02	-7.371472E-03	-4.395732E-09	
3	G	-1.277730E-11	-1.073776E-08	1.867549E-02	6.424069E-02	-2.656322E-02	-1.484095E-08	
4	G	1.189500E-10	-2.919947E-08	5.795542E-02	8.779415E-02	-5.277112E-02	-1.207688E-08	
5	G	8.478357E-11	-2.620355E-08	1.250618E-01	1.045299E-01	-8.153865E-02	1.648216E-08	
6	G	1.273718E-10	-2.596014E-09	2.207153E-01	1.152518E-01	-1.093766E-01	2.570351E-08	
7	G	9.853920E-11	2.034603E-08	3.427072E-01	1.213348E-01	-1.339076E-01	2.049627E-08	
8	G	8.265020E-11	4.163538E-08	4.869550E-01	1.242675E-01	-1.537219E-01	2.487351E-08	
9	G	1.443917E-10	6.310452E-08	6.483413E-01	1.253649E-01	-1.681112E-01	8.187754E-09	
10	G	1.136984E-10	4.827729E-08	8.212749E-01	1.256189E-01	-1.767931E-01	-3.784262E-08	
11	G	1.540406E-10	-4.905004E-09	1.000000E+00	1.256360E-01	-1.796903E-01	-6.074886E-08	
100	G	.0	.0	.0	.0	.0	.0	

EIGENVALUE = -2.183456E-01

R E A L E I G E N V E C T O R N O . 2								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	7.576565E-11	2.472828E-09	2.499775E-03	-3.424457E-02	-7.371472E-03	-3.871140E-10	
3	G	-4.478584E-12	-3.649991E-09	1.867549E-02	-6.424069E-02	-2.656322E-02	-9.834048E-09	
4	G	1.762648E-11	-1.338523E-08	5.795542E-02	-8.779415E-02	-5.277112E-02	-7.743097E-09	
5	G	-7.876090E-11	-1.461366E-08	1.250618E-01	-1.045299E-01	-8.153865E-02	8.549041E-09	
6	G	-1.393435E-10	8.698958E-10	2.207153E-01	-1.152518E-01	-1.093766E-01	1.680920E-08	
7	G	-9.326392E-11	6.915506E-09	3.427072E-01	-1.213349E-01	-1.339076E-01	-1.117986E-08	
8	G	-1.053673E-10	-1.124932E-08	4.869550E-01	-1.242675E-01	-1.537219E-01	-7.835656E-09	
9	G	-8.121134E-11	-4.722858E-09	6.483413E-01	-1.253649E-01	-1.681112E-01	7.580319E-09	
10	G	-1.725437E-10	-1.078508E-08	8.212749E-01	-1.256188E-01	-1.767931E-01	-2.140314E-08	
11	G	-1.482953E-10	-4.360446E-08	1.000000E+00	-1.256359E-01	-1.796903E-01	-3.865582E-08	
100	G	.0	.0	.0	.0	.0	.0	

Figure 13-7 Eigenvalue Table and Eigenvectors for Lateral Beam Buckling

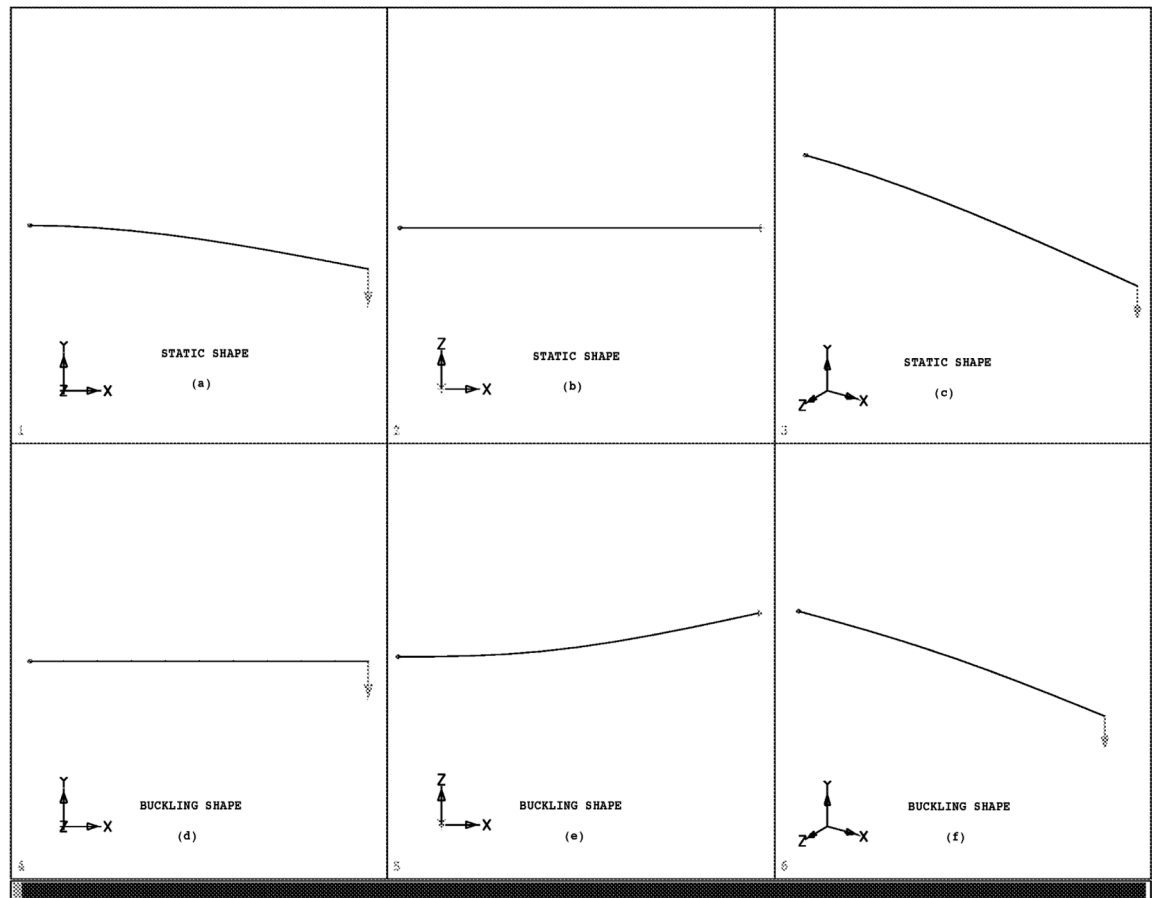


Figure 13-8 Static and Buckling Shapes

The lowest buckling load calculated by MSC.Nastran is

$$P_{cr_1} = \pm 218(1000) = \pm 218N$$

Theoretical Results for Example 2

From Reference 9., the theoretical buckling load can be calculated as follows:

$$\begin{aligned}
 P_{cr} &= \frac{4.013}{l^2} \sqrt{EIGJ} \\
 &= \frac{4.013}{100} \sqrt{7.1 \cdot 10^{10} (6.667 \cdot 10^{-8}) (2.67 \cdot 10^{10}) (2.332 \cdot 10^{-7})} \\
 &= 218 \text{ N}
 \end{aligned}$$

Table 13-3 contains a comparison between the theoretical results versus the MSC.Nastran results.

Table 13-3 Lateral Buckling Results Comparison

MSC.Nastran (N)	Theoretical (N)	% Difference
218	218	0

Example 3 -- Planar Frame Buckling

This example is a planar frame subjected to a compressive load at the top as shown in [Figure 13-9](#). At first glance, since both the structure and loading are symmetrical, you may be tempted to conclude that the structure will buckle in a symmetric mode. As illustrated in [Figure 13-10\(a\)](#) (from MSC.Nastran) and S. P. Timoshenko and J. M. Gere, *Theory of Elastic Stability* (Reference 9.), this is not the case. The lowest buckling mode is actually antisymmetric. The second buckling mode is symmetric ([Figure 13-10\(b\)](#)).

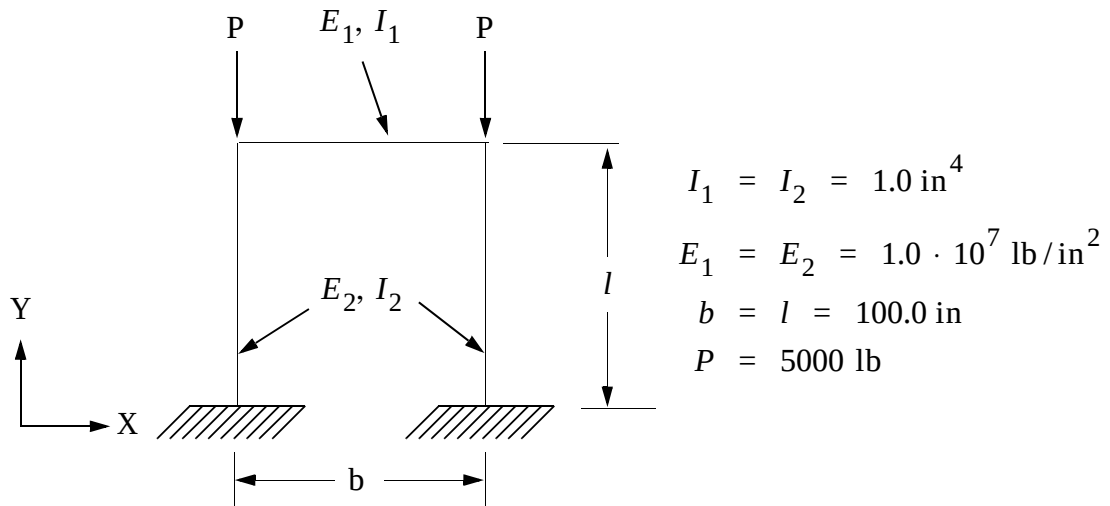


Figure 13-9 Planar Frame Buckling

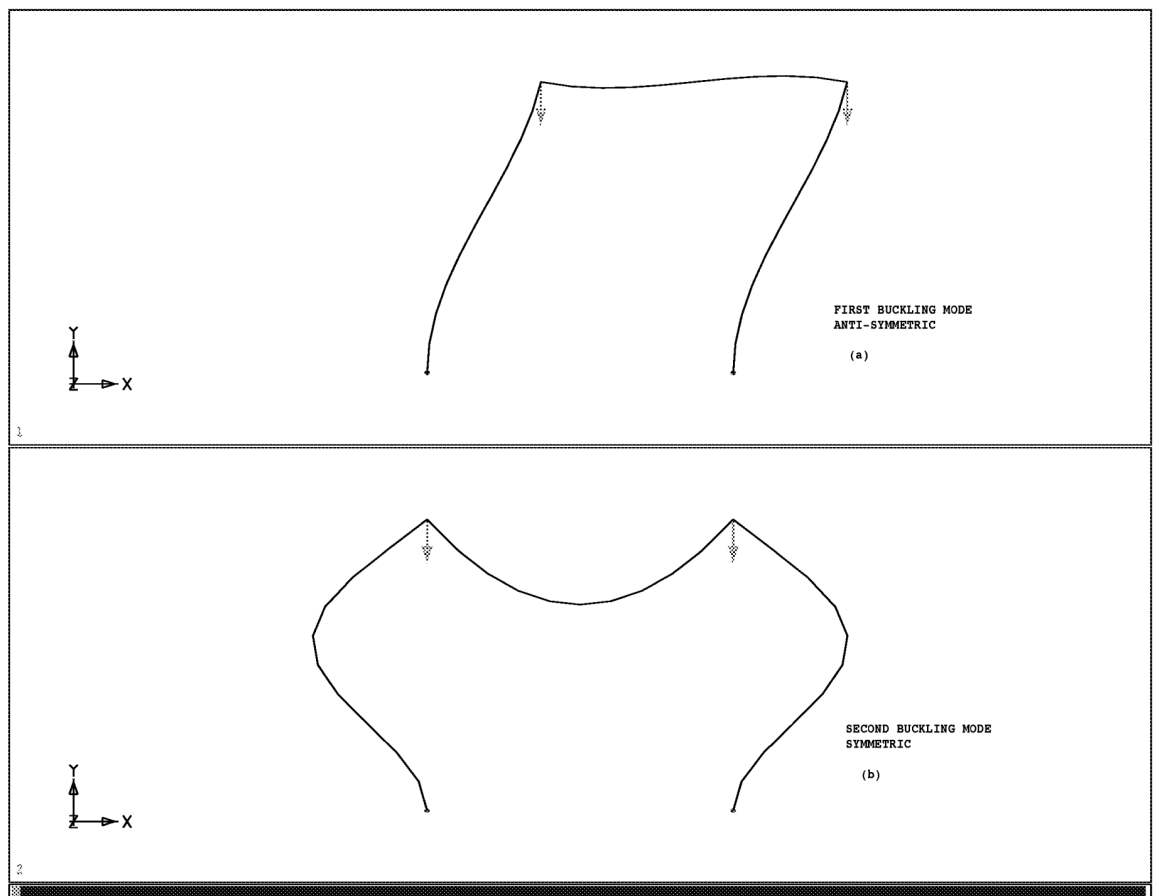


Figure 13-10 Frame Buckling

MSC.Nastran Results for Example 3

A partial listing of the MSC.Nastran input file used to model this problem is shown in [Listing 13-4](#).

Listing 13-4 Input File for Frame Buckling

```
$
$ INPUT FILE - bukframe.dat
$
SOL      105  $
TIME     5
CEND
TITLE = FRAME BUCKLING PROBLEM
SUBTITLE = PLANAR MODEL
DISP = ALL
SPC = 1
$
SUBCASE 1
LABEL = STATIC LOAD CASE
LOAD = 1
$
SUBCASE 2
LABEL = BUCKLING CASE
METHOD = 10
$
BEGIN BULK
$
$ BRING IN REST OF BULK DATA FILE
$
INCLUDE 'bucklat.dat'
$
$ LOADS BULK DATA ENTRIES
$
FORCE    1      11      0      5000.      -1.
FORCE    1      22      0      5000.      -1.
$
$ CONSTRAINTS BULK DATA ENTRIES
$
SPC1      1      123456  1      33
$
$ PROPERTY AND MATERIAL BULK DATA ENTRIES
$
PBAR      1      1      3.46      1.      1.
MAT1      1      1.+7      .3
$
$ FOR CALCULATING BUCKLING MODES
$
EIGRL     10      3
ENDDATA
```

The lowest buckling load calculated by MSC.Nastran is

$$P_{cr_1} = (\lambda_1)(P_a) = 1.4756 (-5000) = -7378 \text{ lb}$$

Theoretical Results for Example 3

From *Theory of Elastic Stability* (Reference 9.), the buckling load P_{cr} can be calculated as

$$P_{cr} = k^2 E_2 I_2 \tag{Eq. 13-10}$$

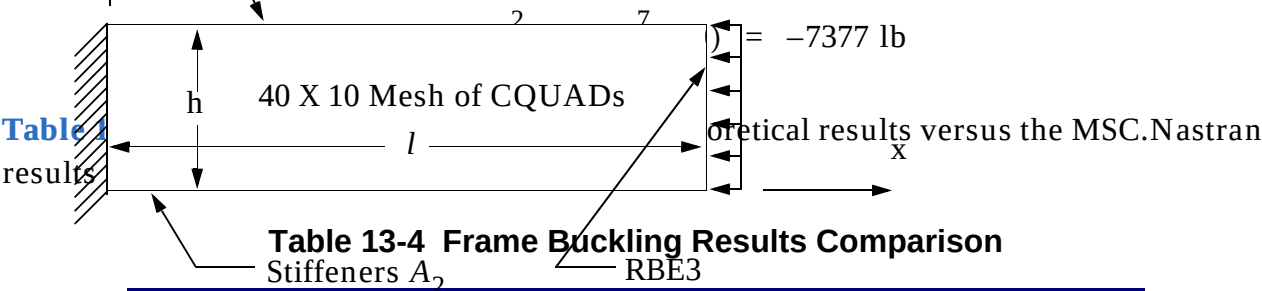
where the value of k can be obtained by solving the following transcendental equation

$$\frac{kl}{\tan kl} = \frac{-6 l I_1}{b I_2} \tag{Eq. 13-11}$$

Solving Eq. 13-11, the lowest value of k is equal to

$$k_1 = 0.02716 \tag{Eq. 13-12}$$

Note that all the roots from the above transcendental equation only yield the anti-symmetric modes. In this case, the lowest buckling load is



	MSC.Nastran (lb)	Theoretical (lb)	% Difference
$A_1 = A_2 = 0.6 \text{ in}^2$; $h = 2.0 \text{ in}$	7378	7377	0.01

Example 4 -- Buckling of a Stiffened Panel with Transverse Shear Flexibility

This problem is selected to illustrate the effect of the transverse shear flexibility in the buckling failure of a stiffened panel. Figure 13-11 consists of a panel reinforced with stiffeners at both the top and bottom. The panel is subjected to a distributed compressive load at the right edge. The failure mode of interest in this case is the in-plane (xy plane) buckling of the stiffened panel.

$t = 0.2 \text{ in}$; $n = A_t / A_w = 4$

Figure 13-11 Buckling of a Stiffened Panel

MSC.Nastran Results for Example 4

The corresponding MSC.Nastran input file for this problem is shown in [Listing 13-5](#). The panel is modeled with CQUAD4s. Since only the in-plane buckling failure mode is of interest, only the membrane stiffness is requested for the CQUAD4s (MID1 only). The stiffeners are modeled with CRODs. Since this is a planar model, the out-of-plane motion is constrained (3, 4, 5, and 6 DOFs).

The panel is subjected to a distributed compressive load at the right end. As an alternative modeling technique, an RBE3 element is connected to the grid points at the right edge so that the load can be applied to a single grid point with the RBE3 spreading the loads to the other grid points.

The SINV method of eigenvalue extraction method is used for this problem. Therefore, the EIGB entry instead of the EIGRL entry is used. Field 3 of the EIGB entry designates the selected method. Fields 4 and 5 provide the range of the eigenvalue of interest. A reduced output showing the eigenvalue table and lowest buckling mode shape is included in [Figure 13-13](#) and [Figure 13-14](#), respectively.

Listing 13-5 Input File for the Buckling of Stiffened Panels

```

$
$   FILENAME - stffqud4.dat
$
SOL   105
TIME  10
CEND
TITLE = PANEL WITH STIFFENERS
DISP = ALL
STRESS = ALL
SPC = 1
$
SUBCASE 1
LABEL = COMPRESSIVE LOAD
LOAD = 1
$
SUBCASE 2
LABEL = BUCKLING SUBCASE
METHOD = 10
$
BEGIN BULK
$
$EIGRL10          1
EIGB   10      SINV   .7   .9
PARAM   POST    0
$
RBE3   1000      1000   123456  2.   12   82   123   +
+      164      205   287   328   369   410   2.   12345   +
+      246      1.0   12   41   451
$
FORCE  1      1000      -1.0   100000.
$
$ THIS SECTION CONTAINS THE LOADS, CONSTRAINTS, AND CONTROL BULK DATA ENTRIES
$
PSHELL 1      1      .2
PROD   2      1      .6
$
MAT1   1      1.+7      .32
$
$ BRING IN THE REST OF THE MODEL
$
include 'stffqud4.blk'
$
ENDDATA

```

R E A L E I G E N V A L U E S						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.811782E-01	8.838429E-01	1.406680E-01	6.161843E+03	4.813498E+03
2	3	4.946603E+00	2.224096E+00	3.539759E-01	1.096963E+07	5.426240E+07
3	2	5.002625E+00	2.236655E+00	3.559746E-01	1.383715E+04	6.922209E+04

Figure 13-12 Eigenvalues for a Stiffened Panel

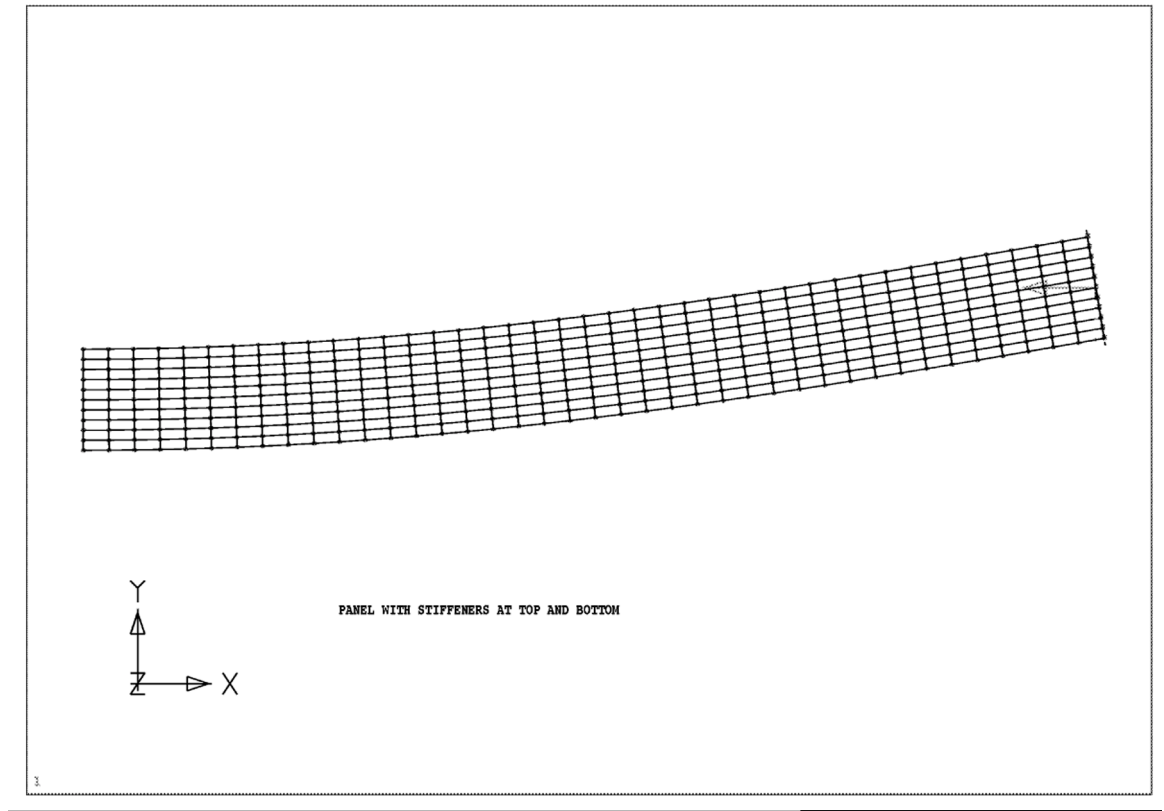


Figure 13-13 Stiffened Panel Buckling Mode

The lowest buckling load calculated by MSC.Nastran is

$$P_{cr1} = (\lambda_1)(P_a) = 0.78118(-(100,000)) = -78,118 \text{ lb}$$

Theoretical Results for Example 4

If this problem is treated as an Euler beam, the critical buckling load P_e is equal to

$$P_e = \frac{\pi^2 EI}{4 l^2} = 82,245 \text{ lb} \quad \text{Eq. 13-13}$$

If the effect of the transverse shear flexibility is included, then (from Reference 9.) can be calculated as follows:

$$P_{cr} = \frac{\sqrt{1 + \frac{4nPe}{A_t G}} - 1}{\frac{2n}{A_t G}} = 78,210 \text{ lb} \quad \text{Eq. 13-14}$$

If the transverse shear flexibility is ignored, then the buckling load deviates by 5.2%.

Table 13-5 contains a comparison between the theoretical results versus the MSC.Nastran results for the buckling of the stiffened panel. Note that the transverse shear flexibility of this structure is automatically included when you are performing a buckling analysis in MSC.Nastran. There is no additional input required on your part.

Table 13-5 Stiffened Panel Buckling Results Comparison

MSC.Nastran (lb)	Theoretical (lb)	% Difference
78,118	78,210	0.1

A review of the stresses also indicates that the structure will yield prior to reaching the linear buckling load level. In other words, the critical failure mode may be due to yielding rather than to the linear buckling of the structure. The knowledge of this linear buckling load level can still be of design significance. If you are interested in detailed stresses for this problem, then a nonlinear analysis using Solution 106 may be more appropriate in this case.

Example 5 -- Buckling of a Cylinder Under Uniform Axial Load

The next example is the buckling of a cylinder subjected to a distributed compressive load at one end and simply supported at the other end. The cylinder has a diameter of 20 inches and a length of 20 inches with a wall thickness of 0.03 inches as shown in **Figure 13-14**. This problem illustrates the phenomenon of the buckling of a thin curved shell structure.

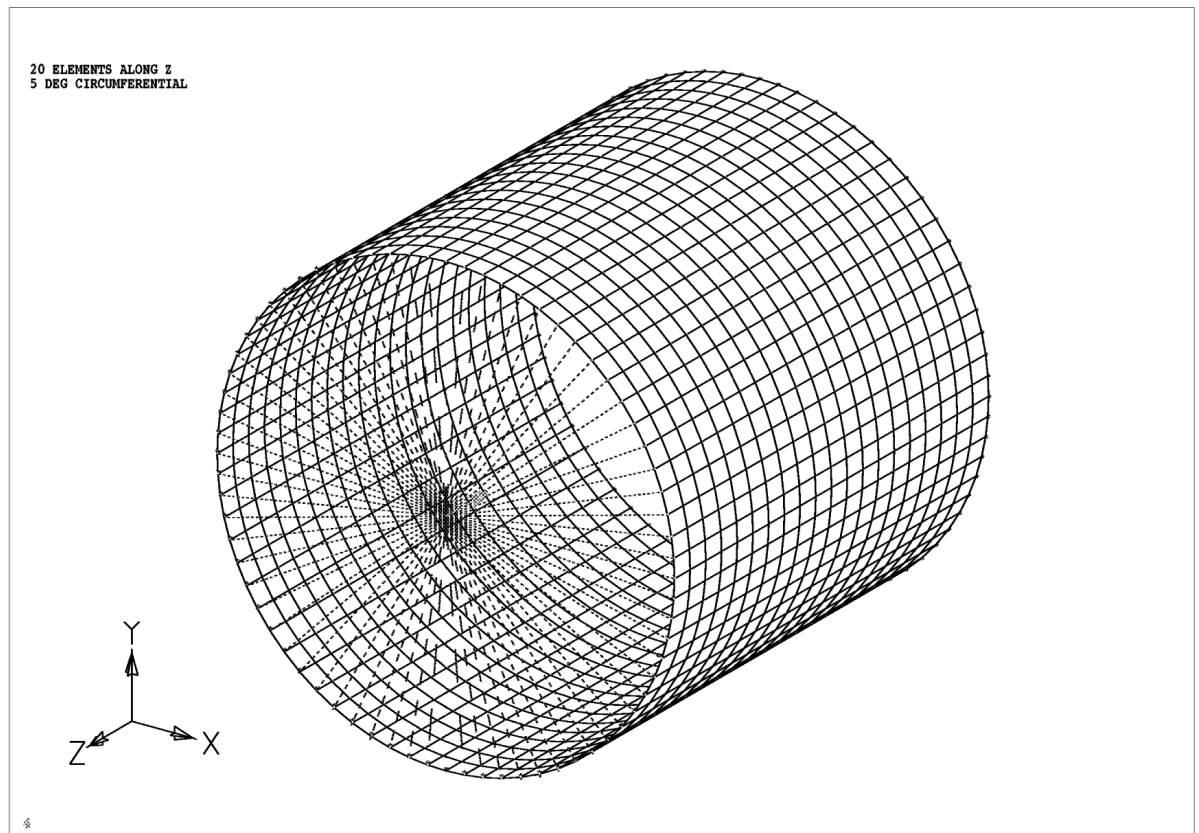


Figure 13-14 Buckling of a Cylinder Under a Compressive Load

MSC.Nastran Results for Example 5

The CQUAD4 elements are used for the MSC.Nastran model. A mesh of 20 elements along the length and 72 elements around the circumference is used for this problem. The corresponding input file containing the pertinent entries is shown in [Listing 13-6](#).

Listing 13-6 Input File for a Cylindrical Buckling Problem

```

$
$
$  FILENAME - buckcy20r.dat
$
ID CYLIN BUCKLING
SOL      105
TIME     200
CEND
TITLE = BUCKLING OF CYLINDER - SIMPLY SUPPORTED
SUBTITLE = 20" x 20" - t=.03" - CP IN CYL COORD SYSTEM - K6ROT = 100.
DISP = ALL
SPC = 1
$
SUBCASE 1
LABEL = STATIC LOAD
LOAD = 1
SPCF = ALL
STRESS = ALL
$
SUBCASE 2
LABEL = EIGENVALUE CALCULATION
METHOD = 1
$
BEGIN BULK
$
PARAM      K6ROT      100.
PARAM      POST       -1
EIGRL      1                      2
$
PSHELL     1          1          .03      1
$
MAT1       1          1.+7          .3
$
include 'full_cp1.blk'
$
$ THIS SECTION CONTAINS ALL DEFINED COORDINATE SYSTEMS
$
CORD2C     1          0          0.0      0.0      0.0      0.0      0.0      1.      +
           1.          0.0      0.0
$
RBE3       5000          5000      123456  1.          123      381      382      +
+          383          384      385      386      387      388      389      390      +
+          391          392      393      394      395      396      397      398      +
+          399          781      782      783      784      785      786      787      +
+          788          789      790      791      792      793      794      795      +
+          796          797      798      1180     1181     1182     1183     1184      +
+          1185         1186     1187     1188     1189     1190     1191     1192      +
+          1193         1194     1195     1196     1197     1579     1580     1581      +
+          1582         1583     1584     1585     1586     1587     1588     1589      +
+          1590         1591     1592     1593     1594     1595
$
$
FORCE      1          5000      0          100000.      -1.
$
ENDDATA

```

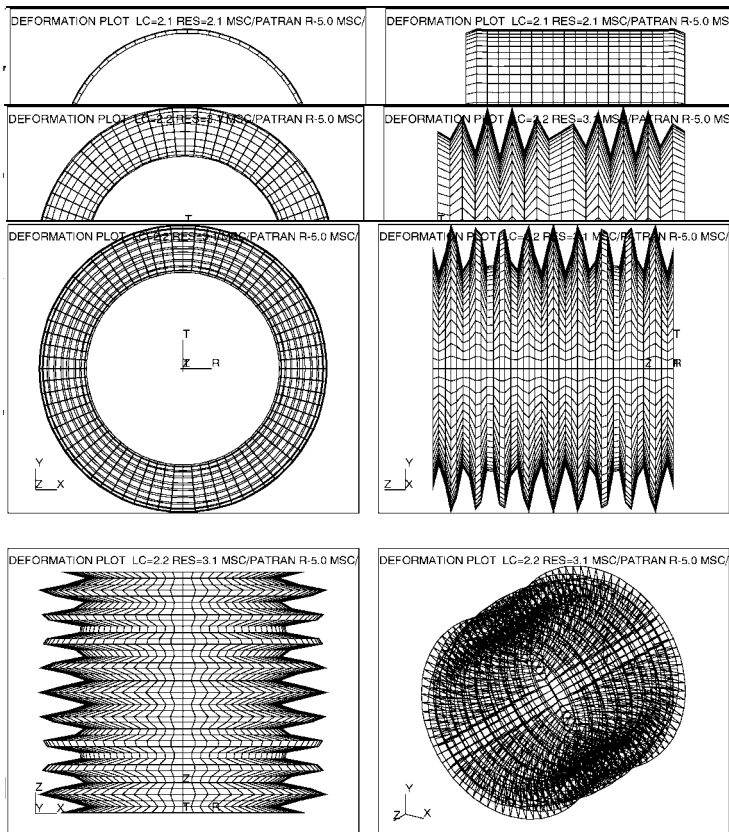
Once again, an RBE3 element is used to distribute the load from a single grid point to the circumference of the cylinder.

A static load of -100,000 lb is applied in Subcase 1 to generate the internal forces for the structure. This load is applied to the RBE3, which, in turn, distributes the loads around the circumference of the cylinder. The static deflection is shown in [Figure 13-15](#).

The Lanczos eigenvalue extraction method is used in this case to request the lowest mode. The lowest calculated eigenvalue is equal to 0.35986. The buckling load is, therefore, equal to

$$P_{cr} = (.34262)(-100,000) = -34,262 \text{ lb}$$

The corresponding buckling mode shape is shown in [Figure 13-16](#). As you can see from the plots, there are two grid points per half sine wave (four grid points per sine wave), which is below the recommended value of a minimum of five grid points per half sine



cedure is shown in “[Generating the](#)

ng the z-direction keeping the same
ew eigenvalue calculated in this case
efore, equal to

$$-34,234 \text{ lb}$$

3-17. In this case, there are three grid
ve), which is still below the minimum
ainly further refine this model until
ement. However, this mesh is
ckling features of MSC.Nastran.

Due to a Compressive Load

20 Elements Along the z-Direction

Figure 13-17 Buckling Shape of a Cylinder with 40 Elements Along the z-Direction

Theoretical Results for Example 5

The first buckling load (Reference 9.) can be calculated as follows:

$$N_x = \frac{E(h)^2}{r\sqrt{3(1-\nu^2)}} = \frac{1 \times 10^7 (.03)^2}{10\sqrt{3(1-.3^2)}} = 544.7 \text{ lb/in}$$

$$P_{cr} = N_x(2)(\pi)r = 544.7(2)(\pi)(10) = 34,225 \text{ lb}$$

Table 13-6 contains a comparison between the results obtained with MSC.Nastran versus the theoretical results.

Table 13-6 Thin Cylinder Buckling Results Comparison

Mesh Density	MSC.Nastran (N)	Theoretical (N)	% Difference
72 x 20	34,262	34,225	.11
72 x 40	34,234	34,225	.03

Example 6 -- Multiple Buckling Analyses in a Single Run

So far, a single static analysis was considered followed by a single buckling analysis. Example 6 shows you how to run multiple static and buckling analyses in a single run. In fact, what is done in this example is to combine the Euler beam buckling (Example 1) and lateral buckling (Example 2) problem into a single run. The corresponding input file is shown in [Listing 13-7](#).

The model used is the same geometric beam model used in Examples 1 and 2. The first subcase (Subcase 2) is a static subcase consisting of a cantilever beam with a vertical tip load applied to the free end (see [Figure 13-6](#)). The second subcase (Subcase 5) is a static subcase consisting of a simply supported beam with an axial load applied to the roller end (see [Figure 13-2](#)). The third subcase (Subcase 11) is for a lateral buckling analysis. The Case Control command (STATSUB = 2 in Subcase 11) tells MSC.Nastran that you want to generate the differential stiffness matrix from the first static subcase (Subcase 2). The fourth subcase (Subcase 21) is for a Euler beam buckling analysis. The Case Control command (STATSUB = 5 in Subcase 21) tells MSC.Nastran that you want to generate the differential stiffness matrix from the second static subcase (Subcase 5). The Bulk Data entries are similar to Examples 1 and 2. Note that different boundary conditions are allowed for different subcases.

Listing 13-7 Input File for Multiple Buckling Analyses

```
$  FILENAME - MULTBUCK.DAT
$
SOL 105
TIME 10
CEND
$
TITLE = LATERAL BUCKLING OF CANTILEVER BEAM
SUBTITLE = METRIC UNITS
DISP = ALL
$
SUBCASE 2
LABEL = CANTILEVER BEAM
LOAD = 10
SPC = 10
$
SUBCASE 5
LABEL = SIMPLY SUPPORTED BEAM
LOAD = 20
SPC = 20
$
SUBCASE 11
LABEL = LATERAL BUCKLING OF CANTILEVER BEAM
METHOD = 10
SPC = 10
STATSUB = 2
$
SUBCASE 21
LABEL = EULER BUCKLING OF SIMPLY SUPPORTED BEAM
METHOD = 10
SPC = 20
STATSUB = 5
$
BEGIN BULK
$
EIGRL  10          2
$
FORCE  10          11          -1000.  0.          1.          0.
FORCE  20          11          -1000.  1.          0.          0.
$
```

Listing 13-7 Input File for Multiple Buckling Analyses (continued)

```

CBEAM  1      1      1      2      100
CBEAM  2      1      2      3      100
CBEAM  3      1      3      4      100
CBEAM  4      1      4      5      100
CBEAM  5      1      5      6      100
CBEAM  6      1      6      7      100
CBEAM  7      1      7      8      100
CBEAM  8      1      8      9      100
CBEAM  9      1      9     10      100
CBEAM 10      1     10     11      100
$
GRID   1              0.      0.      0.
GRID   2              1.      0.      0.
GRID   3              2.      0.      0.
GRID   4              3.      0.      0.
GRID   5              4.      0.      0.
GRID   6              5.      0.      0.
GRID   7              6.      0.      0.
GRID   8              7.      0.      0.
GRID   9              8.      0.      0.
GRID  10              9.      0.      0.
GRID  11             10.      0.      0.
GRID  100             0.      0.     10.
$
MAT1    1      7.1+10      .33     2700.
$
$      RECTANGULAR SECTION OF DIMENSION .1m x .02m
$
PBEAM  1      1      .002  6.667-81.667-6     2.332-7
$
SPC1    10      123456  1
$
SPC1   20      123    1
SPC1   20      23     11
SPC    20      1      4
$
ENDDATA

```

Restarts

- Introduction to Restarts
- Types of Restarts
- Structure of the MSC.Nastran Input File
- User Interface for Read-Write Restart
- Determining the Version for a Restart
- Read-Only Restarts
- Mini-Database Restarts
- Miscellaneous Applications

14.1 Introduction to Restarts

A restart is a logical way of continuing from a previous run without having to start from the beginning. The savings can be in terms of both time and money. You use some forms of restarts practically every day in your life--perhaps without realizing it. An example of a restart can be as simple as reading a book. Normally, you probably do not finish reading a book in one continuous stretch. You may read a hundred pages today and another fifty pages tomorrow, and so on. Each time that you continue from where you left off is a restart. It is much more time consuming and sometimes impractical to start from page one every time that you pick up the book.

This analogy can be applied to MSC.Nastran. In the case of a static analysis, the most expensive and time-consuming part of the run is the decomposition of the stiffness matrix. This fact is especially true for large models. Now suppose after completing the original run, you want to obtain additional output (e.g., stresses, displacements, etc.) or add more load cases. You can always start from the beginning and redo the whole problem, or you can perform a restart at a fraction of the time and cost. In the case of additional output requests and additional load conditions, the decomposition of the stiffness matrix, which was performed in the previous run, is not redone if the restart feature is used.

However, a restart may not always be the best option compared to rerunning the job. Determining whether to save the database for restarting or rerun the analysis is based on the following factors:

- The amount of available disk space.

- Computer speed.

- The types of restarts.

- The size of your model.

Note that the database can be copied to tape, which then provides you with more free disk space. When you are ready to perform a restart, this database can then be copied from the tape back to your disk.

The discussion of restarts is divided into the following sections:

- Types of restarts.

- Structure of the MSC.Nastran input file.

- User interface.

- Determining the version for a restart.

- Miscellaneous applications.

14.2 Types of Restarts

There are two types of restarts available in MSC.Nastran: automatic and manual. With automatic restarts, MSC.Nastran automatically determines which operations need to be performed during the current run based on the changes that are made in your input file. With manual restarts, in addition to providing the necessary input file changes, you are also responsible for telling MSC.Nastran which major operations need to be performed. As you can see, the manual restart is more difficult to use and is more prone to user error. Restarts can use the same solution sequence or a different solution sequence. Restarting from a statics run and requesting stress output is an example of a restart using the same solution sequence. Restarting from a statics run into a normal modes run is an example of a restart across solution sequences. Restarts across solution sequences are beyond the scope of this user's guide. Only automatic restarts are discussed in this chapter. Within automatic restart, it also can be broken down to read-write restart ("[User Interface for Read-Write Restart](#)" on page 525) and read-only restart ("[Read-Only Restarts](#)" on page 537).

14.3 Structure of the MSC.Nastran Input File

Before going into details on how restarts work, a simple review of the MSC.Nastran input file structure may be beneficial.

NASTRAN statements	Optional

This section modifies specific Executive System default parameters. The example problem in “[File Management Section](#)” on page 545 illustrates the use of NASTRAN statements.

File Management statements	Optional

This section attaches and initializes MSC.Nastran DBsets and FORTRAN files. The file management statements are described in detail in “[Database Management](#)” on page 541.

Executive Control statements

This section selects the solution sequence, limits the time allowed, and turns on various MSC.Nastran system diagnostics.

CEND	Required Delimiter

Case Control commands

This section selects the constraints and loads from the Bulk Data entries. It also controls the type of output requests.

BEGIN BULK	Required Delimiter

Bulk Data entries

This section contains entries that describe the model geometry, connectivity, physical properties, loads, and constraints. Some entries, such as constraints and loads, are selected by the appropriate Case Control commands.

ENDDATA	Required Delimiter

This order must be followed as shown for all MSC.Nastran input files. Starting in Version 69, the NASTRAN statements can be placed in the File Management Section. For more information regarding the MSC.Nastran input file structure, see “[Organization of MSC.Nastran Files](#)” on page 19.

14.4 User Interface for Read-Write Restart

The two types of runs, cold start and restart, are described below:

Cold Start Run

The first run, which is called the cold start run, is identical to what you usually do for submitting an MSC.Nastran job with the exception that the database must be saved. In addition to your normal output files (e.g., the .F06 file), four database files are created as a result of this run. The naming convention for the filenames is machine dependent—you should refer to [MSC.Nastran 2003 Installation and Operations Guide](#) for the exact syntax for your machine. For a typical UNIX machine with an MSC.Nastran input file called “stat1.dat”, the following sample submittal command can be used:

```
nastran stat1 scr=no
```

In this case, nastran is the name of the shell script for executing MSC.Nastran. The “scr=no” option in the above submittal command is not required if you have not modified the default value of the submittal command as provided on the delivery media. By default, the following two database files (see “[Overview of the MSC.Nastran Input Files](#)” on page 21) are created as a result of the above command:

```
stat1.DBALL
stat1.MASTER
```

Restart Run

NASTRAN Statement Section

This section is normally the same as your cold start run. Note that the BUFFSIZE must not be changed in a restart run. See “[Database Management](#)” on page 541 regarding the NASTRAN statements.

File Management Section (FMS)

This section tells MSC.Nastran that you are performing a restart run. The RESTART statement is required in any restart run. The general format for the RESTART statement is as follows:

```
RESTART VERSION=a, b
```

In the above statement, “a” is the version from which you restart (the default value for “a” is LAST); “b” indicates whether version “a” will be kept (KEEP) or deleted (NOKEEP) at the end of the run. The default value for “b” is NOKEEP. Due to the default values, the following two restart statements are identical:

```
RESTART VERSION=LAST, NOKEEP
RESTART
```

Whenever a restart is performed, a new version number is automatically created in the database. For each restart, the current version number is incremented by one, regardless of whether the job ran successfully or not. Note that there are two exceptions to this rule (which are discussed later on in the chapter).

Restart--Method 1. You also need to tell MSC.Nastran the database that you want to attach to your current run. There are two ways of accomplishing this goal. One way is to use the ASSIGN statement. For example, if you are restarting from the database created by stat1.dat, the following FMS statements can be used in your current run.

```
RESTART
assign master='stat1.MASTER'
$
$ the following statement is optional
$
assign dball='stat1.DBALL'
```

For UNIX machines, the filenames are case sensitive. They should be entered exactly as they were created in the cold start run and enclosed with a single quotes as shown above. Note that the use of the second ASSIGN statement is optional since the “master” DBset can find the associated DBALL database.

Restart--Method 2. An alternate way to attach a restart database is to use the DBS keyword on the submittal command instead of the ASSIGN statement. Assuming the current run is called stat2.dat, then the equivalent submittal command is as follows:

```
nastran stat2 dbs=stat1
```

The ASSIGN statement is not needed in this case; however, the RESTART statement is needed using either option. While the first method requires more input than the second method, it also provides you with information about the database used for the restart run although this information is also provided in the Execution Summary Table (.F04 for most machines).

Executive Control Section

This section is the same as your normal run.

Case Control Section

The automatic restart logic compares the modifications made to the Case Control and Bulk Data Sections in determining which operations need to be processed upon restart. Therefore, you must be very careful in the changes that you make in your restart run. Adhering to the following rules will avoid unnecessary reprocessing of previously completed operations:

1. You must include in your restart run all solution-type related Case Control commands that are unchanged as compared to the cold start run. In other words, do not make unnecessary load, SPC, or MPC command changes or remove them from the Case Control Section unless these are actual changes. This process is clarified later on with the example problems.

2. Output requests can be modified. A typical example can be a request of the element stress output that was not requested in the cold start run.

Bulk Data Section

As mentioned in the previous section, the automatic restart logic compares the changes made in the Bulk Data Section and determines the solution path to follow. A copy of the Bulk Data is stored for each version. The restart run must not contain any Bulk Data entries that were included with the previous version that you are restarting from since they are already saved in the database for that particular version. The Bulk Data Section in the current restart run should only contain the addition of new entries, the modification of old entries, or the deletion of old entries from the database. The following is a series of three runs illustrating an application of a restart. The first run ([Listing 14-1](#)) is a sample statics run with one loading condition, and it requests displacement output.

Listing 14-1 Input File for a Cold Start Run

```
$
$ filename - stat1.dat
$
ID ROD,TEST
SOL 101
TIME 20
DIAG 8
CEND
TITLE= COLD START RUN
$
SPC = 1
$
SUBCASE 1
LOAD = 1
DISP = ALL
$
BEGIN BULK
$
GRID,1,,0.,0.,0.
GRID,2,,5.,0.,0.
GRID,3,,10.,0.,0.
CROD,1,1,1,2
CROD,2,1,2,3
PROD,1,1,.2
MAT1,1,1.+7,,.32
FORCE,1,2,,1000.,1.,0.,0.
SPC1,1,123456,1
PARAM,AUTOSPC,YES
$
ENDDATA
```

The second run ([Listing 14-2](#)) is a restart run with an additional load condition. Displacement and stress outputs are desired for this new loading condition. Furthermore, stress output for all the elements is desired for Subcase 1. However, since you already printed the displacement for Subcase 1 in your original run, you may elect not to reprint this information by removing the displacement request from Subcase 1. There are no model changes as compared to the original run. Note that the Bulk Data entries that were present in the original run must not be included in the current run. The only entry in the

Bulk Data Section is the new FORCE entry for Subcase 2, which was not included in the original run. Since this load is a new loading condition, the FORCE entry must have a unique ID (2 in this case) as compared to the existing FORCE entry ID (1 in this case) that is stored in the database.

In the FMS Section, only two statements are needed. They are the “restart” statement to tell MSC.Nastran that this current run is a restart run and the “assign” statement that tells MSC.Nastran the name of the database you want to attach. This type of restart run constitutes an efficient restart since the re-decomposition of the stiffness matrix is not performed; for a large problem, this can result in substantial CPU savings.

Listing 14-2 Input File for a Restart Run with an Additional Load Case

```
$
$ filename - stat2.dat
$
restart
assign master='stat1.MASTER'
ID ROD,TEST
SOL 101
TIME 20
DIAG 8
CEND
TITLE= RESTART RUN
$
SPC = 1
$
SUBCASE 1
LABEL = ADDITIONAL STRESS OUTPUT REQUEST
LOAD = 1
STRESS = ALL
$
SUBCASE 2
LABEL = ADDITIONAL LOAD CASE
LOAD = 2
DISP = ALL
STRESS = ALL
$
BEGIN BULK
$
$
$ NOTE THAT THE FORCE ID IS UNIQUE
$ WITH RESPECT TO EXISTING LOAD ID IN
$ THE DATABASE.
$
FORCE,2,2,, -500.,1.,0.,0.
$
ENDDATA
```

The third run ([Listing 14-3](#)) is a restart run with a modification in the material property. The same two loading conditions are used as run number two. A new material entry is added in this run and the old material entry (the eighth sorted Bulk Data from the stat2.dat run) is deleted. The general format for deleting a range of entries from the database is

$$/, x, y$$

where x and y are the range of the first and last sorted Bulk Data count to be deleted from the database, respectively. You can have as many of these entries as you need. If you are only deleting one entry ($y=x$), then either one of the following two formats can be used:

$$\begin{aligned} &/, x, x \\ &/, x \end{aligned}$$

The sorted Bulk Data is output by default. If the ECHO Case Control command is specified in the cold start run, it should be set to output the sorted Bulk Data (e.g., ECHO=SORT).

There may be times when you may be making substantial changes to the model. In this case, it may be easier to delete the whole Bulk Data listing from the database and insert a complete new Bulk Data file as shown below:

```
/, 1, z
$
$   include a complete new bulk data listing
$
```

In this case, z stands for an integer larger than the total number of Bulk Data entries (e.g., 999999). As an alternative method, you may consider deleting the database files and run the problem as a cold start run.

Listing 14-3 Input File for a Restart Run with Modified Material Property Entry

```
$  
$ filename - stat3.dat  
$  
restart  
assign master='stat1.MASTER'  
ID ROD,TEST  
SOL 101  
TIME 20  
DIAG 8  
CEND  
TITLE= RESTART RUN  
$  
SPC = 1  
$  
SUBCASE 1  
LABEL = DISPLACEMENT AND STRESS OUTPUT REQUEST  
LOAD = 1  
DISP = ALL  
STRESS = ALL  
$  
SUBCASE 2  
LABEL = ADDITIONAL LOAD CASE  
LOAD = 2  
DISP = ALL  
STRESS = ALL  
$  
BEGIN BULK  
$  
/,8  
mat1,1,3.+7,,.32  
$  
ENDDATA
```

In this example, the old MAT1 entry is the eighth sorted Bulk Data (from run stat2.dat) as shown by the following partial output:

Listing 14-4 Sorted Bulk Data Listing from a Cold Start Run

		S O R T E D B U L K D A T A E C H O													
CARD COUNT		.	1	..	2	..	3	..	4	..	5	..	6	..	7
1-	CROD		1		1		1		2						
2-	CROD		2		1		2		3						
3-	FORCE		1		2				1000.		1.		0.		
4-	FORCE		2		2				-500.		1.		0.		
5-	GRID		1				0.		0.		0.				
6-	GRID		2				5.		0.		0.				
7-	GRID		3				10.		0.		0.				
8-	MAT1		1		1.+7				.32						
9-	PARAM				AUTOSPC		YES								
10-	PROD		1		1		.2								
11-	SPC1		1		123456		1								
	ENDDATA														
	TOTAL COUNT=						12								

This type of restart (run number 3) requires the reformulation of the stiffness matrix since the material was changed. For a nonsuperelement run, this type of restart does not save you much in terms of runtime, if any. This run (run number 3) is included merely to show you how to delete information from the database. However, for superelement type runs, this type of restart can still save you time since you may only be resolving a portion of the structure. The use of superelements is beyond the scope of this user's guide.

The model used in the above series of runs is quite simple. In most cases, such simple models do not warrant the use of a restart since the savings are insignificant for such a small model. This simple example is chosen to illustrate the mechanics of how simple the restart feature is without being distracted by the details of a complex model.

The same restart procedure can now be applied to a larger model. The model used is "BCLL16". "BCLL16" is one of the standard test problems that is run for each new version of MSC.Nastran. It is basically a $16 \times 16 \times 16$ cube of CHEXA elements overlaid with a CQUAD4 elements on the face of each CHEXA element to simulate a relatively dense stiffness matrix. The model consists of 29,400 degrees of freedom. The following table illustrates the benefit of using restarts for large models.

Table 14-1 BCLL16 Performance Summary

Run Number	Description	CPU Time (Sec)
1	Static analysis with one load case and the following output request:	372.1
	SET n=a,b,c,d DISP=n STRESS (PLOT)=All	
2	After the completion of run number 1, you now decide that you want additional printed stress output for a few selective elements. One option is to rerun the same job from the beginning with the following modified output request in the Case Control Section: SET m=x,y,z STRESS=m	364.5
3	The second option is to restart from run number 1 with the following modified output request in the Case Control Section: SET m=x,y,z STRESS=m	5.0

Note the tremendous savings in the CPU time (364.5 versus 5 seconds) between the run numbers 2 and 3 with both runs accomplishing the same goal.

The cold start and restart input files used for run numbers 1 and 3 are shown in [Listing 14-5](#) and [Listing 14-6](#).

See “[Mini-Database Restarts](#)” on page 538 for efficient method of restarts when only data recovery is desired.

Listing 14-5 Input File for a BCELL16 Cold Start Run

```
$  
$ FILENAME - BCLL16A.DAT  
$  
INIT DBALL, LOGICAL=(DBALL(200000))  
INIT SCRATCH, LOGICAL=(SCRATCH(200000))  
ID MSC, BCLL16A  
TIME 300  
SOL 101  
CEND  
TITLE = STATIC ANALYSIS OF A CELLULAR CUBE          BCLL16A  
SUBTITLE = 16 X 16 X 16 CUBE  
ECHO=NONE  
STRESS(PLOT)=ALL  
SET 100 = 30080, 40088, 70880, 80888  
DISP = 100  
LOAD=100  
SPC=10  
BEGIN BULK  
$  
$   BRING IN THE REST OF THE BULK DATA FILE  
$  
include 'bcllblk.dat'  
$  
ENDDATA
```

Listing 14-6 Input File for a BCELL16 Restart Run

```
$  
$ FILENAME - RBCLL16A.DAT  
$  
RESTART  
ASSIGN MASTER='bc116a.MASTER'  
$  
$ THE ABOVE STATEMENT ASSUMES THAT YOUR  
$ COLDSTART INPUT FILENAME AS  
$  
$ bc116a.dat  
$  
ID MSC, BCLL16A  
TIME 100  
SOL 101  
CEND  
TITLE = STATIC ANALYSIS OF A CELLULAR CUBE BCLL16A  
SUBTITLE = 16 X 16 X 16 CUBE  
ECHO=NOSORT  
SET 100 = 30080,40088,70880,80888  
set 200 = 1001,2001,3001,4001  
$DISP = 100  
stress = 200  
LOAD=100  
SPC=10  
BEGIN BULK  
$  
$ SINCE THERE IS NO MODEL CHANGES, NO BULK  
$ DATA ENTRIES ARE NEEDED  
$  
ENDDATA
```

Note the user interface is quite straightforward considering the tremendous benefits in terms of CPU reduction.

14.5 Determining the Version for a Restart

Not all versions in the database are restartable. For each run, a message is printed near the top of the .F04 file indicating the version number of the current job. In general, if a job completes without any fatal messages and the database is saved, then that particular version is restartable. It is a good idea to keep a brief log of all the restartable versions since this is probably the most reliable source of information regarding whether a version is restartable. If a restart job failed, e.g., due to Bulk Data error, then this newly created version is not restartable, and it is indicated as such by the following error message at the bottom of the .F06 file.

```
*** USER WARNING MESSAGE 784 (XCLNUP)
VERSION =   yyy PROJECT =  "zzz"  OF THIS DATA BASE IS NOT VALID
FOR RESTART PURPOSES.

USER ACTION:
SUBSEQUENT RESTARTS SHOULD REFERENCE VERSION =   xxx OR A PRIOR
VALID VERSION
```

In this case, you must restart from a previous valid version. The “xxx” and “yyy” above denote version numbers. The “zzz” denotes a project description provided by you. This project description is alphanumeric and can contain up to 40 characters with the default being blank. The project description is rarely used and is an optional statement.

If for some reason the records for the old runs are no longer available, then the DBDIR FMS statement can be used to query the database contents to find out which versions are stored in the database. The following simple setup is all that is required for this purpose:

```
ASSIGN MASTER='ddddd.MASTER'
DBDIR VERSION=*, PROJECT=*
ENDJOB
```

Neither the Executive, Case Control, nor Bulk Data Section is required in this case. Furthermore, a new version is not created in this case because the ENDJOB statement is present. Near the top of the .F06 output, a PROJECT VERSION DIRECTORY TABLE is printed listing all the versions in the database. A “**” next to a version number indicates that this particular version was deleted from the database. This deletion may be due to the “NOKEEP” option or the use of the DBCLEAN statement when performing a restart run. A version number with a “**” next to it is not restartable. A version number without a “**” next to it is restartable if the run that created the version did not terminate with UFM 784. This DBDIR FMS statement can also be used to check the database directory for the existence of data blocks (e.g., UG). If the job fails very early in the run (e.g., error in the FMS section), then a new version may not be created.

It is always a good practice to back up the database on a regular basis. If the system aborts the run (e.g., disk space is exhausted or the time quota is exceeded on a system level), then there is a chance that the database is corrupted. Another good practice, which ensures that only good models are retained in the database, is to perform the following:

1. Use RESTART VERSION=a,KEEP
where “a” is a version number.
2. If a version contains errors, or is no longer of interest, then you can use the FMS statement DBCLEAN to remove obsolete or incorrect versions of the model from the database. Using DBCLEAN allows the Executive System to reuse some of this space for new versions. The DBCLEAN statement is discussed in detail in **“Database Management”** on page 541.

14.6 Read-Only Restarts

Read-only restart is very similar to read-write restart as described in “[User Interface for Read-Write Restart](#)” on page 525 with one exception: The original database is accessed in a read-only mode. No new data is added to the existing database. Instead, all new data is written to the new database, leaving the original database intact. The new database can be deleted, or it can be saved for restart purposes. The new database created during the restart run will retain an internal link to the original database.

To use read-only restart for the same example problem as in [Listing 14-2](#), all you need to do is modify the FMS statements from:

Restart

Assign master='stat1.MASTER'

to:

Restart logical=run2

Assign run2='stat1.MASTER'

In this case, run2 is an arbitrary logical name.

14.7 Mini-Database Restarts

Starting in Version 69, it is possible to save on the database only those data blocks that are required for data recovery. The size of the database, called a mini-database, is on average about 25% the size of a full database. For pure data recovery run (no spc or loading changes), this type of restart is very efficient.

To use this feature, use the following submittal command

```
nastran jobname scr=mini
```

in the cold start run.

In the restart run, no special input is required because MSC.Nastran automatically detects a mini-databases and will process any output request from the table below.

The mini-database restart processes the following Case Control commands and user parameters in SOL 101:

DISPLACEMENTSTRAIN	
ELSDCON	STRFIELD
ESE	STRESS
FORCE	SUBCOM
GPFORCE	SUBSEQ
GPSDCON	SURFACE
GPSTRESS	SYMCOM
MPCFORCES	SYMSEQ
PARAM,POST	THERMAL
REPCASE	VOLUME
SPCFORCES	

For cyclic symmetry analysis (see “[Cyclic Symmetry](#)” on page 559) the HOUTPUT and NOUTPUT cannot be changed in the restart run.

14.8 Miscellaneous Applications

Aside from the conventional restarts described in the previous sections, there are other applications in which restarts can be used.

Bulk Data Listing

There may be occasions when for some reason an old input file is no longer available, but the database is still archived somewhere. The following sample run ([Listing 14-7](#)) allows you to either print or punch a copy of the Bulk Data from the database. The term “punch” is an old term from the earlier days when punched cards were generated. The “punch output” in this case actually refers to an output file, and this file can subsequently be used as an input file.

Listing 14-7 Input File for Printing/Punching a Listing of Bulk Data File from the Database

```
$  
$ FILE - bulklist.dat  
$  
restart  
assign master='archive1.MASTER'  
sol 101  
time 10  
cend  
title = run to get listing of bulk data  
echo = sort  
$  
$ replace echo=sort with echo=punch if a  
$ punched output is desired instead of a listing  
$  
param,serst,semi  
begin bulk  
$  
enddata
```

Summary

By now you can see the advantages of automatic restarts. They are easy to use, and they can potentially save you a substantial amount of time--both CPU and elapsed “wallclock” time--by not having to resolve a problem from the beginning. One disadvantage is that you have to save the database. However, this database can be downloaded to tape or deleted when it is no longer needed. Using restarts also requires a bit more bookkeeping.

For conventional static analysis runs (i.e., those not involving superelements), if a restart involves model changes, the savings is probably minimal. However, in the case of additional output requests or additional load cases, the savings can be substantial as indicated by [Table 14-1](#).

CHAPTER

15

Database Management

- Introduction to Database Management
- Definitions
- MSC.Nastran Database
- File Management Section
- Guidelines for Large Problems

15.1 Introduction to Database Management

This chapter is intended to give you an overview of how the database and the File Management Section (FMS) work so that you can allocate your resources efficiently, especially for large models. For many problems, you may not need to concern yourself with the knowledge or use of the FMS--in such cases, MSC.Nastran uses built-in, default values to handle database allocation and file assignments. However, there are cases in which knowledge of the FMS may be required.

This chapter is divided into the following sections:

- Definitions.
- MSC.Nastran database.
- File Management Section.
- Guidelines for large problems.

15.2 Definitions

Before presenting the details of the database structure, the definitions for some of the basic terms that are used throughout this chapter need to be presented.

DBset	Database set. DBset consists of an MSC.Nastran logical name, such as MASTER, DBALL, etc., which refers to an entity within the MSC.Nastran database. It consists of a collection of physical files.
Database	The collection of all DBsets assigned to a run.
Data block	A matrix or table (e.g., KAA, LAMA) that is stored in the database.
Logical name	The local internal name (log-name) used by MSC.Nastran for a file or DBset.
Word	For 32- and 64-bit machines, each word is equivalent to four and eight bytes, respectively.
BUFSIZE	The length of an MSC.Nastran buffer in terms of words (32- or 64-bit words). It controls the physical record size for data storage / transfer that is contained in many MSC.Nastran logical units. The default and maximum allowable BUFSIZE is machine dependent. The default value is recommended except for large problems. It can be modified by using the following NASTRAN statement: <code>NASTRAN BUFSIZE = xxxxx</code>
Block	A block is often referred to as an MSC.Nastran GINO block. Each GINO block contains one MSC.Nastran buffer. The size of each block is equal to BUFSIZE-1 words. (GINO is an acronym for general input/output.)
{ }	A brace indicates that the quantity within this bracket is mandatory. The underlined item within { } is the default value.
[]	A square bracket indicates that the quantity within this bracket is optional.
_____	The underlined value is the default value for a particular option.

15.3 MSC.Nastran Database

When submitting a MSC.Nastran job, a series of standard output files is created (e.g., the .f06 file). The naming convention for the filenames is machine dependent. Furthermore, by default, two permanent DBsets (MASTER, DBALL) and one scratch (SCRATCH) DBset are created during the run.

MASTER	This is the “directory” DBset that contains a list of all the DBsets used in the job, all the physical file names assigned, and an index pointing to all the data blocks created and where they are stored. In addition, it also contains the MSC.Nastran data definition language (NDDL). The NDDL is the internal MSC.Nastran language that describes the database. You do not need to understand the NDDL in order to use MSC.Nastran effectively. The default maximum size for MASTER is 5,000 blocks.
DBALL	The DBset where the permanent data blocks are stored by default. The default size is 250,000 blocks.
SCRATCH	The DBset used as the temporary workspace for MSC.Nastran. In general, this DBset is deleted at the end of the run. The default size is 250,000 blocks.
SCR300	The DBset used as the temporary workspace for the modules. This space is released at the end of the module execution to be reused by subsequent modules. The default size is 250,000 blocks.

For a typical UNIX-based workstation with an MSC.Nastran input file called “stat1.dat”, the following sample submittal command can be used:

```
nastran stat1 scr=no
```

In the command above, nastran is the name of the shell script used to execute MSC.Nastran. The following two permanent physical database files are created as a result of the above command.

```
stat1.MASTER  
stat1.DBALL
```

Note that the temporary scratch files are allocated and used by MSC.Nastran during the course of the run and are deleted at the end of the run. Unless otherwise stated, the input filename is assumed to be “stat1.dat” in this chapter.

15.4 File Management Section

The File Management Section (FMS) is intended primarily for the attachment and initialization of MSC.Nastran database sets (DBsets) and FORTRAN files. For many problems due to the default values, the FMS is handled automatically and is not required to be specified by you. At first glance, the FMS may be overwhelming, which is due in part to its flexibility and the many options it has to offer. However, in its most commonly used form, it can be quite simple. Examples of its most commonly used forms are provided throughout this chapter.

You can allocate the database file size in any of the following units:

- Bytes
- Words
- Megabytes (1,000,000 bytes)
- Megawords (1,000,000 words)
- Kilobytes (1,000 bytes)
- Kilowords (1,000 words)
- MSC.Nastran GINO blocks (default)
- Mixture of the above units

Internally, MSC.Nastran converts them to GINO blocks. There are about 20 FMS statements available in MSC.Nastran. This section covers the commonly used FMS statements in static analysis and each is presented in its most commonly used format, which in general, is a simplified format.

If an FMS statement is longer than 72 characters, it must be continued on the next line. You can have a maximum of 200 files (FORTRAN and DBsets) assigned (online) at a time. This limit may be less for some machines due to their operating system limit.

INIT

Purpose

Creates/initializes permanent and/or temporary DBsets. The INIT statement has two basic formats: one for all the DBsets and one specifically for the SCRATCH DBsets.

Format (Simplified) for All DBsets Except SCRATCH DBsets

```
INIT DBset-name [LOGICAL=(log-name1(max-size1) ,
log-name2(max-size2), ... log-namei(max-sizei) ,
... log-namen(max-sizen))]
```


DBset-name	The logical name of the DBset being used (e.g., DBALL).
log-name _i	The i-th logical name for the DBset-name referenced in this INIT statement. You can have up to 20 logical names for each DBset ($1 \leq n \leq 20$). An ASSIGN statement may be used to attach a physical file to the i-th logical file.
max-size _i	The maximum allowable size that may be written to the i-th file.

Example A

The following statement creates the DBALL DBset with a logical name of DBALL and the maximum size of 50,000 MSC.Nastran blocks instead of 250,000 blocks, which is the default value for DBALL:

```
INIT DBALL LOGICAL=(DBALL(50000))
```

Assuming that your input file is called “stat1.dat”, the physical file has the name stat1.DBALL unless an ASSIGN statement is also used.

The following statement creates the DBALL DBset with logical names of DB1 and DBTWO:

```
INIT DBALL LOGICAL=(DB1(35000),DBTWO(60000))
```

The two physical files stat1.DB1 and stat1.DBTWO are created with a maximum of 35,000 and 60,000 MSC.Nastran blocks, respectively.

Format (Simplified) for the SCRATCH DBset

```
INIT SCRATCH [LOGICAL=(log-name1(max-size1) ,
log-name2(max-size2), ... log-namei(max-sizei)) ,
SCR300=(log-namei+1(max-sizei+1), ... log-namen(max-sizen))]
```

Log-name₁ through log-name_i are allocated for regular scratch files as temporary workspace. This space is not released until the end of the job. SCR300 is a special keyword that indicates the log-names are members reserved for DMAP module internal scratch files. The space occupied by these files is for the duration of the execution of the module. This space is released at the end of the module execution. You can have up to a combined total of 20 logical names for the SCRATCH DBset ($1 \leq n \leq 20$).

Example B

The following statement creates the SCRATCH DBset with logical names of SCR1, SCR2, SCRA, and SCRB:

```
INIT SCRATCH LOGICAL=(SCR1(150MW), SCR2(100MW)) ,
SCR300=(SCRA(250MW), SCRB(300MW))
```

The two physical files stat1.SCR1 and stat1.SCR2 are created with maximum sizes of 150 and 100 megawords, respectively. These two files are regular scratch files. Two additional physical files stat1.SCRA and stat1.SCRB are created with maximum sizes of 250 and 300 megawords, respectively. These last two files are SCR300-type files.

ASSIGN

Purpose

Assigns physical filenames to logical filenames or special FORTRAN files that are used by other FMS statements or DMAP modules.

Format (Simplified) to Assign Logical Files

```
ASSIGN log-namei='filenamei' [TEMP DELETE]
```

log-namei	The i-th logical name for the DBset created by the INIT statement.
TEMP	Requests that filenamei be deleted at the end of the job.
DELETE	Requests that filenamei be deleted if it exists before the start of the run. This is optional; however, if this option is not used and the file exists prior to the current run, then the job may fail with the following messages:

```
*** USER FATAL MESSAGE 773 (DBDEF)
    THE FOLLOWING PHYSICAL FILE ALREADY EXISTS.
    LOGICAL NAME   = xxxx
    PHYSICAL FILE  = yyyy
    USER INFORMATION: NO ASSOCIATED DEFAULT FILES OR ASSIGNED DBSETS CAN
    EXIST PRIOR TO THE DATA BASE INITIALIZATION RUN.
    USER ACTION: DELETE THIS FILE AND RESUBMIT THE JOB.
```

DELETE is not a suggested option if you are using RESTART since you can delete your database inadvertently. Manual deletion of unwanted databases is a safer approach.

Example C

The following statements create a logical name DB1 for the DBset DBALL in the current directory:

```
ASSIGN DB1='sample.DB1'
INIT DBALL LOGICAL=(DB1(50000))
```

The physical file sample.DB1 is created in this case. Without the ASSIGN statement, the physical filename created is called stat1.DB1, assuming once again that your input file is called stat1.dat.

The following statements create the two logical names DB1 and DB2 for the DBset DBALL:

```
ASSIGN DB1='/mydisk1/se/sample.DB1'
ASSIGN DB2='/mydisk2/sample.DB2'
INIT DBALL LOGICAL=(DB1(50000),DB2(40000))
```

DB1 points to a physical file called sample.DB1 that resides in the file system (directory) /mydisk1/se. DB2 points to a physical file called sample.DB2 that resides in the file system (directory) /mydisk2.

Format (Simplified) to Assign FORTRAN Files

```
ASSIGN log-key='filenamef' [STATUS={new,old},UNIT=u ,  
FORM={FORMATTED, UNFORMATTED}, DELETE]
```

log-key This is the logical keyword for the FORTRAN file being assigned. This file may already exist. The default value depends on the keyword. Acceptable keywords are

DBC, DBMIG, INPUTT2, INPUTT4, OUTPUT2, OUTPUT4,
DBUNLOAD, DBLOAD, and USERFILE.

See the [MSC.Nastran Quick Reference Guide](#) for detailed descriptions of these keywords and their default values.

filenamef	The physical name of the FORTRAN file.
STATUS	Specifies whether the FORTRAN file is created (STATUS = new) or is an existing file (STATUS = old).
UNIT	Specifies the FORTRAN unit (e.g., UNIT = 12).
FORM	Specifies whether the file written is in ASCII (FORM = FORMATTED) or binary (FORM = UNFORMATTED) format. Note that if the log-key is OUTPUT2, the FORMATTED output is in compressed ASCII format.
DELETE	Requests that filenamef be deleted, if it exists before the start of the run.

Example D

The following example creates a new FORTRAN file to be used for OUTPUT2 operations:

```
ASSIGN OUTPUT2='sample.out', STATUS=NEW, UNIT=11, FORM=FORMATTED
```

This FORTRAN file is in compressed ASCII format with a filename of sample.out and is assigned to unit 11. Note that a compressed ASCII file can be transferred directly across machines, but it should not be edited. For more information regarding the transfer of OUTPUT2 files across machines, see the [MSC.Nastran Common Questions and Answers](#).

EXPAND

Purpose

Concatenates files into an existing DBset in order to increase the allowable disk space available for MSC.Nastran. The EXPAND statement is normally used in a restart run when you exceed the disk space allocation in your previous run.

Format

```
EXPAND DBset-name LOGICAL=(log-namei(max-sizei), ... )
```

DBset-name	The logical name of the DBset to be expanded by the addition of new members to an existing DBset previously defined with an INIT statement.
log-name <i>i</i>	The logical name of the <i>i</i> -th member of the DBset. An ASSIGN statement should be used to point this logical name to a physical file.
max-size <i>i</i>	The maximum size of the <i>i</i> -th member.

Only one EXPAND statement is allowed per run.

Example E

The original run creates a database with the name stat1.DBALL. However, this database was filled and the job failed with the following error messages in the .F06 file:

```
*** USER FATAL MESSAGE 1012 (GALLOC)
DBSET DBALL IS FULL AND NEEDS TO BE EXPANDED.
```

For small to medium problems, it is best to rerun the job from the beginning with a larger file allocation. For large problems, if rerunning the job is not practical, then the database can be expanded with the following FMS statements:

```
RESTART
ASSIGN MASTER='stat1.MASTER'
ASSIGN DBADD='morespace.DB'
EXPAND DBALL LOGICAL=(DBADD(50000))
```

These statements assign an additional member, with a logical name of DBADD, to the existing DBset DBALL. This member points to a new physical file called morespace.DB, which may contain up to a maximum of 50,000 MSC.Nastran blocks. You are restarting from “stat1.MASTER” in this case.

The EXPAND statement cannot be used for the scratch files since they are deleted at the end of each job.

RESTART

Purpose

Allows you to continue from the end of the previous run without solving the problem from the beginning.

Format

```
RESTART [PROJECT='proj-ID', VERSION={version-ID, LAST},
{KEEP, NOKEEP}]
```

proj-ID	Project identifier used in the original run. It can have up to 40 characters. This option is optional and is normally not used. The default proj-ID is blank.
version-ID	The version number that you are restarting from.
KEEP	If this option is used, then the version that you are restarting from is also saved at the end of the current run.
NOKEEP	If this option is used, then the version that you are restarting from is deleted at the end of the current run.

Example F

The following statement causes the current run to use the last version in the database for the restart:

```
RESTART
```

At the end of the run, this last version that you are restarting from is deleted from the database. This is probably the most commonly used form for RESTART.

The following statement instructs the current run (version 6 or higher) to use version 5 in the database for restart:

```
RESTART VERSION=5, KEEP
```

At the end of the run, version 5 is also retained in the database. This format is used most often when you want to ensure that a specific version is saved in the database (e.g., a large run from which you may want to request additional data recovery in the future).

The following statement specifies that the current run (version 4 or higher with a proj-ID of xyz) uses version 3 with a proj-ID of xyz in the database for restart:

```
RESTART PROJ='xyz' VERSION=3
```

At the end of the run, version 3 with a proj-ID of xyz is deleted from the database.

DBCLEAN

Purpose

Deletes unwanted versions from the database. The DBCLEAN statement does not reduce the size of the database. This statement deletes the data blocks for the specified versions to allow other data blocks to reuse this space for subsequent restarts without potentially increasing the database size. Up to ten DBCLEAN statements can be used in the FMS for each run.

Format

```
DBCLEAN VERSION={version-ID, *} [PROJECT={'project-ID', *}]
```

version-ID	The version number that you want to remove from the database.
project-ID	The identifier of the project to be deleted. The default is blank, which is the most commonly used form.
*	This is a wildcard command. It removes all versions or projects from the database.

Example G

The following statements delete versions 3 and 6 with a blank proj-ID from the database in the current run:

```
DBCLEAN VERSION=3
DBCLEAN VERSION=6
```

DBDIR

Purpose

The DBDIR statement prints the database directory.

Format

```
DBDIR [VERSION={version-ID, *} ] [PROJECT={proj-ID, *} ]
      [FORMAT={format-no} ]
```

version-ID	Version number.
proj-ID	Project number.
*	This is a wildcard command. It refers to all versions or projects.
format-no	This controls the type of directory information printed. The value is the sum of the desired types listed below. The default value is 63, which means the first six types will be printed.

- 1 = Project version table
- 2 = NDDL data blocks
- 4 = NDDL parameters
- 8 = NDDL empty data blocks
- 16 = NDDL data block trailers
- 32 = NDDL path value table for NDDL entries
- 64 = Scratch data blocks

Example H

The following statement causes the printing of the database directory for all versions of the current project-ID in the database:

DBDIR

INCLUDE

Purpose

The INCLUDE statement inserts an external file at the location where the INCLUDE statement is used.

Inserts an external file at the location where this INCLUDE statement is used. It is not a pure FMS statement because it can be used anywhere in the input file, not just in the FMS section.

Format

Include 'filename'

filename Physical filename of external file to be inserted at this location.

The following run reads a file called sub1.dat with all the Case Control commands contained in it:

```
Sol 101
cend
include 'sub1.dat'
begin bulk
$
include 'bulk1.dat'
include 'bulk2.dat'
$
$          rest of bulk data file
$
.
.
enddata
```

This run also brings two additional files (bulk1.dat and bulk2.dat) into the Bulk Data Section. You may, for example, want to include all your grid entries in file bulk1.dat and all your element connectivities in bulk2.dat. As you can see, the INCLUDE statement can be a handy tool. For parametric studies, you can potentially save a tremendous amount of disk space by using the INCLUDE statement instead of having multiple files with duplicate input data.

15.5 Guidelines for Large Problems

As mentioned previously, for most small to medium problems, the default values for all the file assignments should be more than adequate. However, for a large problem, larger disk space allocation may be necessary. Several examples are shown in this section that illustrate various options and consequences with each option. The submittal command for each example is shown followed by a partial listing of the corresponding input file. In each case, it is assumed that you are running on a UNIX machine with “nastran” as the name of the shell script executing MSC.Nastran.

Example I

The following submittal command and FMS statements allocate 300,000 and 5,000 MSC.Nastran GINO blocks for DBALL and MASTER, respectively:

```

nastran runa scr=no
$
$ filename - runa.dat
$
INIT DBALL, LOGICAL=(DBALL(300000))
INIT SCRATCH, LOGICAL=(SCRATCH(200000)) ,
    SCR300=(SCR300(200000))
$
$ THE REST OF YOUR INPUT FILE
$
```

Furthermore, the following physical files are created and saved in your current directory:

```

runa.DBALL
runa.MASTER
```

These statements also allocate 200,000 and 200,000 MSC.Nastran GINO blocks for SCRATCH and SCR300, respectively. These two files are allocated to the default scratch directory during the course of the run and are deleted at the end of the run. Since the database is saved, this job is restartable. Other than the allocation size of the databases, this is identical to the default setup.

When using the “scr = yes” option on the submittal command, the permanent data blocks that are normally written to the DBALL DBset are now written to the SCRATCH DBset.

Example J

The submittal command and FMS statements below allocate 50, 500, and 800 megabytes for DBALL, SCRATCH, and SCR300, respectively. Furthermore, by default, 5,000 MSC.Nastran GINO blocks are allocated each for the MASTER dbset. During the course of the run, all of these files are allocated to your default scratch directory. Because you are using the “scr=yes” option on the submittal command, the permanent data blocks that are normally written to the DBALL DBset are now written to the SCRATCH DBset. In other words, you must allocate the disk space to the SCRATCH DBset, which you would normally allocate to the DBALL DBset, when you use the “scr=yes” option on the submittal

command. At the end of the run, all of these files are deleted automatically; this action is different from the previous versions of MSC.Nastran. Since the database is not saved, this job is not restartable.

```
nastran runb scr=yes

$
$ filename - runb.dat
$
INIT DBALL, LOGICAL=(DBALL(50MB))
INIT SCRATCH, LOGICAL=(SCRATCH(500MB)) ,
    SCR300=(SCR300(800MB))
$
$ THE REST OF YOUR INPUT FILE
$
```

Example K

If you have sufficient disk space in your current and default scratch file systems (disk packs), the method shown in “[Example A](#)” on page 546 or “[Example B](#)” on page 546 is the preferred procedure since it is the simplest method. However, if you do not have sufficient disk space in these two file systems, but have other file systems (e.g., / disk2, / disk3, and / disk4) mounted to your system, then the procedure listed below can be used. You must have read and write privileges in order to use these file systems. For this example, the three new file systems are / disk2, / disk3, and / disk4.

```
nastran runc scr=no

$
$ filename - runc.dat
$
ASSIGN DBALL1='runc.db1'
ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrch2'
ASSIGN SC3B='/disk4/user_guide/statics/runc.sc3b'
$
INIT DBALL, LOGICAL=(DBALL1(200000KB), DBALL2(100000KB))
INIT SCRATCH, LOGICAL=(SCRATCH1(100000), SCRATCH2(100000)) ,
    SCR300=(SC3A(150000), SC3B(50000))
$
$ THE REST OF YOUR INPUT FILE
$
```

The permanent database is broken up into two separate logical files (DBALL1 and DBALL2). The above run allocates 200,000 and 100,000 kilobytes for DBALL1 and DBALL2, respectively. The run also allocates 5,000 GINO blocks for the MASTER dbset. DBALL1 is allocated to the current directory that you are running on with the physical filename of “runc.db1”. DBALL2 is allocated to the “/ disk2/ user_guide/ statics” directory with a physical filename of “runc.db2”. The following additional physical file is also created in your current directory:

runc.MASTER

These three files are saved at the end of the run and the job is restartable.

The regular SCRATCH and SCR300 files are each divided into two separate files. Two logical files (SCRATCH1 and SCRATCH2) are assigned to the regular SCRATCH file. SCRATCH1 is assigned to the default scratch directory requesting 100,000 GINO blocks. SCRATCH2 is assigned to the “/disk3/user_guide/statics” directory with a filename of “runc.scrтч2” and requesting 100,000 GINO blocks. The SCR300 file is also divided into two files (SC3A and SC3B). SC3A requests 150,000 blocks and assigns this disk space to the default scratch directory. SC3B requests 50,000 blocks and assigns this disk space to the “/disk4/user_guide/statics” directory with a filename of “runc.sc3b”. All four of these scratch files are deleted at the end of the run.

```
nastran rund scr=yes
```

Example L

Even though the setup in “[Example D](#)” on page 548 looks virtually the same as that in “[Example C](#)” on page 547, the “scr=yes” keyword causes the permanent data blocks—that are normally written to the DBALL DBset—to be written to the SCRATCH DBset. In other words, except for the amount used for overhead, the space that you have allocated to DBALL1 and DBALL2 is not used at all. You must adjust the disk space allocation to the SCRATCH DBset that you normally would allocate to the DBALL DBset when you use the “scr=yes” option on the submittal command. The correct action in this case is to increase the disk space allocation for SCRATCH1 and SCRATCH2, and decrease the disk space allocation for DBALL1 and DBALL2. Once again, since the database is not saved, this job is not restartable.

```
$
$ filename - rund.dat
$
ASSIGN DBALL1='rund.db1'
ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrтч2'
ASSIGN SC3B='/disk4/user_guide/statics/runc.sc3b'
$
INIT DBALL, LOGICAL=(DBALL1(200000KB), DBALL2(100000KB))
INIT SCRATCH, LOGICAL=(SCRATCH1(100000), SCRATCH2(100000)) ,
    SCR300=(SC3A(150000), SC3B(50000))
$
$ THE REST OF YOUR INPUT FILE
$
```

BUFFSIZE

BUFFSIZE is the length of an MSC.Nastran buffer in terms of words (32- or 64-bit words). It controls the physical record size for data storage/transfer that is contained in many MSC.Nastran logical units. The default and maximum allowable bufsize is machine dependent. The default value is recommended except for large problems. Each 32-, and

64-bit word contains 4 and 8 bytes, respectively. This feature can be invoked by including the following NASTRAN statement in your input file (“[Structure of the MSC.Nastran Input File](#)” on page 524).

NASTRAN BUFFSIZE=xxxxx

where $xxxxx = (n \cdot \text{disk-block-size}) + 1 \leq \text{limit}$ and n is a positive integer.

The acceptable values of disk-block-size and limit are machine dependent and are defined in [Table 15-1](#) for most machines.

A larger BUFFSIZE takes fewer requests of the computer operating system to transfer the same amount of data, with each transfer involving a larger physical record size. This can reduce the I/O time which, in turn, reduces the elapsed time. In general, the effect on CPU time is insignificant. A larger BUFFSIZE decreases the number of I/O operations but may increase the database size. Since each data block uses at least one buffer and the read/write operation requires a minimum of one buffer, the default BUFFSIZE is recommended except for large problems. See [MSC.Nastran 2003 Installation and Operations Guide](#) for the default BUFFSIZE for various machines.

Disk space allocation is affected by the BUFFSIZE if the disk space units are in terms of GINO blocks. GINO blocks are related to words by the following equation:

$$\text{Number of words} = \text{Number of GINO blocks} \cdot (\text{BUFFSIZE} - 1) \quad \text{Eq. 15-1}$$

“[Example E](#)” on page 549 illustrates the use of a non-default BUFFSIZE.

Example M

Assume the machine that you are running on has a default BUFFSIZE of 2049. If you increase the BUFFSIZE to 17291, using [Eq. 15-1](#), this example allocates the same maximum amount of physical disk space as “[Example A](#)” on page 546, which uses the default BUFFSIZE.

```
$
$ filename - rune.dat
$
NASTRAN BUFFSIZE=17291
INIT DBALL, LOGICAL=(DBALL(35536))
INIT SCRATCH, LOGICAL=(SCRATCH(23691)), SCR300=(SCR300(23691))
$
$ THE REST OF YOUR INPUT FILE
$
```

Assuming you have disk space, is there any other reason why you would not want to use a larger BUFFSIZE for a large problem? The answer is “no” for most machines, provided you have sufficient memory available on your machine. For some machines, optimum values are preselected, and you should use these default values. See [Reference 16](#) for further details regarding your particular machine. Total memory allocation can be controlled by the “mem” keyword on the submittal line. The following example allocates 100 mb of total memory for the run.

```
nastran stat1 mem=100mb
```

A portion of this total memory is allocated to the Executive System. The rest of the memory is then available to the functional modules in MSC.Nastran. Assuming that you are using the default SCRATCH(MEM) and BUFFERPOOL, the portion of the memory that is available for your problem is denoted as “User OPENCORE (HICORE)”, and can be estimated by [Eq. 15-2](#).

$$\text{User OPENCORE} \cong \text{mem} - (\text{BUFFSIZE} \cdot 148) - 89,315 \quad \text{Eq. 15-2}$$

As you can see from [Eq. 15-2](#), the amount of memory allocated to the Executive System is a function of the BUFFSIZE. As the BUFFSIZE increases, a larger portion of the total memory is allocated to the Executive System. A summary of the MSC.Nastran memory utilization is printed near the beginning of the .F04 file. Some typical examples of the memory summary tables are shown in [Table 15-1](#), [Table 15-2](#), and [Table 15-3](#). You can also let MSC.Nastran estimate the amount of required memory by using the following submittal command:

```
nastran stat1 mem=estimate
```

Table 15-1 Memory Summary Table
(mem = 2MW, bufsize = 2049)

** MASTER DIRECTORIES ARE LOADED IN MEMORY.

USER OPENCORE (HICORE)	=	1607959	WORDS	
EXECUTIVE SYSTEM WORK AREA	=	81854	WORDS	
MASTER(RAM)	=	30000	WORDS	
SCRATCH(MEM) AREA	=	204900	WORDS (	100 BUFFERS)
BUFFER POOL AREA (EXEC)	=	76183	WORDS (	37 BUFFERS)
TOTAL MSC/NASTRAN MEMORY LIMIT	=	2000896	WORDS	

Table 15-2 Memory Summary Table
(mem = 4MW, bufsize = 2049)

** MASTER DIRECTORIES ARE LOADED IN MEMORY.

USER OPENCORE (HICORE)	=	3607831	WORDS	
EXECUTIVE SYSTEM WORK AREA	=	81854	WORDS	
MASTER(RAM)	=	30000	WORDS	
SCRATCH(MEM) AREA	=	204900	WORDS (	100 BUFFERS)
BUFFER POOL AREA (EXEC)	=	76183	WORDS (	37 BUFFERS)
TOTAL MSC/NASTRAN MEMORY LIMIT	=	4000768	WORDS	

Table 15-3 Memory Summary Table
(mem = 4MW, bufsize = 17291)

** MASTER DIRECTORIES ARE LOADED IN MEMORY.

USER OPENCORE (HICORE)	=	1352015	WORDS	
EXECUTIVE SYSTEM WORK AREA	=	249516	WORDS	
MASTER(RAM)	=	30000	WORDS	
SCRATCH(MEM) AREA	=	1729100	WORDS (	100 BUFFERS)
BUFFER POOL AREA (EXEC)	=	640137	WORDS (	37 BUFFERS)
TOTAL MSC/NASTRAN MEMORY LIMIT	=	4000768	WORDS	

Summary

Due to the choice of the default values, very little knowledge of the MSC.Nastran FMS statements and database structure is required for small to medium problems. For large problems, however, some knowledge of the FMS statements and database structure can help you to allocate and optimize your computer resources.

Cyclic Symmetry

- Introduction to Cyclic Symmetry
- Description of Geometric Symmetry in MSC.Nastran
- Mathematical Description of Structural Symmetry
- Bulk Data Entries for Cyclic Symmetry
- Case Control Section for Cyclic Symmetry
- Axisymmetric Example

16.1 Introduction to Cyclic Symmetry

The symmetry of a structure can be exploited in order to reduce the quantity of input data and to reduce the cost of computation. If your loading has the same symmetry as the structure, only a single analysis is required for a symmetrical subregion of the structure with symmetrical boundary conditions. If your loading is unsymmetrical, the following additional tasks must be performed:

1. Separate applied loads into symmetry sets (e.g., symmetrical and antisymmetrical sets for a structure with one plane of symmetry).
2. Apply separate constraint sets to a subregion of the structure in order to model the boundary conditions corresponding to each symmetry condition.
3. Compute separate solutions for each symmetry condition.
4. Combine the results for each symmetry condition to obtain the complete solution for the complete structure.

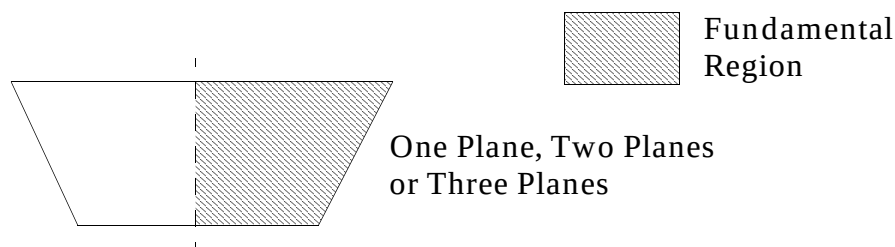
In MSC.Nastran's cyclic symmetry option, Solution 114, all of these tasks are performed automatically. This option can be used for several types of symmetry, including reflective symmetry, axisymmetry, and rotational symmetry (i.e., identical components symmetrically arranged about an axis).

MSC.Nastran also includes a way of handling the special case of unsymmetric loads on a structure with one or two planes of symmetry (discussed in “[Modeling Guidelines](#)” on page 311). With this method you must perform Tasks 1 and 2 in the above list by hand. Task 3 is accomplished by creating subcases for each symmetry condition, with different SPC sets and loading conditions for each subcase. Task 4 is accomplished with SYMCOM and SYMSEQ Case Control commands or with SUBCOM and SUBSEQ Case Control commands (see “[Organization of MSC.Nastran Files](#)” on page 19).

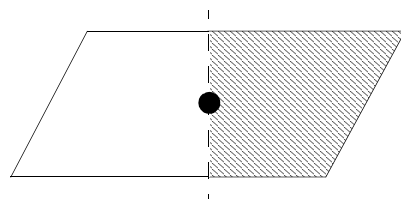
16.2 Description of Geometric Symmetry in MSC.Nastran

When an engineer speaks of a symmetrical structure, he usually means a structure that has one or more planes of reflective symmetry. Reflective symmetry is, however, only one of the several different types of geometric symmetry shown in [Figure 16-1](#).

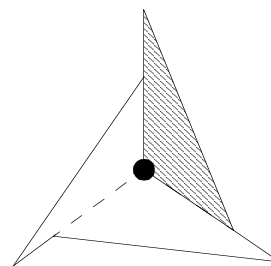
(a) Reflective Symmetry



(b) Rotational Symmetry



Two Segments



Three Segments

(c) Dihedral Symmetry

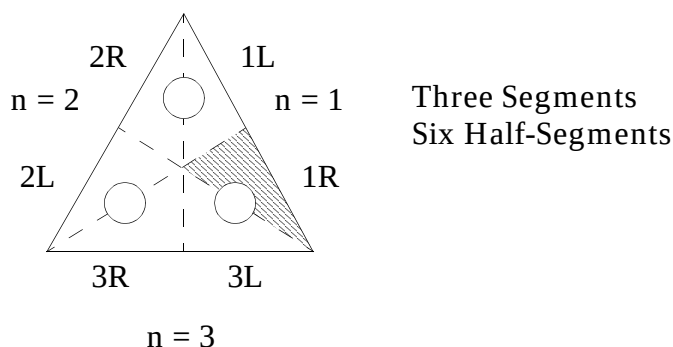
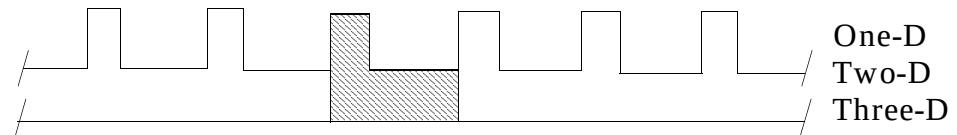


Figure 16-1 Types of Geometrical Symmetry

(d) Translational Symmetry



(e) Axisymmetry

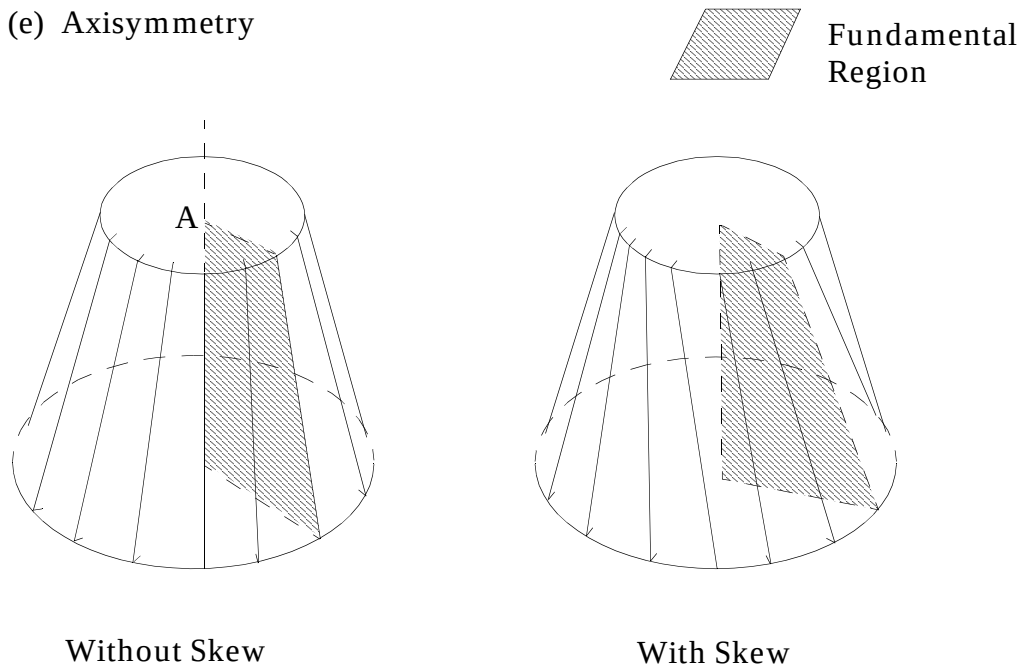


Figure 16-1 Types of Geometrical Symmetry (continued)

In rotational symmetry, [Figure 16-1\(b\)](#), the structure is made up of identical segments that are symmetrically arranged about an axis. If the structure has only two segments, the second segment is created by rotating the first segment through 180° about an axis A. In general, the n -th segment is created by rotating the fundamental region (i.e., the first segment) through $360(n-1)/N$ degrees about axis A where N is the total number of segments. Common examples of structures with rotational symmetry are skew slabs and many types of rotating machinery including gears, electric motors, turbines, and helicopter rotors.

Dihedral symmetry, [Figure 16-1\(c\)](#), is a special case of rotational symmetry that combines rotational and reflective symmetry since each segment has its own plane of reflective symmetry. Thus, in [Figure 16-1\(c\)](#), half-segment 1L is the mirror image of half-segment 1R (the fundamental region), and these two halves of segment 1 are rotated together through 120° to form segment 2 and through 240° to form segment 3. Water towers and reinforced domed roofs are familiar examples of structures with dihedral symmetry.

In translational symmetry, **Figure 16-1(d)**, the fundamental region is endlessly repeated by translation in a straight line. Translational symmetry can clearly be extended to two and three dimensions. An example of a structural problem with this type of symmetry is the analysis of the buckling of a long rail with periodic flexible supports.

In axisymmetry, **Figure 16-1(e)**, the object is generated by rotating the fundamental region continuously about an axis, thereby forming a solid of revolution. Two subclasses may be distinguished. In the first subclass, the fundamental region is a plane surface, and in the second subclass, the fundamental region is a skewed surface. These two subclasses are distinguishable only if the mechanical properties of the fundamental region are anisotropic. Examples of structures with skewed axisymmetry may be found in nature (e.g., the trunks of pine trees) and in spiral-wound reinforced shells. Examples of axisymmetry without skew are too numerous to mention.

Just as axisymmetry with skew and axisymmetry without skew are subclasses of axisymmetry, you may consider rotational symmetry, dihedral symmetry and axisymmetry to be subclasses of the general class of symmetry defined by the rotation of a fundamental region about an axis, which is called cyclic symmetry. Within this class, axisymmetry without skew is seen to be a limiting case of dihedral symmetry as the number of segments is increased without limit, and axisymmetry with skew is seen to be a limiting case of rotational symmetry. In practice, the axisymmetric case is approached with sufficient accuracy by having a large, but finite, number of segments.

Also, reflective symmetry with one or two planes of symmetry is a special case of dihedral symmetry. As illustrated in **Figure 16-2**, reflective symmetry with one plane of symmetry is identical to dihedral symmetry with one segment, and reflective symmetry with two planes of symmetry is identical to dihedral symmetry with two segments.

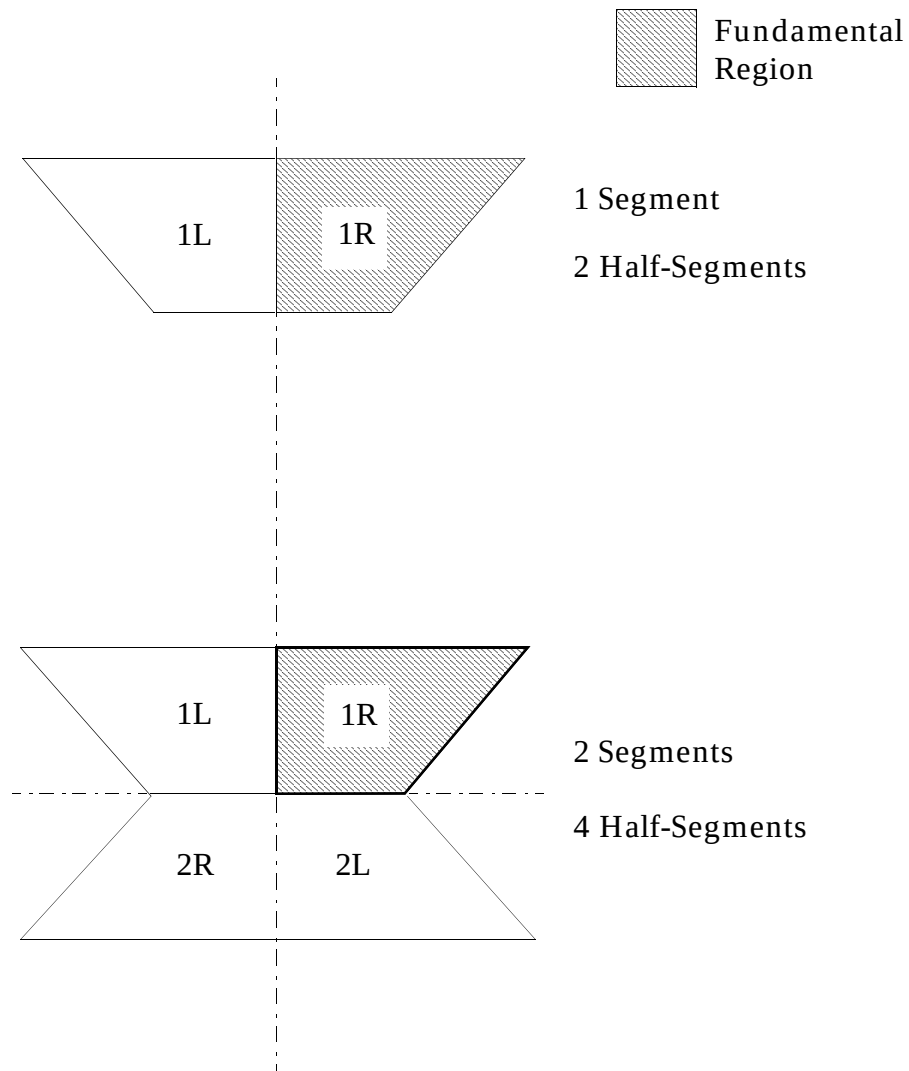


**Figure 16-2 Reflective Symmetry with One or Two Planes of Symmetry --
A Special Case of Dihedral Symmetry**

Thus, most of the types of symmetry shown in [Figure 16-1](#) are subclasses of cyclic symmetry. Even translational symmetry in one direction, see [Figure 16-1\(d\)](#), can be considered to be a limiting case of rotational symmetry in which the axis of symmetry is infinitely far away.

The relationship of the different classes of symmetry to the general class called cyclic symmetry is summarized in [Table 16-1](#). With few exceptions, all important types of symmetry are subclasses of cyclic symmetry, and they can, therefore, be analyzed with the cyclic symmetry capability in MSC.Nastran.

Table 16-1 Types of Symmetry

Type	Subclass of Cyclic Symmetry
Reflective Symmetry	
One Plane	Yes
Two Planes	Yes
Three Planes	No
Rotational Symmetry (Repetition about an axis)	Yes
Dihedral Symmetry (Reflective symmetry plus rotational symmetry)	Yes
Translational Symmetry	
One-D	Yes (special case)
Two-D	No
Three-D	No
Axisymmetry	
With Skew	Yes
Without Skew	Yes

The existence of geometric symmetry can be used to reduce the cost of finite element analysis in the following three ways:

1. By restricting the input data describing the finite element model to the fundamental region.
2. By restricting the computation of g -size stiffness and mass matrices to the fundamental region.
3. By exploiting the orthogonality of symmetrical component sets, such as the symmetrical and antisymmetrical sets in reflective symmetry, to reduce the cost of matrix operations.

If the loads on the structure have the same symmetry as the structure itself, the entire analysis reduces to a solution for the fundamental region with the appropriate boundary conditions. This case does not require elaborate procedures and can be implemented using the regular MSC.Nastran procedures. The only possible difficulty is in knowing the correct boundary conditions.

When the loads on the structure do not have the same symmetry as the structure itself, the more complex sequence of operations described earlier must be carried out, and it is here when automatic procedures become useful, saving you both time and computer costs.

The cyclic symmetry option in MSC.Nastran provides a high degree of automation and economy for each of its three subclasses: rotational symmetry (ROT), dihedral symmetry (DIH), and axisymmetry (AXI).

The cyclic symmetry option can be used in several solutions; however, only static analysis using Solution 114 is discussed in this user's guide.

16.3 Mathematical Description of Structural Symmetry

Further discussion of symmetry as applied to structural analysis requires the introduction and manipulation of mathematical symbols. The treatment in this section is limited to those mathematical aspects that are necessary for an understanding of the input and output for cyclic symmetry and its subclasses as applied in MSC.Nastran.

Reflective Symmetry

A structure is said to have reflective symmetry with respect to a yz plane if each of its relevant physical properties Q (such as mass density and elastic modulus) satisfies the equation

$$Q(x, y, z) = Q(-x, y, z) \quad \text{Eq. 16-1}$$

or in other words, if the physical properties are identical at the image points on the two sides of the yz plane. The plane of symmetry divides the structure into two halves that are arbitrarily called the right half and the left half. The right half is also called the fundamental region because, in analysis, it is the half for which a finite element model is prepared, whereas the finite element model for the left half is only implied.

The physical properties included in Q do not include the loads applied to the structure or its responses (displacements, internal forces, or stresses). Let the symbols u_r , and u_l represent the values of some particular response u at some point in the right half and at its image point in the left half of the structure. u_r and u_l do not necessarily satisfy the reflective symmetry property [Eq. 16-1](#), but their average u_s clearly does. Thus,

$$u_s = \frac{1}{2}(u_r + u_l) \quad \text{Eq. 16-2}$$

so that

$$u_s(x, y, z) = \frac{1}{2}(u(x, y, z) + u(-x, y, z)) \quad \text{Eq. 16-3}$$

and

$$u_s(-x, y, z) = \frac{1}{2}(u(-x, y, z) + u(x, y, z)) \quad \text{Eq. 16-4}$$

Since the right-hand sides of [Eq. 16-3](#) and [Eq. 16-4](#) are equal, it follows that

$$u_s(x, y, z) = u_s(-x, y, z) \quad \text{Eq. 16-5}$$

The quantity u_s is called the symmetrical part of the response. The antisymmetrical part of the response u_a is defined as one-half of the difference of u_r , and u_l . Thus,

$$u_a = \frac{1}{2}(u_r - u_l) \quad \text{Eq. 16-6}$$

The antisymmetrical part of the response has the property that

$$u_a(x, y, z) = -u_a(-x, y, z) \quad \text{Eq. 16-7}$$

Eq. 16-2 and **Eq. 16-6** show how to obtain the symmetrical and antisymmetrical parts of the response from the values of the response for the left and right halves of the structure. They may be written more compactly in matrix form

$$\begin{Bmatrix} u_s \\ u_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} u_r \\ u_l \end{Bmatrix} \quad \text{Eq. 16-8}$$

The vector

$$\begin{Bmatrix} u_s \\ u_a \end{Bmatrix}$$

is called the vector of symmetrical components, and the vector

$$\begin{Bmatrix} u_r \\ u_l \end{Bmatrix}$$

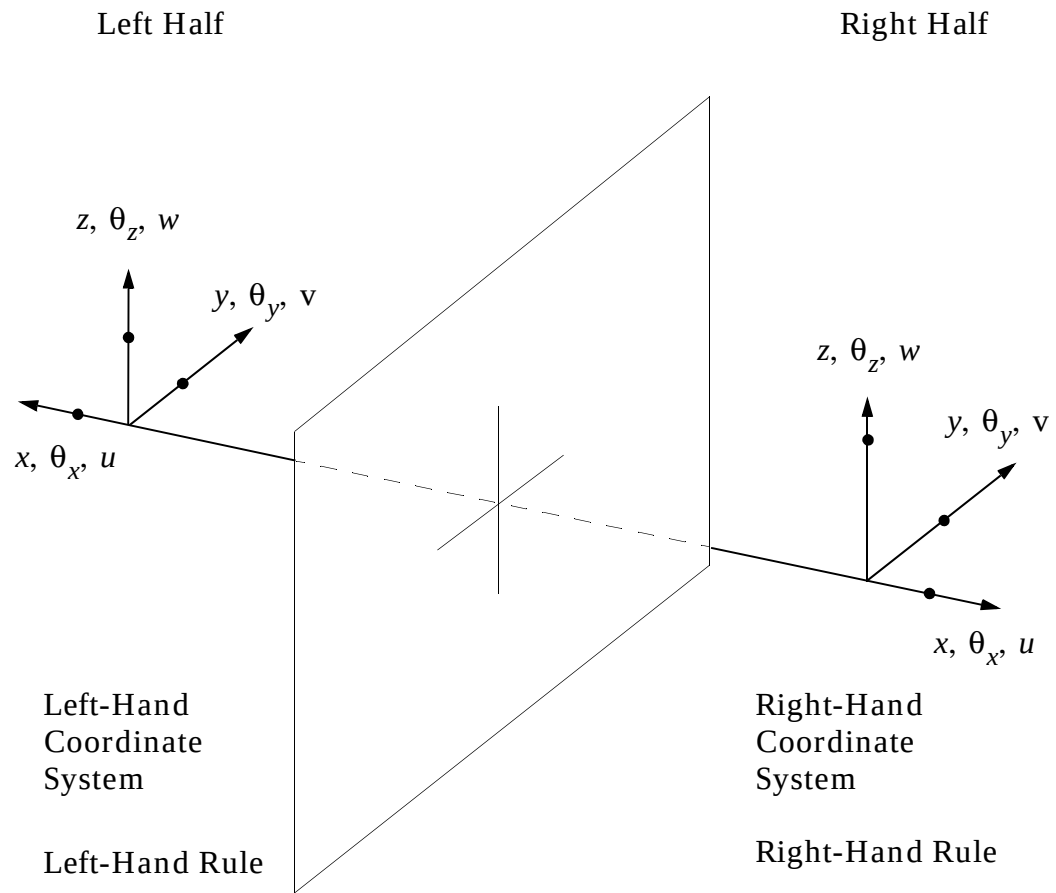
is called the vector of physical components. The physical components may be found from the symmetrical components by solving for them from **Eq. 16-8**.

$$\begin{Bmatrix} u_r \\ u_l \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} u_s \\ u_a \end{Bmatrix} \quad \text{Eq. 16-9}$$

Eq. 16-8 and **Eq. 16-9** apply to any response quantity, and they also apply to the applied loads. Thus, any particular applied load P has symmetrical components P_s and P_a evaluated as follows:

$$\begin{Bmatrix} P_s \\ P_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} P_r \\ P_l \end{Bmatrix} \quad \text{Eq. 16-10}$$

In order to include vectors as well as scalars in u and P , it is necessary to select the coordinate systems in the right and left halves to be mirror images of each other as shown in **Figure 16-3**. The coordinate system in the right half is right-handed, to conform with standard MSC.Nastran practice. Note, however, that the coordinate system in the implied left half is left-handed. This means, for example, that loads applied indirectly to the left half, by listing the images of their points of application in the right half, must be expressed in the particular left-hand coordinate system shown in **Figure 16-3**.



Symmetric Boundary Conditions: $u = 0$; $\theta_y = 0$; $\theta_z = 0$

Antisymmetric Boundary Conditions: $v = 0$; $w = 0$; $\theta_x = 0$

Figure 16-3 Reflective Symmetry

If the loads applied to a symmetrical structure are symmetrical, the response is symmetrical. The responses also include the displacements in the plane of symmetry, i.e., u , v , w , θ_x , θ_y , and θ_z at $x = 0$. Since the positive directions of u , θ_y , and θ_z are opposite in the two halves, continuity requires that they be zero at $x = 0$ in a symmetrical solution. Thus, $u = 0$, $\theta_y = 0$, and $\theta_z = 0$ are boundary conditions that can be applied with SPCs if only the right half is represented by a finite element model. The boundary conditions for v , w , and θ_x are natural, i.e., their reaction forces in the plane of symmetry are zero by symmetry, and they may be left unconstrained.

In like manner, if the loads applied to the structure are antisymmetrical, the response is antisymmetrical. By similar reasoning it can be shown that the antisymmetrical boundary conditions for a finite element model of the right half are $v = w = \theta_x = 0$.

If the applied loads are neither symmetrical nor antisymmetrical, the response may be obtained by the following procedure (which is valid only if the response is a linear function of the applied loads):

1. Compute the symmetrical and antisymmetrical load sets P_s and P_a by means of [Eq. 16-10](#).
2. Apply P_s to a finite element model of the right-hand side with symmetric boundary conditions. The response u_s is obtained.
3. Apply P_a to a finite element model of the right-hand side with antisymmetric boundary conditions. The response u_a is obtained.
4. Find the physical responses u_r and u_l by means of [Eq. 16-9](#).

Everything that has been said about one plane of symmetry can be extended to two or three orthogonal planes of symmetry. The steps that were outlined are performed automatically in MSC.Nastran's cyclic symmetry capability, using the identity between reflective symmetry with one or two planes of symmetry and dihedral symmetry with one or two segments, respectively. The steps may also be performed using basic MSC.Nastran procedures but with considerable nonautomated work (see the example in "[Modeling Guidelines](#)" in Chapter 9).

Axisymmetry

A structure is said to be axisymmetric if each of its relevant physical properties Q (such as mass density and elastic modulus) satisfies the equation

$$Q(r, \theta, z) = Q(r, 0, z) \quad \text{Eq. 16-11}$$

where r , θ , and z are the components of position in a cylindrical coordinate system. The cross section for $\theta = 0$ is the fundamental region whose properties are mapped into the entire structure by [Eq. 16-11](#).

Note: Axisymmetric without skew is the terminology introduced in "[Description of Geometric Symmetry in MSC.Nastran](#)" on page 561,

The response of the structure and its applied loads do not necessarily satisfy [Eq. 16-11](#). Let $u(r, \theta, z)$ be some particular response quantity (displacement, internal force, or load). If u is a component of a vector $\vec{u}$, let the coordinate system that expresses $\vec{u}$ be the cylindrical coordinate system used in connection with [Eq. 16-11](#). Then the Fourier series representation of $u(r, \theta, z)$ may be written as

$$u(r, \theta, z) = \bar{u}^0(r, z) + \sum_{k=1}^{\infty} (\bar{u}^{ks}(r, z) \sin(k\theta) + \bar{u}^{kc}(r, z) \cos(k\theta)) \quad \text{Eq. 16-12}$$

or, dropping the explicit dependence on r and z ,

$$u(\theta) = \bar{u}^0 + \sum_{k=1}^{\infty} (\bar{u}^{ks} \sin(k\theta) + \bar{u}^{kc} \cos(k\theta)) \quad \text{Eq. 16-13}$$

The harmonic response coefficients

$$\bar{u}^0, \bar{u}^{ks}, \bar{u}^{kc}$$

and may be regarded as symmetrical component sets that replace $u(\theta)$ in structural analysis. They are evaluated from the physical component $u(\theta)$ by the following Fourier integrals:

$$\begin{aligned}\bar{u}^0 &= \frac{1}{2\pi} \int_0^{2\pi} u(\theta) d\theta \\ \bar{u}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \sin(k\theta) d\theta \\ \bar{u}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \cos(k\theta) d\theta\end{aligned}\tag{Eq. 16-14}$$

Applied loads $P(\theta)$ can also be expanded in a Fourier series. The symmetrical component load sets are, by analogy with [Eq. 16-14](#),

$$\begin{aligned}\bar{P}^0 &= \frac{1}{2\pi} \int_0^{2\pi} P(\theta) d\theta \\ \bar{P}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \sin(k\theta) d\theta \\ \bar{P}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \cos(k\theta) d\theta\end{aligned}\tag{Eq. 16-15}$$

Fourier series are often used in the analysis of axisymmetric structures. The first step is to find the harmonic load coefficients from [Eq. 16-15](#). The next step is to find the harmonic response coefficients as functions of the harmonic load coefficients. It can be shown from a consideration of the properties of trigonometric functions that, in linear analysis,

$$\bar{u}^0$$

depends only on

$$\bar{P}^0$$

that the harmonic response coefficients for a particular harmonic

$$\bar{u}^{ks}$$

and

$$\bar{u}^{kc}$$

depend only on the harmonic load coefficients

$$\bar{P}^{ks} \text{ and } \bar{P}^{kc}$$

for the same harmonic index k . Computational effort can be further reduced by noting that higher harmonics where is problem dependent do not contribute significantly to the solution and need not be computed. Once the harmonic response coefficients are found, the physical components can be computed by means of [Eq. 16-12](#).

Rotational Symmetry

A structure is said to have rotational symmetry if each of its relevant physical properties Q repeats at even angular intervals around an axis A . Thus, referring to the structure shown in [Figure 16-4](#),

$$Q(\theta + \theta_n) = Q(\theta) \quad \text{Eq. 16-16}$$

where:

$$\theta_n = \frac{2\pi(n-1)}{N} \quad \text{and } n = 1, 2, 3, \dots, N.$$

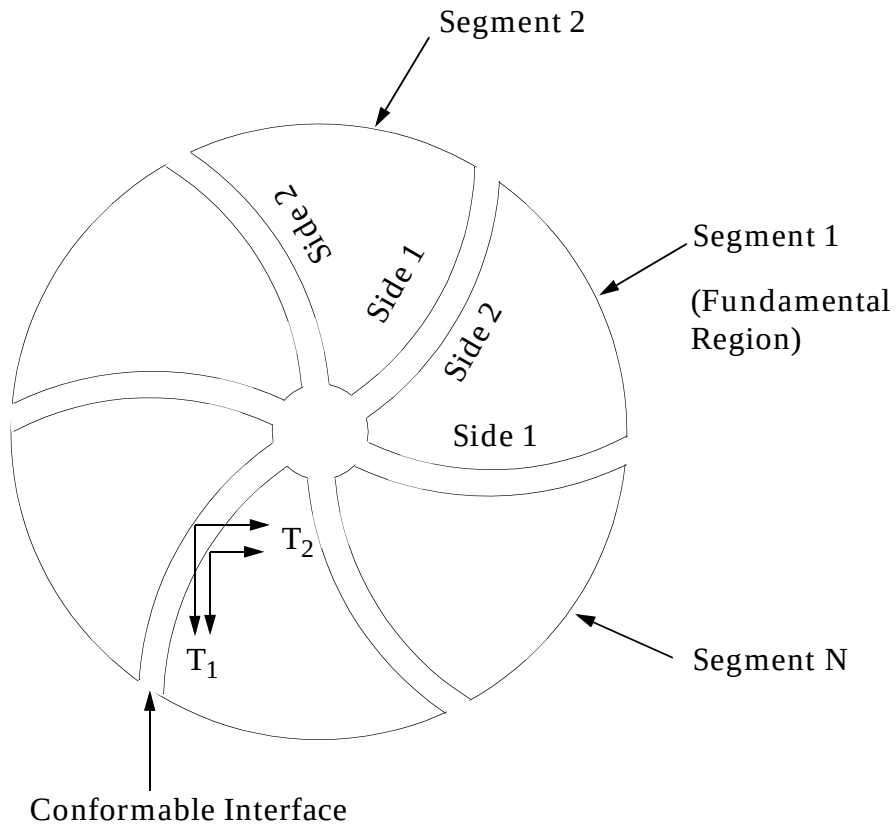


Figure 16-4 Conforming Boundaries for the ROT Option

You can apply rotational symmetry by using the following procedure:

1. Model one segment.
2. Make sure each segment has its own coordinate system that rotates with the segment.
3. Segment boundaries may be curved surfaces. The local displacement coordinate system must conform at the joining points. Specify a paired list of points on side 1 and side 2 that are to be joined.

For the purpose of finite element analysis, the structure is divided into N identical segments. The boundaries between segments may be arbitrarily curved surfaces. A finite element model is constructed for the first segment, which is called the fundamental region.

Let the response of the structure (i.e., a displacement, internal force, or stress) at some point p in the n -th segment be represented by the symbol $u^n(p)$. The response $u^n(p)$ can be expanded into a Fourier series in the same manner as was done for an axisymmetric structure except that the number of terms in the series is finite rather than infinite. The particular form of the Fourier series used in MSC.Nastran is

$$u^n(p) = \bar{u}^0(p) + \sum_{k=1}^{k_L} (\bar{u}^{ks}(p) \sin(k\theta_n) + \bar{u}^{kc}(p) \cos(k\theta_n)) + (-1)^{n-1} \bar{u}^{N/2}(p) \quad \text{Eq. 16-17}$$

The coefficients

$$\bar{u}^0(p), \bar{u}^{ks}(p), \text{ and } \bar{u}^{kc}(p)$$

are symmetrical component sets that may be used instead of $u^n(p)$, $n = 1, 2, \dots, N$ in a subsequent analysis. The last term is present only if the number of segments N is an even number. k_L is the largest integer less than $N/2$. The total number of symmetrical component sets is exactly equal to N , whether N is odd or even. Thus, considering $u^n(p)$ to be a physical degree of freedom in segment n , you can see that the total number of degrees of freedom in the symmetrical component sets is exactly equal to the number of physical degrees of freedom in the structure. As a result of this fact, it can be shown that the analysis of the finite element model using the symmetrical components is exact, even though there are only a finite number of terms in the Fourier series.

If the number of segments is large, the higher harmonics may contribute very little to the total solution. In MSC.Nastran you have the option to select the harmonics (values of k) to be used in the analysis.

The symmetrical components are evaluated from the physical components by means of the following summations that replace the Fourier integrals used in axisymmetric analysis. The argument (p) is dropped for convenience.

$$\begin{aligned}
\bar{u}^0 &= \frac{1}{N} \sum_{n=1}^N u^n \\
\bar{u}^{ks} &= \frac{2}{N} \sum_{n=1}^N u^n \sin(k\theta_n) & k = 1, 2, \dots, k_L \\
\bar{u}^{kc} &= \frac{2}{N} \sum_{n=1}^N u^n \cos(k\theta_n) & k = 1, 2, \dots, k_L \\
\bar{u}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} u^n
\end{aligned}
\tag{Eq. 16-18}$$

You should note that stiffness damping, and mass matrices become uncoupled for $\bar{u}^k$.

Eq. 16-18 also applies to the expansion of applied loads into symmetrical component sets. Thus

$$\begin{aligned}
\bar{P}^0 &= \frac{1}{N} \sum_{n=1}^N P^n \\
\bar{P}^{ks} &= \frac{2}{N} \sum_{n=1}^N P^n \sin(k\theta_n) & k = 1, 2, \dots, k_L \\
\bar{P}^{kc} &= \frac{2}{N} \sum_{n=1}^N P^n \cos(k\theta_n) & k = 1, 2, \dots, k_L \\
\bar{P}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} P^n
\end{aligned}
\tag{Eq. 16-19}$$

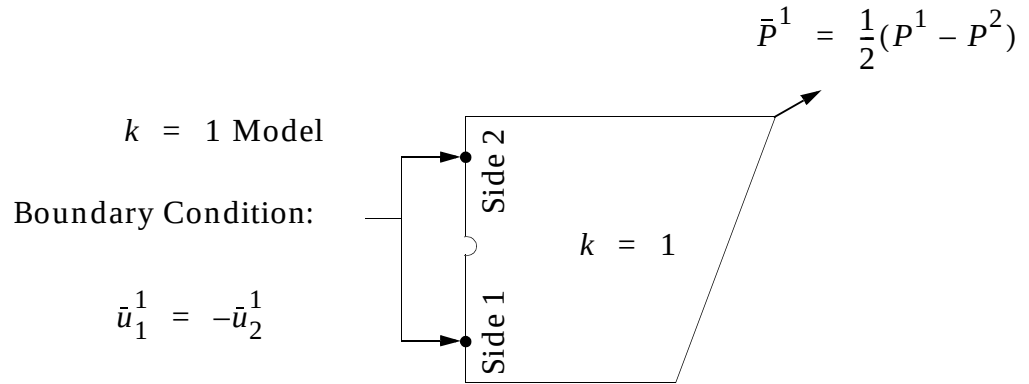
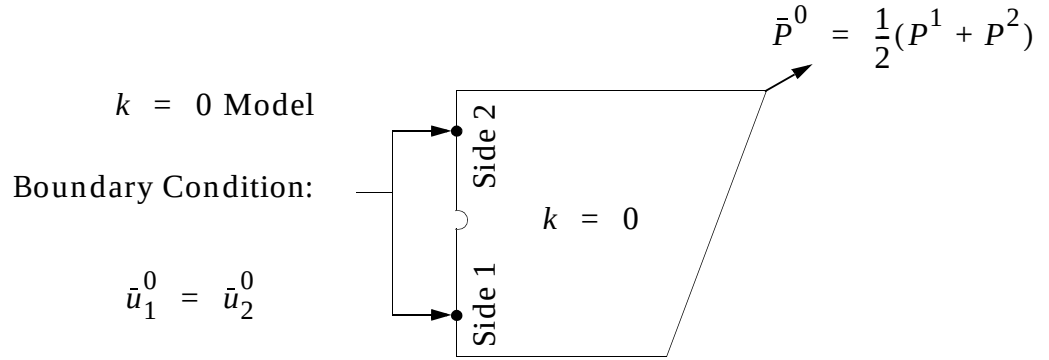
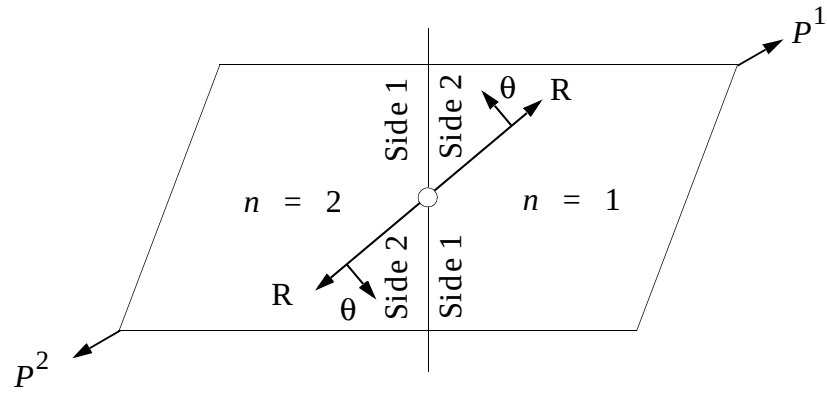
The harmonic index k plays an important role in the analysis of structures with rotational symmetry or in any other form of cyclic symmetry. The harmonic index k has the values $k = 1, 2, \dots, N/2$. It can be shown that, if the only nonzero symmetrical load components are for a particular value of k , then the only nonzero symmetrical response components are for the same value of k , provided only that the structure is linear and satisfies **Eq. 16-16**. This fact leads directly to an efficient method of analysis in which each symmetric load component is computed from **Eq. 16-19** and applied to a finite element model that is very much smaller than the complete structure.

The implied coordinate systems in the physical segments remain fixed in the segments and are rotated with them. In order to satisfy continuity between segments, the grid point on side 2 of segment 1 must coincide with the grid point on side 1 of segment 2 and their coordinate systems must be aligned (see **Figure 16-4**). As a result, you must take care to align the grid points on sides 1 and 2 of the fundamental region so that their positions and

displacement coordinate systems differ only by a rotation through the angle $a = 2\pi/N$ about the axis of symmetry. This is most easily satisfied by using a cylindrical coordinate system for the grid points on sides 1 and 2. If this is done, the boundary conditions are satisfied by imposing the following constraints on the symmetrical components of displacement at corresponding points on sides 1 and 2 (indicated by subscripts).

$$\begin{aligned}\bar{u}_1^0 &= \bar{u}_2^0 \\ \bar{u}_1^{kc} \cos(ka) + \bar{u}_1^{ks} \sin(ka) &= \bar{u}_2^{kc} \\ -\bar{u}_1^{kc} \sin(ka) + \bar{u}_1^{ks} \cos(ka) &= \bar{u}_2^{kc} \\ k &= 1, 2, \dots, k_L \\ -\bar{u}_1^{N/2} &= \bar{u}_2^{N/2}\end{aligned}\tag{Eq. 16-20}$$

As an example of the application, consider the simple problem illustrated in **Figure 16-5**. Note that the skew slab is divided into two identical segments and that the displacement coordinate system for the second segment is rotated 180° from the first. A cylindrical coordinate system is used (at least along sides 1 and 2) to ensure the compatibility of displacement components at the points in the boundaries between segments.



Combination of Solutions:

$$u^1 = \bar{u}^0 + \bar{u}^1 \quad u^2 = \bar{u}^0 - \bar{u}^1$$

Figure 16-5 Example of Rotational Symmetry

Since $N = 2$, there are two symmetrical component sets $\bar{u}^0$ and $\bar{u}^{N/2} = \bar{u}^1$. The symmetrical load components, as computed from [Eq. 16-19](#), are

$$\bar{P}^0 = \frac{1}{2}(P^1 + P^2)$$

$$\bar{P}^1 = \frac{1}{2}(P^1 - P^2)$$

Eq. 16-21

$\bar{P}^0$ and $\bar{P}^1$ are applied to separate copies of the finite element model for the fundamental region labeled the $K = 0$ model and the $K = 1$ model. The boundary condition linking all six degrees of freedom at corresponding pairs of grid points on sides 1 and 2, as computed from [Eq. 16-20](#), are

$$\begin{aligned}\bar{u}_1^0 &= \bar{u}_2^0 \\ \bar{u}_1^1 &= -\bar{u}_2^1\end{aligned}\tag{Eq. 16-22}$$

These boundary conditions can be supplied in Solution 101 with multipoint constraints; however, they are provided automatically in MSC.Nastran's cyclic symmetry capability.

The $K = 0$ and $K = 1$ models are solved in the usual way to obtain the symmetrical response components

$$\bar{u}^0 \text{ and } \bar{u}^1$$

As a final step, the physical responses are evaluated as follows from [Eq. 16-17](#)

$$\begin{aligned}u^1 &= \bar{u}^0 + \bar{u}^1 \\ u^2 &= \bar{u}^0 - \bar{u}^1\end{aligned}\tag{Eq. 16-23}$$

The static condensation procedure described in “[Matrix Operations](#)” on page 455 can be used to advantage in the sample problem that was just described. With this procedure all of the interior degrees of freedom are placed in the u_o set and eliminated, resulting in a greatly reduced stiffness matrix for the remaining degrees of freedom on the boundary (sides 1 and 2). The advantage of this procedure is that, since it needs to be done only once, there is a net saving of computer time.

Dihedral Symmetry

Dihedral symmetry is a special case of rotational symmetry in which each segment has an internal plane of reflective symmetry as illustrated in [Figure 16-6](#). The fundamental region, for which a finite element model is prepared, is the right half of the first segment. The plane that forms the boundary between the right and left halves of segment 1 is labeled side 2. The other boundary of the fundamental region, which must also be planar in dihedral symmetry, is labeled side 1. You must give separate lists for the degrees of freedom on sides 1 and 2. The displacement coordinate systems on the boundaries must be aligned parallel and perpendicular to the boundaries.

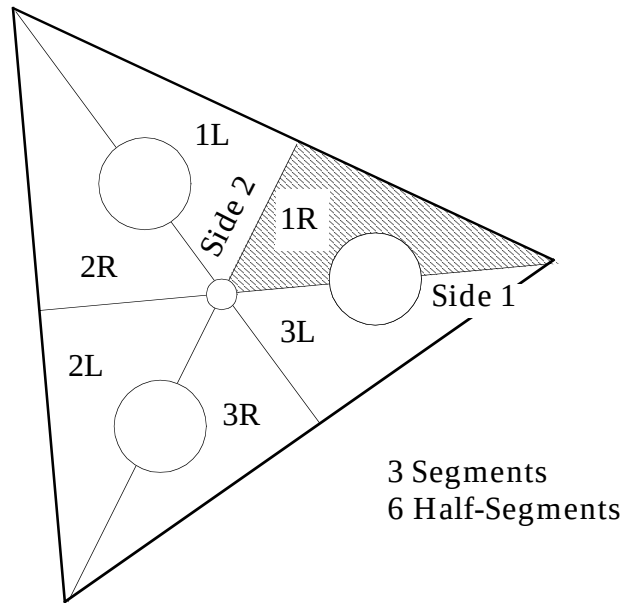


Figure 16-6 Dihedral Symmetry

The typical procedure that you follow when using dihedral symmetry is as follows:

1. You model one half-segment (1R).
2. At a boundary between segments or half-segments, the displacement components must be parallel and perpendicular to the boundary. The boundaries are planes passing through the axis of symmetry.
3. You provide separate lists of the points on side 1 and side 2.

The implied coordinate system for segment 1L is the mirror image of that for segment 1R, the fundamental region. This fact has important implications for the input of loads data and the interpretation of output. The coordinate systems for other half-segments in [Figure 16-6](#) are either rotated, or rotated and reflected as indicated in [Table 16-2](#).

Table 16-2 Coordinate Systems for Half Segments

Half-Segment	Type of Coordinate System	Rotation About the Axis of Symmetry
1R	Right-handed	0°
1L	Left-handed	0°
2R	Right-handed	120°
2L	Left-handed	120°
3R	Right-handed	240°
3L	Left-handed	240°

As in rotational symmetry, the response quantities at corresponding points in the physical segments can be expressed as a truncated Fourier series with the polar angle of the n -th segment $\theta_n = 2\pi(n-1)/N$ as the argument where N is the number of segments. The series for response quantities in the right half-segments is

$$u^{nR} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} + \bar{u}^{kc*}) \cos(k\theta_n) + (\bar{u}^{ks} + \bar{u}^{ks*}) \sin(k\theta_n)] \quad \text{Eq. 16-24}$$

where k_L is the largest integer less than or equal to $N/2$ (depending on whether n is odd or even) and

$$\bar{u}^{kc}, \bar{u}^{kc*}, \bar{u}^{ks}, \text{ and } \bar{u}^{ks*}$$

are symmetrical component sets. The series for response quantities in the left half-segments is

$$u^{nL} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} - \bar{u}^{kc*}) \cos(k(\theta_n + a)) - (\bar{u}^{ks} - \bar{u}^{ks*}) \sin(k(\theta_n + a))] \quad \text{Eq. 16-25}$$

where, in addition to previously defined terms, $a = 2\pi/N$.

The symmetrical component sets are evaluated from the physical components as follows:

$$\begin{Bmatrix} \bar{u}^{kc} \\ \bar{u}^{kc*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \cos(k\theta_n) \pm u^{nL} \cos(k(\theta_n + a))] \quad \text{Eq. 16-26}$$

$$\begin{Bmatrix} \bar{u}^{ks} \\ \bar{u}^{ks*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \sin(k\theta_n) \mp u^{nL} \sin(k(\theta_n + a))] \quad \text{Eq. 16-27}$$

where, in addition to previously defined terms, $\delta = 1/2$ for $k = 0$ and $N/2$, and $\delta = 1$ for all other values of k .

The number of symmetrical components sets is equal to the number of half-segments in the structure.

Note: Neglecting the identically null sets $\bar{u}^{ks}$ and $\bar{u}^{ks*}$ for $k = 0, N/2$.

As in the case of rotational symmetry, the symmetrical load components for a particular value of k produce only the symmetrical response components for the same value of k , or, to put it another way, the analyses for the different values of k are uncoupled.

Furthermore, the analyses for the starred (*) and unstarred symmetrical components are uncoupled. For $K = 0$ and $k = N/2$, the complete finite element model for the unstarred terms consists of a single copy of the fundamental region (1 R). For $0 < k < N/2$, the complete finite element model consists of two copies of the fundamental region with coupling between their boundaries. The boundary conditions are different for each value

of k . The finite element models for the starred (*) terms are identical to those for the unstarred terms for $k > 0$. Only the symmetrical load components, which are computed by [Eq. 16-26](#) and [Eq. 16-27](#), are different.

As an example, consider the case of a structure with two planes of reflective symmetry as shown in [Figure 16-2](#). Applying dihedral symmetry to this case, you can see that

- The number of segments $N = 2$.
- The segment angles $\alpha = n$.
- $\theta_1 = 0$.
- $\theta_2 = \pi$.
- The harmonic indices $k = 0, 1$.

Substitution of these values into [Eq. 16-26](#) and [Eq. 16-27](#) gives

$$\begin{aligned}
 \bar{u}^{0c} &= \frac{1}{4}(u^{1R} + u^{1L} + u^{2R} + u^{2L}) \\
 \bar{u}^{1c} &= \frac{1}{4}(u^{1R} - u^{1L} - u^{2R} + u^{2L}) \\
 \bar{u}^{0c*} &= \frac{1}{4}(u^{1R} - u^{1L} + u^{2R} - u^{2L}) \\
 \bar{u}^{1c*} &= \frac{1}{4}(u^{1R} + u^{1L} - u^{2R} - u^{2L}) \\
 \bar{u}^{0s} &= \bar{u}^{0s*} = \bar{u}^{1s} = \bar{u}^{1s*} = 0
 \end{aligned}
 \tag{Eq. 16-28}$$

Since all of the signs are positive in the equation for $\bar{u}^{0c}$, it follows that $\bar{u}^{0c}$ is symmetrical with respect to both side 1 and side 2 in [Figure 16-2](#). The other nonzero symmetrical components are antisymmetrical with respect to one or both of the sides as shown in [Table 16-3](#).

Table 16-3 Symmetrical Components for Reflective Symmetry Example

Symmetrical Component	Symmetry	
	Side 1	Side 2
$\bar{u}^{0c}$	S	S
$\bar{u}^{1c}$	S	A
$\bar{u}^{0c*}$	A	A
$\bar{u}^{1c*}$	A	S

It can be seen that regardless of the number of segments, the starred terms are always antisymmetrical, and the unstarred terms are always symmetrical with respect to side 1 of the fundamental region. Advantage may be taken of this fact to reduce the amount of calculation when it can be determined beforehand that the applied loading has a plane of symmetry or antisymmetry that coincides with side 1 of the fundamental region, or, in other words, if the problem has an overall plane of symmetry. The amount of computation can be reduced still further if the problem has a second overall plane of symmetry perpendicular to the first. The second overall plane of symmetry can exist only if there is an even number of segments.

16.4 Bulk Data Entries for Cyclic Symmetry

The only Bulk Data entries that are unique to cyclic symmetry in MSC.Nastran are listed in [Table 16-4](#).

Table 16-4 Bulk Data Entries Unique to Cyclic Symmetry

Bulk Data Entry	Function
CYJOIN	Defines the grid points on sides 1 and 2 of the fundamental region.
CYSYM	Specifies the number of segments in the entire model and the type of symmetry. Allowable symmetry types are ROTational, DIHedral, and AXIsymmetric.
CYAX	List of grid points that lie on the axis of symmetry.
CYSUP	List of supported components at a single grid point that are “constrained” in order to prevent rigid body motion for free bodies.
LOADCYN	Defines applied loads in terms of physical components.
LOADCYH	Defines applied loads in terms of their harmonic components.
LOADCYT	Specifies loads as a function of azimuth angle for the AXI symmetry type.

Each of the entries shown in [Table 16-4](#) is discussed below in terms of its general characteristics. Also, examples of the use of these entries are provided.

The boundary points of a segment in cyclic symmetry problems using the CYJOIN entry are as follows:

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

Field	Contents
SIDE	Side identification.
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems.
Gi	Grid or scalar point identification numbers.

One CYJOIN entry plus all necessary continuation entries are used to specify the GRID and/or scalar points (SPOINT), on side 1 of the fundamental region. A second CYJOIN entry plus all necessary continuation entries are used to specify the GRID and/or scalar points (SPOINT) on side 2 of the fundamental region. As will be shown, points that lie on the axis of symmetry appear on a CYAX Bulk Data entry but not on a CYJOIN entry.

1. For the ROT option, the two CYJOIN lists must be of the same length and, since the boundary conditions for the fundamental region involve these two lists, it is required that the two lists be ordered. That is, the n -th point on side 1 is related through boundary conditions to the n -th point of side 2. Side 2 is related to side 1 as indicated in [Figure 16-4](#).

Note that displacement compatibility must be maintained at the boundaries. This requirement necessitates that the displacement coordinate systems for the boundary points be either cylindrical or spherical.

2. The AXI option is a subset of the ROT option in which all the grid points in the fundamental region lie either on side 1 or on side 2. The rules for the CYJOIN entry are identical for both the ROT and AXI options of cyclic symmetry.
3. For the DIH option, side 1 denotes the boundary between segments, and side 2 denotes the side that is contained in the plane of reflective symmetry of the segment. Since the boundary conditions do not relate sides 1 and 2, it is not necessary that the two lists be of equal length. It is, however, necessary under the DIH option to provide an entry in the third field for the parameter C. The appropriate entries are listed in [Table 16-5](#).

It may be deduced that, unlike the ROT option (in which case, one and only one logical CYJOIN entry must exist for each of side 1 and side 2), any number of CYJOIN entries may be used to define the points on a side of the fundamental region in the DIH option. Thus, each point along a boundary may appear on a separate CYJOIN entry if, for example, each point has a different coordinate system. In any event, for the DIH option, the displacement coordinate systems associated with boundary points must be oriented so that the components of motion are normal and parallel to the boundary planes.

Table 16-5 Parameter C for the DIH Option

Type of Point	C	Description
Grid	T1	T1 component of translation motion normal to boundary.
	T2, R, or C	T2 component of translation motion normal to boundary.
	T3 or S	T3 component of translation motion normal to boundary.
	T1	If the user-defined translation is normal to the boundary or if the user-defined rotation vector is in the plane containing the boundary.
	Blank, T2, or T3	If the user-defined translation is parallel to the boundary or if the user-defined rotation vector is normal to the boundary.

Specifies the number of segments NSEG that comprise the total structure and the cyclic symmetry option STYPE to be used in the analysis. The format of the CYSYM entry is as follows:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

Field	Contents

NSEG	Number of segments.

STYPE	Symmetry type (“ROT”, “AXI”, “DIH”).

The polar angle θ subtended by the element model for the fundamental region, must be such that $\theta = 360^\circ / \text{NSEG}$ exactly for the ROT and AXI options. For the DIH option, θ must equal $180^\circ / \text{NSEG}$ exactly if there are any points on side 1. The polar angle θ is smaller in the DIH option because the fundamental region is the right half of segment 1 in this case. If there are no points on side 1, θ must be less than or equal to $180^\circ / \text{NSEG}$ in the DIH option because otherwise the segments would overlap. The number of segments for the AXI option should be large enough (typically 360 segments) to approximate true axisymmetry.

The CYAX Bulk Data entry provides you with the option to have points on the axis of symmetry.

The format of the CYAX entry is as follows:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

Field	Contents

Gi	A list of grid points on the axis of symmetry.

Special rules, which are enumerated below, are inherently necessary for the existence of points on the line of axisymmetry.

1. The coordinate system that locates points on the axis of symmetry and the displacement coordinate system for these points must be rectangular with the z-component aligned with the axis of symmetry. In addition, if the DIH option is selected, the y-axis must be perpendicular to side 1 of the fundamental region.
2. Grid points on the axis of symmetry may not appear on a CYJOIN Bulk Data entry.
3. Grid points on the axis of symmetry may not be referenced as dependent points by MPCs and/or rigid elements.
4. Grid points on the axis of symmetry may have SPCs applied in accordance with the following rule that is necessary to satisfy symmetry. If $\text{NSEG} > 3$, SPCs must be applied to both components 1 and 2 or to neither. The same rule applies to components 4 and 5.

5. Grid points listed on CYAX Bulk Data entries must be in the a -set.

The CYSUP Bulk Data entry provides for free body supports in cyclic symmetry analysis.

The format of the CYSUP entry is as follows:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

Field	Contents
GID	Grid point identification number.
C	Component numbers.

The total structure can have, at most, six rigid body degrees of freedom. Supports that remove only the rigid body motions are applied to the $k = 0$ and $k = 1$ harmonics of the components of motion listed on this entry. These components may not be constrained by SPCs, may not be referenced as dependent degrees of freedom by MPCs and / or rigid elements, and may not be referenced on OMIT or OMIT1 Bulk Data entries (see Reference 6.).

Rigid body motion is possible only for harmonics $k = 0$ and $k = 1$. Translation along and rotation about the axis of symmetry are the only possible rigid body motions for $k = 0$, while translations normal to the axis of symmetry and rotations about these translational directions are the only possible rigid body motions for $k = 1$. Special rules relative to the proper “support” of rigid body motion are listed on the CYSUP entry.

LOADCYN, LOADCYH, and LOADCYT Bulk Data Entries

The LOADCYN, LOADCYH, and LOADCYT Bulk Data entries are selected through the LOAD commands in the Case Control Section. The rules to be described are illustrated in Figure 16-7.

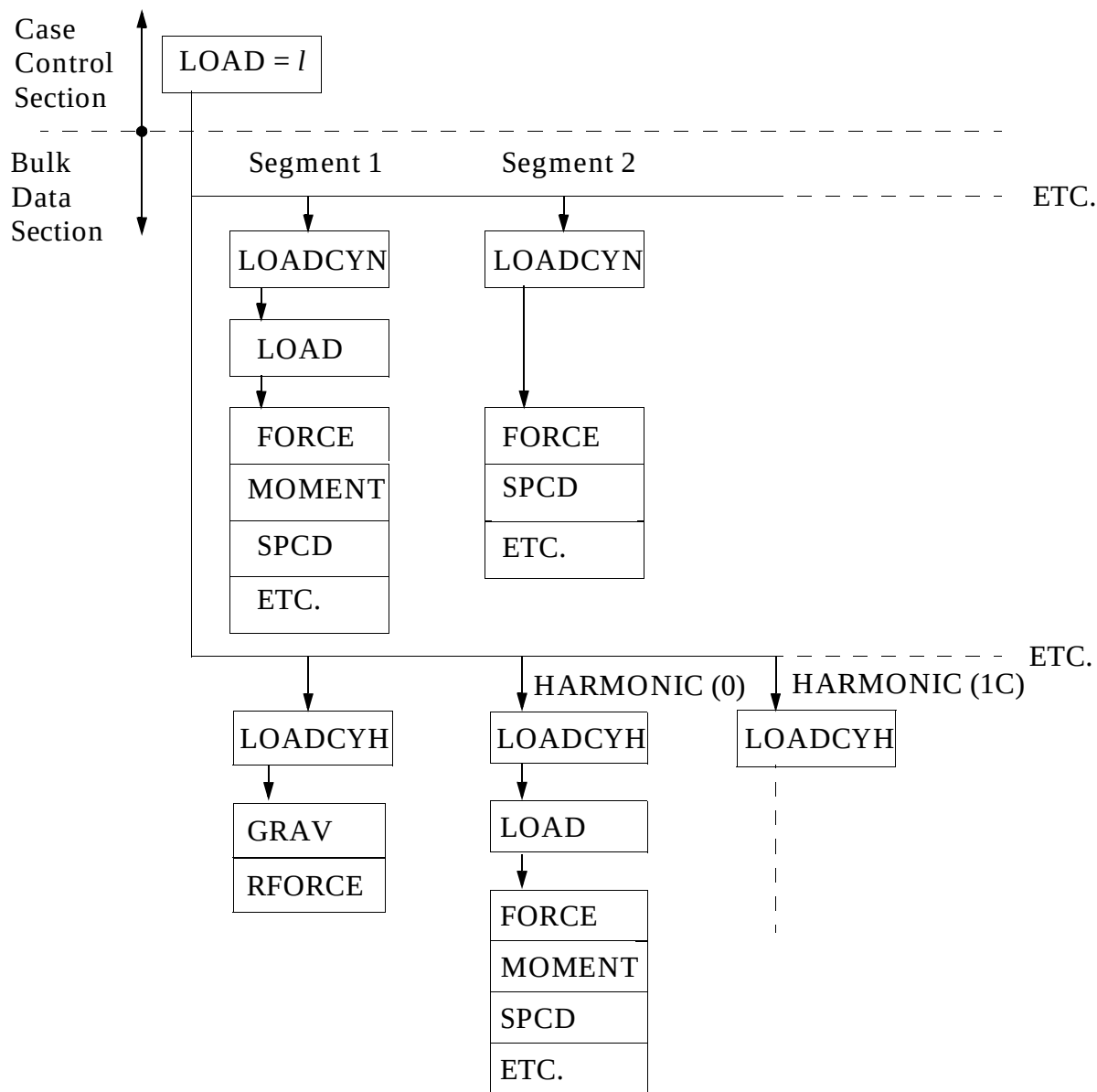


Figure 16-7 Cyclic Symmetry Flowchart

The LOADCYN entry provides for the identification of the segments on which referenced loads, defined in terms of physical components, are to be applied.

The format of the LOADCYN entry is as follows:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYPE	S1	L1	S2	L2	

Field	Contents
SID	Load set identification number.
S	Scale factor.
SEGID	Segment identification number.
SEGTYPE	Segment type.

Si Scale factors.

Li Load set ID numbers.

The loads are referenced by listing the identification numbers of one or two LOAD Bulk Data entries. Scale factors on the LOADCYN Bulk Data entry are used to form a linear combination of the load sets referenced on the LOAD Bulk Data entries. Alternatively, the LOADCYN Bulk Data entries may directly reference load sets comprised of the types of static load definition entries shown in [Table 16-6](#).

Table 16-6 Static Load Definitions for Cyclic Symmetry

Point Loads	Distributed Loads	Temperature Loads	Enforced Displacement
FORCE	PLOAD1	TEMP	SPCD
FORCE1	PLOAD2	TEMPP1	
FORCE2	PLOAD4	TEMPRB	
MOMENT		TEMD	
MOMENT1			
MOMENT2			
SLOAD			

The loads listed on these entries are applied at the images in the segment referenced in field 4 of the LOADCYN entry of the grid points and elements listed on these entries. The coordinate systems for image points are rotated and /or reflected from the coordinate system in the fundamental region.

Note that GRAV and RFORCE Bulk Data entries are not included in the above list. These static load entries are handled automatically in a very straightforward fashion through the LOADCYH Bulk Data entry.

The LOADCYH entry provides for the identification of referenced loads that are defined in terms of harmonic components. The format of the LOADCYH entry is as follows:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

Field

Contents

SID	Load set identification number.
S	Scale factor.
HID	Harmonic component.
HTYPE	Harmonic type.

Si Scale factor on Li.

Li Load set identification number.

This option is particularly useful for the definition of gravity (GRAV) and centrifugal loads (RFORCE) because you only need to specify GRAV or RFORCE under the HTYPE field on the entry and the correct harmonic components of the specified load are automatically generated. The component of gravity parallel to the axis of symmetry produces harmonic loads of the order $k = 0$, and components of gravity perpendicular to the axis of symmetry produce loads of the order $k = 1$. Components of centrifugal force produce harmonic loads of the order $k = 0$ if the spin axis coincides with the axis of symmetry, and of the order $k = 0$, $k = 1$, and $k = 2$ if it does not.

The LOADCYT Bulk Data entry is used with the AXI option to specify loads in terms of physical components as a tabular function of the azimuth angle (measured in degrees). The format of the LOADCYT entry is as follows:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TABLEID1	LOADSET1	METHOD1	TABLEID2	LOADSET2	METHOD2		

Field	Contents
SID	Load set identification number.
TABLEIDi	Table ID for table load input for load set Li.
LOADSETi	Load set Li.
METHODi	Method of interpolation. 0 interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. 1 interpolate the magnitude of the load at corresponding grid points in all segments.

This capability is conveniently implemented by referencing a set of load definition entries under the heading LOADSET on the LOADCYT entry.

These entries define a magnitude for the load. The azimuthal variation of the applied load is specified on the TABLEDi entry referenced under the heading TABLEID. The net applied load is the product of the magnitude given on the load definition entry times the appropriate value on the TABLEDi entry. The heading METHODDi on the LOADCYT entry is utilized to specify the type of interpolation to be used in assigning loads to degrees of freedom.

16.5 Case Control Section for Cyclic Symmetry

The general structure of the Case Control Section is discussed in this section.

The only Case Control commands that are unique to cyclic symmetry are listed in [Table 16-7](#).

Table 16-7 Case Control Commands Unique to Cyclic Symmetry

Command	Function
HARMONICS	Specifies harmonics for which the computation is desired.
DSYM	Provides for either one or two planes of overall symmetry under the DIH option.
NOUTPUT	Controls requests for the output of physical components.
HOUTPUT	Controls requests for the output of harmonic components.

The specific features of the Case Control Section are summarized as follows:

1. References to SPC and /or MPC sets in the Case Control Section must appear above the subcase level, i.e., before the first subcase.
2. A HARMONICS command must appear above the subcase level (i.e., before the commands that describe the first subcase) to specify the harmonics for which solutions are to be computed. The HARMONIC command cannot reference a set definition that specifies “ALL”. If you need to use all of the harmonics, specify:

HARMONICS = ALL
3. One subcase is required per loading condition. Loads, enforced deformations, and enforced displacements are requested in the standard manner, i.e., through LOAD, TEMP(LOAD), DEFORM, and SPC set selections. As described above, loads can be input in terms of physical components and /or harmonic components. Under the AXI option, loads may also be specified in terms of physical components that are tabulated as a function of the azimuth angle (in degrees).
4. When the DIH option of cyclic symmetry is selected, advantage may be taken of overall planes of symmetry when the applied loads are symmetrically and /or antisymmetrically disposed with respect to these planes. In these cases, loads need only be specified for segments in the first half plane when there is one plane of overall symmetry and in the first quadrant when there are two planes of overall symmetry. The existence of one or two planes of overall symmetry is specified on the DSYM Case Control command. One DSYM command is permitted per subcase.

5. Requests for printed output for either or both of the harmonic components and the physical components of all types of standard output quantities for static analysis are available through the HOUTPUT and NOUTPUT Case Control commands, respectively. These commands provide the capability to specify the segments for which output is desired and, if harmonic output is requested, the cyclic components for which output is desired.
 - If a subcase contains TEMP(Load) and / or DEFORM commands, irrespective of the presence of a LOAD command, element force and stress output are only correct in terms of physical components if the TEMP(Load) and the DEFORM loads are defined in terms of physical components. Similarly, if the TEMP(Load) and DEFORM data are provided in terms of harmonic components, element force and stress output are only correct in terms of harmonic components.
 - If, under the DIH option of cyclic symmetry, a DSYM command is explicitly present under a subcase, the output is, by default, limited to the segments in the first half plane if there is one plane of overall symmetry, and to the segments in the first quadrant if there are two planes of overall symmetry. This default applies irrespective of the segments listed on NOUTPUT commands.
 - The specification of applied loads and the interpretation of the output data requires an awareness of the orientation of the coordinate system associated with the individual segments. For the ROT and AXI options, the coordinate system used to define the model of the fundamental region is rotated through the symmetry operation that “generates” each of the other identical segments. For the DIH option, the coordinate system used to define the model of the fundamental region (segment 1 R) is rotated through the necessary symmetry operations that generate the R-half of each of the identical segments. The coordinate system for the L-half of a segment is the mirror image of the coordinate system associated with the R-half of the segment in question; thus, the L-half of any segment has a left-handed coordinate system.
6. Undeformed and deformed plots of the fundamental region may be requested.
7. Cyclic Symmetry does not support the use of SUBCOM / SUBSEQ commands. If you need to combine loads, use the LOAD Bulk Data entry.

To illustrate some of the rules tabulated above, typical Case Control Sections for the cylinder (**Figure 16-8**) and square plate (**Figure 16-9**) problems are shown in **Listing 16-2** and **Listing 16-3**, respectively.

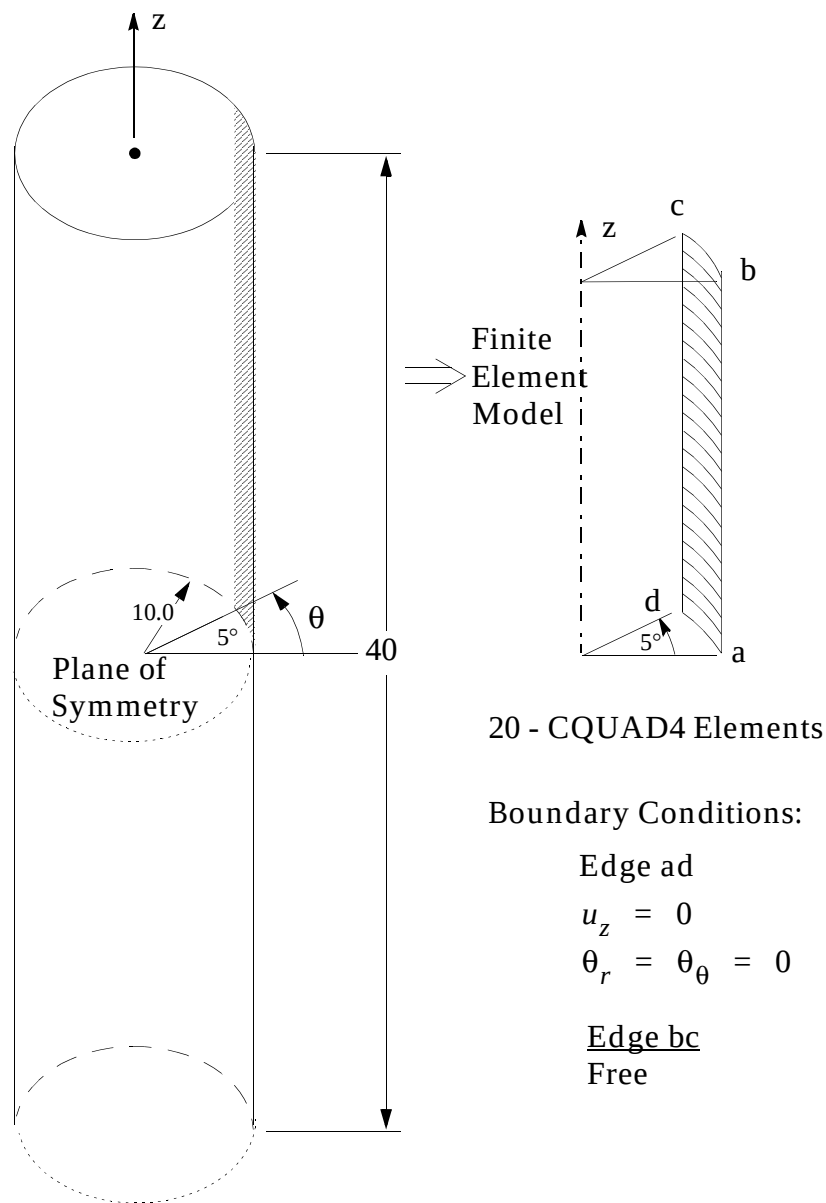


Figure 16-8 Cylinder Example

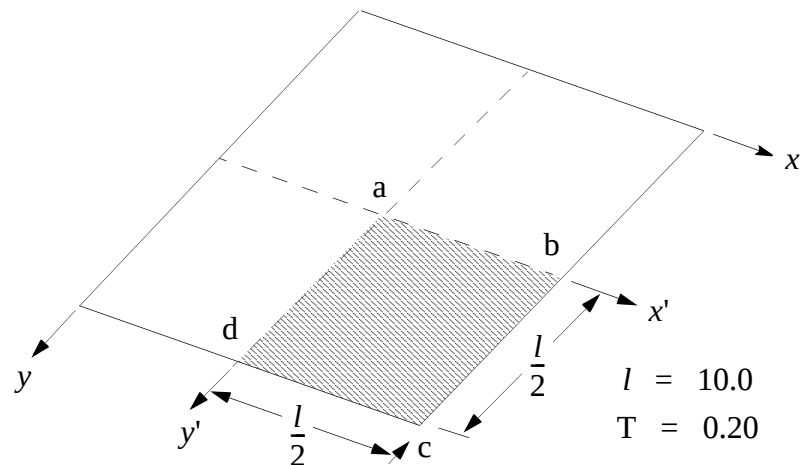


Figure 16-9 Square Plate Example

Listing 16-1 ROT or AXI Symmetry Type

```
TITLE=CYCLIC SYMMETRY-STATICS
SUBTITLE=FIVE DEGREE SEGMENT OF CYLINDER
LABEL=AXISYMMETRIC LOADING
SPC=1
SET 1=0
SET 2=1
HARMONICS=1
NOUTPUT=2
DISP=ALL
STRESS=ALL
FORCE=ALL
SUBCASE 1
  LABEL=BAND LOADING
  LOAD=1
SUBCASE 2
  LABEL=CENTRIFUGAL LOADING
  LOAD=2
BEGIN BULK
```

Listing 16-2 DIH Symmetry Type

```
TITLE=CYCLIC SYMMETRY-STATICS
SUBTITLE=SIMPLY-SUPPORTED SQUARE PLATE
SPC=1
HARMONICS=ALL
DISPLACEMENT=ALL
NOUTPUT=ALL
DSYM=SS
SUBCASE 1
  LABEL=LOAD AT CENTER OF PLATE
  LOAD=1
SUBCASE 2
  LABEL=UNIFORM PRESSURE LOAD
  LOAD=2
SUBCASE 3
  LABEL=TEMPERATURE LOAD
  TEMP(LOAD)=3
SUBCASE 4
  LABEL=INPLANE GRAVITY LOAD
  LOAD=4
  DSYM=SA
BEGIN BULK
```

The following comments are applicable to the above Case Control Section.

1. In **Listing 16-1**, the HARMONICS command specifies that solutions are to be computed only for the zero harmonic. If the applied loads are axisymmetric, no higher harmonics need to be considered. For **Listing 16-2**, HARMONICS = ALL specifies that solutions are to be computed for all meaningful harmonics.
2. A DSYM command appears in all four subcases of **Listing 16-2**. The first three subcases specify that the boundary conditions for both side 1 and side 2 are to be symmetric. This specification has two effects: 1) loads need only to be specified for the first quadrant of the structure (the fundamental region in this case), and 2) output is only processed for the segments in the first quadrant of the model irrespective of the analyst's requests for output in other segments. The particular lateral gravity condition considered in the fourth subcase is symmetric with respect to side 1 of the fundamental region and antisymmetric with respect to a plane perpendicular to side 1 of the fundamental region. It must be emphasized that the appropriate boundary conditions for each of the symmetry conditions involved in this example are internally applied by the program.
3. Note that the specifications of loading conditions in the Case Control Section conform to the procedural rules for static analysis in MSC.Nastran.
4. Requests for output for selected segments are controlled through the NOOUTPUT Case Control command. For **Listing 16-1**, output requests are limited to the first segment, i.e., the fundamental region. For **Listing 16-2**, output requests for all four subcases are limited to segments in the first quadrant because of the presence of the DSYM commands, even though output for ALL right-handed segments is requested. The specification of requests for specific output quantities in the Case Control Section conform with the procedural rules for static analysis in MSC.Nastran.

16.6 Axisymmetric Example

The following cyclic symmetry example problem uses the same cylinder model that was discussed in “[Example 5 -- Buckling of a Cylinder Under Uniform Axial Load](#)” on page 514.

A five degrees strip, as shown in [Figure 16-10](#), is modeled using the axisymmetric features.

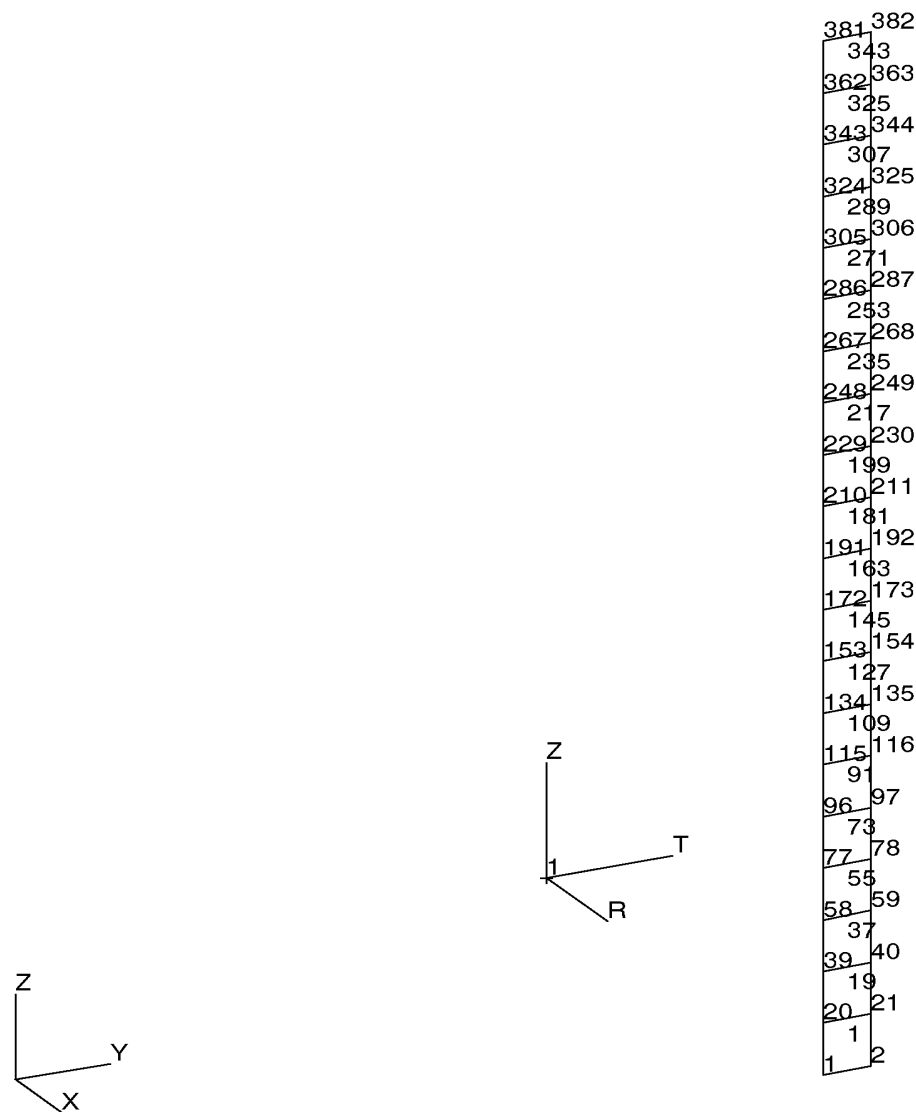


Figure 16-10 A Five Degrees Strip of a Cylinder Using Axisymmetry

A copy of the input file is shown in [Figure 16-3](#). Since only a five degrees strip is modeled, seventy-two segments are required to generate the full cylinder model internally. The CYSYM entry accomplishes this purpose. Furthermore, the boundary points for segment one are defined on the CYJOIN entries--one each for side one and side two. Note that with axisymmetry, no interior points are allowed. The applied load is a uniform compressive load applied at the top of the cylinder, using the LOADCYT/TABLED1/FORCE combinations. The TABLED1 defines the spatial distribution around the circumference of the cylinder-which in this case is evenly distributed. Since the structure is axisymmetric, and the load is constant around the circumference, only the zeroth harmonic is required for

the analysis. The displacements at the top of the cylinder and stresses at the top and bottom of the cylinders are shown in **Figure 16-11**. The same output using the full model is shown in **Figure 16-12** for comparison.

Listing 16-3 Input File for Five Degrees Strip of Cylinder

```
$
$ filename - cyclic1.dat
$
SOL 114
TIME 600
CEND
SUBCASE 1
  TITLE= A 5 DEG STRIP
  SET 100 = 381,382
  SET 200 = 1,343
  SET 1000 = 1
  SET 1 = 0
  HARMONICS = 1
  NOUTPUT = 1000
  LOAD = 1
  SPC = 1
  DISPLACEMENT= 100
  SPCFORCES=ALL
  STRESS= 200
BEGIN BULK
PARAM  POST  -1
PARAM,SNORM,20.
$ Elements and Element Properties for region : pshell.1
CYSYM,72,AXI
CYJOIN,1,T2,1,20,39,58,77,96
,115,134,153,172,191,210,229,248,
,267,286,305,324,343,362,381
$
CYJOIN,2,T2,2,21,40,59,78,97
,116,135,154,173,192,211,230,249,
,268,287,306,325,344,363,382
$
SPC1,1,123,1,2
SPC1,1,12,381,382
$
LOADCYT,1,10,11,1
TABLED1,10
,0.,1.,360.,1.,ENDT
FORCE,11,381,1,1388.889,,,-1.
$
```

Listing 16-3 Input File for Five Degrees Strip of Cylinder (continued)

```

PSHELL 1 1 .03 1 1
CQUAD4 1 1 1 2 21 20 0. 0.
CQUAD4 19 1 20 21 40 39 0. 0.
CQUAD4 37 1 39 40 59 58 0. 0.
CQUAD4 55 1 58 59 78 77 0. 0.
CQUAD4 73 1 77 78 97 96 0. 0.
CQUAD4 91 1 96 97 116 115 0. 0.
CQUAD4 109 1 115 116 135 134 0. 0.
CQUAD4 127 1 134 135 154 153 0. 0.
CQUAD4 145 1 153 154 173 172 0. 0.
CQUAD4 163 1 172 173 192 191 0. 0.
CQUAD4 181 1 191 192 211 210 0. 0.
CQUAD4 199 1 210 211 230 229 0. 0.
CQUAD4 217 1 229 230 249 248 0. 0.
CQUAD4 235 1 248 249 268 267 0. 0.
CQUAD4 253 1 267 268 287 286 0. 0.
CQUAD4 271 1 286 287 306 305 0. 0.
CQUAD4 289 1 305 306 325 324 0. 0.
CQUAD4 307 1 324 325 344 343 0. 0.
CQUAD4 325 1 343 344 363 362 0. 0.
CQUAD4 343 1 362 363 382 381 0. 0.
$
MAT1 1 1.+7 3.84+6 .3 0. 0. 0. 0.
$ Nodes of Group : group5
GRID 1 1 10. 0. 0.1
GRID 2 1 10. 5. 0.1
GRID 20 1 10. 0. 1.1
GRID 21 1 10. 5. 1.1
GRID 39 1 10. 0. 2.1
GRID 40 1 10. 5. 2.1
GRID 58 1 10. 0. 3.1
GRID 59 1 10. 5. 3.1
GRID 77 1 10. 0. 4.1
GRID 78 1 10. 5. 4.1
GRID 96 1 10. 0. 5.1
GRID 97 1 10. 5. 5.1
GRID 115 1 10. 0. 6.1
GRID 116 1 10. 5. 6.1
GRID 134 1 10. 0. 7.1
GRID 135 1 10. 5. 7.1
GRID 153 1 10. 0. 8.1
GRID 154 1 10. 5. 8.1
GRID 172 1 10. 0. 9.1
GRID 173 1 10. 5. 9.1
GRID 191 1 10. 0. 10.1

```

Listing 16-3 Input File for Five Degrees Strip of Cylinder (continued)

```

GRID      229      1      10.      0.      12.1
GRID      230      1      10.      5.      12.1
GRID      248      1      10.      0.      13.1
GRID      249      1      10.      5.      13.1
GRID      267      1      10.      0.      14.1
GRID      268      1      10.      5.      14.1
GRID      286      1      10.      0.      15.1
GRID      287      1      10.      5.      15.1
GRID      305      1      10.      0.      16.1
GRID      306      1      10.      5.      16.1
GRID      324      1      10.      0.      17.1
GRID      325      1      10.      5.      17.1
GRID      343      1      10.      0.      18.1
GRID      344      1      10.      5.      18.1
GRID      362      1      10.      0.      19.1
GRID      363      1      10.      5.      19.1
GRID      381      1      10.      0.      20.1
GRID      382      1      10.      5.      20.      1
$
CORD2C      1              0.      0.      0.      0.      0.      1.
+          A
+          A 1.      0.      0.
ENDDATA

```

```

0
SUBCASE = 1      SEGMENT = 1

DISPLACEMENT VECTOR

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
      381      G      .0      .0      -1.058190E-01  2.021944E-16  -2.435790E-02  5.526086E-19
      382      G      .0      .0      -1.058190E-01  1.009687E-16  -2.435790E-02  1.583684E-17

0
HARMO
1  A 5 DEG STRIP      FEBRUARY  4, 1997  MSC/NASTRAN  1/23/97  PAGE  49

0
SUBCASE = 1      SEGMENT = 1

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)
ELEMENT      FIBRE      STRESSES IN ELEMENT COORD SYSTEM      PRINCIPAL STRESSES (ZERO SHEAR)
ID.      DISTANCE      NORMAL-X      NORMAL-Y      SHEAR-XY      ANGLE      MAJOR      MINOR      VON MISES
0      1      -1.500000E-02      -7.481326E+03      -5.592213E+04      2.894851E-10      .0000      -7.481326E+03      -5.592213E+04      5.258216E+04
      1.500000E-02      -5.769141E+03      -5.021485E+04      2.903178E-10      .0000      -5.769141E+03      -5.021485E+04      4.759325E+04
0      343      -1.500000E-02      -7.481326E+03      -5.592213E+04      -5.481948E-10      .0000      -7.481326E+03      -5.592213E+04      5.258216E+04
      1.500000E-02      -5.769141E+03      -5.021485E+04      -5.431988E-10      .0000      -5.769141E+03      -5.021485E+04      4.759325E+04

```

Figure 16-11 Abridged Output Using Axisymmetry

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		
381	G	.0	.0	-1.058189E-01	.0	-2.433120E-02	3.729655E-16		
382	G	.0	.0	-1.058189E-01	.0	-2.442414E-02	-5.255163E-16		
S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)									
ELEMENT ID.		FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON MISES
			NORMAL -X	NORMAL -Y	SHEAR -XY	ANGLE	MAJOR	MINOR	
0	1	-1.500000E-02	-7.482788E+03	-5.592399E+04	-1.636876E-10	.0000	-7.482788E+03	-5.592399E+04	5.258343E+04
		1.500000E-02	-5.769486E+03	-5.021298E+04	-1.697938E-10	.0000	-5.769486E+03	-5.021298E+04	4.759125E+04
0	343	-1.500000E-02	-7.482788E+03	-5.592399E+04	-4.949596E-10	.0000	-7.482788E+03	-5.592399E+04	5.258343E+04
		1.500000E-02	-5.769486E+03	-5.021298E+04	-2.932691E-10	.0000	-5.769486E+03	-5.021298E+04	4.759125E+04

Figure 16-12 Abridged OUtput Using Full Model

APPENDIX

A

Glossary of Terms

A

Anisotropic Material. A material that has a different property in each direction at any given location in the structure. There is no material plane of symmetry associated with an anisotropic material. This material can be described by 21 independent elastic constants in the stress-strain relationship. See [Homogeneous Material](#), [Isotropic Material](#), and *orthotropic material*.

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 can be geometrically represented as surfaces or solids of revolution).

B

Basic Coordinate System. MSC.Nastran's default, built-in rectangular coordinate system. All user-defined (local) coordinate systems ultimately must be able to be transformed back into the basic coordinate system. The basic coordinate system is often called the global coordinate system in many finite element programs and textbooks. The global coordinate system has a special meaning in MSC.Nastran. See also [Global Coordinate System](#) and [Local Coordinate System](#).

Beam Orientation Vector. A user-defined vector $\vec{v}$ that is used to orient cross-sectional properties of CBAR and CBEAM elements with respect to the model's geometry. This information defines a unique beam orientation relative to the model 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. This section follows the Case Control Section.

C

Case Control Section. The section of the MSC.Nastran input (.DAT) file containing commands that 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) to be used in analysis subcases. The Case Control Section follows the Executive Control Section and precedes the Bulk Data Section.

Comment Entry. An entry used to insert user-specified comments anywhere in the input (.dat) file. The first character is a dollar sign (\$) in column 1, followed by any character. When echoed, only the first 80 characters will be printed in your .f06 file.

Composite Material. When two or more materials are combined together on a macroscopic scale. Generally each of these materials by itself may not provide the desired material properties for your design. However, by combining these materials into a

composite material, the desired material properties may be obtained. When speaking of composites, most people think of fibrous composites (e.g., kevlar, graphite, etc.). There are many other types of composite that are encountered every day, such as plywood and steel reinforced concrete. Also see [Anisotropic Material](#) and [Orthotropic Material](#).

Computational Zero. See [Machine Zero](#).

Consistent Load. The equivalent loads at the grid points computed from the applied loads on elements. The same shape function that is used in deriving the element stiffness and mass matrices is used for the derivation of the load-hence the name “consistent load.”

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. Also see [MPC](#) and [SPC](#).

Continuation. An extension of an entry when the entry requires more than one line of input data. Continuations may or may not be required, depending on the particular Bulk Data entry and its options. See Reference [6](#). for a detailed description of the use of a continuation for specific Bulk Data entries.

Coordinate System. See the specific type ([Basic Coordinate System](#), [Displacement Coordinate System](#), [Global Coordinate System](#), [Local Coordinate System](#), [Material Coordinate System](#), and [Output File](#)).

CPU Time. Stands for central processing unit time. The CPU time is a measure of the computational speed of your computer. Also see [Elapsed Time](#), [I/O Time](#), and [Turnaround Time](#).

D

Data Block. Basic units used by MSC.Nastran to store your data. The data block can be a matrix or table. As an example, the g -set stiffness matrix is stored as the k_{gg} data block; the l -set stiffness matrix is stored as the k_{ll} data block.

.dat File. Also called the input file, the .dat file contains the complete MSC.Nastran finite element model. The input file is submitted to MSC.Nastran which then executes the analysis. The input file contains the following principal sections:

NASTRAN statement	Optional
File Management Section (FMS)	Optional
Executive Control Section	Required
CEND	Required Delimiter
Case Control Section	Required

Begin Bulk	Required Delimiter
Bulk Data Section	Required
ENDDATA	Required Delimiter

.DBALL File. A file created by running MSC.Nastran that contains the permanent database. By default, the extension .DBALL is appended to the filename. This database file is needed if you want to perform a restart. See also **.MASTER File**. By default, all information pertaining to your model and solution is stored in this file. See “**Database Management**” on page 541 for ways of assigning this information to other files.

Decomposition. The first step in solving a 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.

Degrees-of-Freedom (DOFs). The motion of each grid point can be described by 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. Unlike a grid point, a scalar point has only one (non-dimensional) degree-of-freedom associated with it.

Delimiter. An entry in the MSC.Nastran input (.DAT) file that indicates the beginning or end of a section. CEND, BEGIN BULK, and ENDDATA are required delimiters in all input files.

Differential Stiffness. The stiffness term resulting from the inclusion of the higher-order terms of the strain-displacement relationships. The differential stiffness softens the linear stiffness matrix in the case of a compressive axial load and stiffens the linear stiffness matrix in the case of a tensile axial load. The differential stiffness is often called the geometric stiffness.

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 both knowledge of the structure and engineering judgement.

Displacement Coordinate System. Each grid point may have a unique displacement coordinate system, as selected in field 7 (the CD field) of the GRID Bulk Data entry. Displacements, constraints, and other grid point-based quantities are calculated and printed based on this coordinate system. The basic coordinate system is MSC.Nastran’s default displacement coordinate system.

Displacement Method. A method of structural analysis in which displacements are the unknown quantities to be determined. MSC.Nastran uses the displacement method.

DMAP. Acronym for Direct Matrix Abstraction Program. DMAP is MSC.Nastran's high-level programming language. DMAP allows advanced users to access MSC.Nastran's internal modules to modify existing solution sequences or to create customized solution methods. See the *MSC.Nastran 2005 DMAP Programmer's Guide* for further details.

DMAP Alters. A convenient method of providing you with a new analysis capability, increased efficiency, alternate solution methods, additional printouts, and / or otherwise unavailable user conveniences. A library of useful alters is delivered with your delivery media (see the *MSC.Nastran Reference Manual*, the *MSC.Nastran Common Questions and Answers*, and the *MSC.Nastran 2005 DMAP Programmer's Guide* for further details). You can also create your own DMAP Alters to suit your need.

DOF(s). See *Degrees-of-Freedom (DOFs)*.

E

Echo. A listing of the input file (.dat) written into the .f06 results file. The input file can be unsorted (it appears exactly like the .dat listing), sorted (it is alphabetized and comment entries removed), or both, as specified by the ECHO Case Control Command. This command can also be used to create a sorted copy of the whole Bulk Data file.

Elapsed Time. Also known as the wallclock time. The elapsed time is the time from the start of your job's execution to the end of the job. This time includes the effect of your computer speed (CPU), the amount of memory available on your machine, the I/O speed, and the swapping of multiple jobs on your machine. See *CPU Time*, *I/O Time*, and *Turnaround Time*.

Element Coordinate System. Each element has an element coordinate system based on the element's particular geometry and grid point ordering sequence. For many elements, 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, which is often denoted as [k]. The stiffness matrix describes the element's displacement response for a given load. See also *Global Stiffness Matrix*.

Epsilon. A measure of numerical accuracy and roundoff error provided in the .f06 results file of linear static analysis runs. A small value of epsilon, approximately less than $|10^{-9}|$, indicating a numerically stable problem. A large value of epsilon is evidence of numerical ill-conditioning. An acceptable value may also be machine dependent. See also *Ill-conditioning*.

Executive Control Section. A required section of the input file that appears before the Case Control Section. This section contains requests for the solution sequence type (the SOL statement), the CPU time limits (the TIME statement), and an optional identification entry (the ID statement). User-provided DMAP Alters are also inserted in this section.

F

.f04 File. A file created by running MSC.Nastran that contains database information and the module execution summary. By default, the extension .f04 is appended to the filename. The .F04 file is a valuable tool for debugging and evaluating the performance of your run.

.f06 File. A file created by running MSC.Nastran that contains the results of your analysis (e.g., stresses, forces, displacements, etc.). By default, the extension .f06 is appended to the filename.

Fatal Error. See *User Fatal Message*.

Fiber Direction. A term used when describing composite materials. The fiber direction is commonly referred to as the longitudinal direction or 1-direction. This is the direction that possesses the highest strength for the fiber composite material. See [Composite Material](#) and [Matrix Direction](#).

File Management Section (FMS). An optional input file section used primarily to attach or initialize MSC.Nastran databases and FORTRAN files. The FMS section, if used, precedes the Executive Control Section. This section is mostly used in conjunction with large problems.

Finite Element. The basic building block of the finite element method. Finite elements are the regularly, or nearly regularly, shaped mathematical idealizations of simple structures (e.g., beams, plates, solids) with known mathematical solutions. When individual elements are combined to represent a complex irregular structure, the resulting mathematical model approximates the behavior of the real structure.

Free Field Format. Input data format in which data fields are separated by commas or blanks.

G

Geometric Stiffness. See [Differential Stiffness](#).

Global Coordinate System. The union of all the coordinate systems defined on all the CD fields on all the GRID entries. The global coordinate system in MSC.Nastran is, therefore, not a single unique system. Note that the basic coordinate system may also, but not necessarily, be a part of the global coordinate system. Many commercial finite element programs and textbooks use the term “global coordinate system” to describe the coordinate system that MSC.Nastran calls its basic (default) coordinate system. See also [Basic Coordinate System](#), [Displacement Coordinate System](#), and [Local 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 in terms of the global coordinate system. See also [Elemental Stiffness Matrix](#).

Grid Point. A geometric point that defines model geometry and provides a point to which finite elements are connected. Grid points are located in space with respect to a particular coordinate system and are displaced 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.

Grid Point Stress. A rational scheme to interpolate and /or extrapolate known element and vertex stresses over a surface or volume (see “[Modeling Guidelines](#)” on page 311 for further details).

H

Hencky-von Mises Stress. See *von Mises stress*.

Homogeneous Material. A material that has the same property in all directions at all locations in the structure. See [Anisotropic Material](#), [Isotropic Material](#), and [Orthotropic Material](#).

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 \(UIM\)](#).

Input File. See [.dat File](#).

I/O Time. Stands for input/output time. I/O time is the time it takes your computer to transfer data between different locations in your computer during runtime (e.g., from memory to scratch disk). See [CPU Time](#), [Elapsed Time](#), and [Turnaround Time](#).

Isotropic Material. A material that has the same material property in all directions at a given location in a structure. This material can be defined by two independent elastic constants in the stress-strain relationship. See [Anisotropic Material](#), [Homogeneous Material](#), and [Orthotropic Material](#).

L

Lamina. A term used to describe composites. A lamina is a single layer consisting of a group of unidirectional fibers (or sometimes woven fibers) arranged to form a flat plate or curved shell. A lamina makes up the basic building block of a two-dimensional composite element.

Laminae. More than a single lamina.

Laminate. A stack of laminae bonded together. The principal direction of each lamina may be oriented in a different direction to obtain the desired strength and stiffness properties.

Large Field Format. Input format for Bulk Data entries in which the 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. A line element is typically connected by two grid points and is also known as a one-dimensional (1-D) element. One-dimensional elements do not imply that the elements can only be used for a 1-D structure; rather, it can be used for a 1-D, 2-D, or 3-D structure. An example of an application of 1-D elements in a 3-D structure is the use of CBARs / CRODs for frames and stringers of aircraft structures.

Linear Structure. A structure whose displacements are linearly proportional to the applied load. Once the load is removed, the structure returns to its original undeformed configuration.

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 coordinate system that accommodates the input 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 or another local coordinate system. The only rule in MSC.Nastran is that the local coordinate system ultimately must be able to be transformed back into the basic coordinate system. See also [Basic Coordinate System](#), [Displacement Coordinate System](#), and [Global Coordinate System](#).

.log File. A file created by running MSC.Nastran that contains system information and system error messages. By default, the extension .LOG is appended to the filename. You should always check this file if your job appears to fail for no apparent reason. This failure can result from such problems as insufficient disk space or that the job was cancelled by the operator. In many cases, the .LOG file provides you with a clue to the cause of the problem.

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 1.2345E-12 may be considered a machine zero. This term is also known as computational or numeric zero.

.MASTER File. A directory file created by running MSC.Nastran containing a list of all of the database sets used in your run, an index pointing to all the data blocks created and where they are stored, and the NDDL. By default, the extension .MASTER is appended to the filename. This file is needed if you want to perform restarts. See also **NDDL** and **.DBALL File**.

Material Coordinate System. An optional coordinate system used to orient orthotropic or anisotropic material properties. For isotropic materials, a material coordinate system is not needed.

Matrix Direction. A term used to describe composite materials. Matrix direction is commonly known as the transverse direction or 2-direction. The matrix material by itself has very low strength; however, it holds the fibers together to form a structure that produces a high strength to weight ratio (see **Composite Material** and **Fiber Direction**).

Mechanism. A mechanism occurs when part of a structure is capable of rigid body (strain-free) motion. In linear static analysis, the presence of a mechanism produces a singularity failure in the solution.

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

MPC. See **Multipoint Constraint**.

Multipoint Constraint. A convenient way to impose a linear relationship between two or more degrees of freedom. Common applications of the multipoint constraint include: defining the relative motion between two grid points as a degree of freedom or defining the average of the motions of several grid points as a degree of freedom.

N

NASTRAN Statement. An optional statement that, 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 “**nastran Command and NASTRAN Statement**” on page 1 of the *MSC.Nastran Quick Reference Guide* for further information.

NDDL. Stands for MSC.Nastran Data Definition Language and is the internal MSC.Nastran language that describes the database. The NDDL describes the content of the data blocks, checks whether data blocks and parameters are permanently stored, and sets up dependency checks for efficient restarts. Knowledge of the NDDL is not required to use MSC.Nastran efficiently.

Nonlinear (Geometric). Structural displacements that are larger than those allowed by small displacement-based theory are said to be geometrically nonlinear. Large displacements require the use of special nonlinear solution sequences in MSC.Nastran (e.g., Solution 106).

Nonlinear (Material). A material in which stress is not linearly proportional to strain, such as rubber. When the load is removed, your structure may not return to its original undeformed state. Nonlinear materials require the use of special nonlinear solution sequences in MSC.Nastran (e.g., Solution 106).

Numeric Zero. See [Machine Zero](#).

O

One-Dimensional (1-D) Element. See [Line Element](#).

Orthotropic Material. This is a special anisotropic material that can be used for plate and shell elements. It contains three orthogonal planes of material symmetry at a given location in the structure. This material can be defined by nine independent elastic constants in the stress-strain relationships. See [Anisotropic Material](#), [Homogeneous Material](#), and [Isotropic Material](#).

Output File. See the [.f04 File](#), [.f06 File](#), and [.log File](#).

P

PARAM,AUTOSPC. A parameter that controls the detection and constraint of obvious singularities in the model. PARAM,AUTOSPC is in operation by default in all solution sequences except for Solution 24. PARAM,AUTOSPC produces a grid point singularity table in the .f06 output file; this table lists the singularities that were detected and constrained. You should always check this singularity table to make sure that it is your intention for these DOFs to be constrained.

Parameter. Parameters are used to request special program features and to input data relating to these features. Parameters are specified on PARAM Bulk Data entries and PARAM Case Control commands (see “[Generating the Cylinder in MSC.Patran](#)” on page 681). A complete listing of parameter functions is included in “[Parameters](#)” on page 601 of the *MSC.Nastran Quick Reference Guide*.

.pch. A file created by running MSC.Nastran with the ECHO=PUNCH Case Control command. By default, the extension .pch is appended to the filename. The punched file is an ASCII file that can be used as part of a future input file.

.plt. A file created by running MSC.Nastran that contains the NASTRAN plotter file. By default, the extension .plt is appended to the filename.

Postprocessor. A graphics package that is designed to help you interpret and display your finite element results. Common postprocessing operations include x-y plots of numerical data, deformed shape plots of model geometry, and color stress contour plots.

Preprocessor. A graphics package designed to help to create the finite element model. Typical preprocessors create geometry, create mesh elements, apply loads and constraints, and perform certain types of error checks.

R

Restart. A procedure that allows you to systematically continue from a previous analysis without re-doing your problem from the beginning.

Results File. See the [.f06 File](#).

Rigid Body Motion. Rigid body motion occurs when the structural model is free to displace in one or more directions (displacement without strain). A simple example of rigid body motion occurs when you move a pencil from one location on your desk to another. No strain occurs in the pencil-only translation and rotation as a rigid body. In static analysis, the possibility of rigid body motion due to an insufficiently constrained structure results in a singularity in the stiffness matrix. Consequently, the solution of the problem fails during decomposition of the stiffness matrix.

Rigid Element. Also known as an R-type element. The name “rigid” element is somewhat misleading since it includes both rigid and interpolation elements. Rigid elements consist of RBAR, RBE1, RBE2, RROD, and RTRPLT; these elements create a rigid connection between the connected degrees of freedom. Interpolation elements include the RBE3 and RSPLINE. The RBE3 is often used to distribute loads and mass. The RSPLINE is often used to model mesh transition.

Rotation. Displacement about a coordinate axis. A grid point has three rotational degrees of freedom, one about each axis. See also [Degrees-of-Freedom \(DOFs\)](#) and [Translation](#).

S

Scalar Point. A scalar point is defined by the SPOINT entry. It is associated with only one degree of freedom and has no geometric location associated with it.

Set. A collection or grouping of selective items in the MSC.Nastran model. It is primarily used for limiting the amount of output data.

Single-Point Constraint (SPC). The constraint of one or more degrees of freedom at a grid point, thereby enforcing displacement (often zero displacement) of the grid point in the affected component directions. For example, the grid point at the fixed end of a cantilever beam is constrained (i.e., SPC'd) in all six DOFs. Reaction forces, called forces of single-point constraint (SPCF), may be 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 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 “slowly applied” so that no significant dynamic effects are caused.

Small Displacements. A requirement of linear structural analysis. Displacements must be sufficiently small enough so that they do not 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 the data fields are eight columns wide.

Solid Element. Elements resembling bricks (eight corners), wedges (six corners), or pyramids (four corners). Also called three-dimensional elements. Popular MSC.Nastran solid elements include the CHEXA, CTETRA, and CPENTA.

Solution Sequence. A prepackaged set of DMAP instructions designed to solve a particular type of engineering problem. The SOL command in the Executive Control Section is used to inform MSC.Nastran which solution sequence to use; for example, SOL 101 is used to specify linear static analysis. See also [DMAP](#).

SPC. See [Single-Point Constraint \(SPC\)](#).

Spring Element. An element representing a simple, single degree of freedom extensional or rotational spring. Also called a zero-dimensional element or scalar element. The CELASi family of elements are spring elements.

Static. In the finite element sense, static means that the structural model is constrained to prevent rigid body motion (static equilibrium exists) and that loads are assumed to be “slowly applied,” thereby inducing no dynamic effects.

Stiffness Matrix. See [Global Stiffness Matrix](#) and [Global Stiffness Matrix](#).

Subcase. Subcases allow multiple individual load cases and/or multiple boundary conditions to be analyzed in the same MSC.Nastran run, thereby achieving greater computational efficiency than with separate runs.

Superelement Analysis. A very efficient method used by MSC.Nastran to perform substructure analysis.

Surface Element. Elements, such as thin plates (which are flat) or shells (which are curved). Also called two-dimensional (2-D) elements. Popular MSC.Nastran surface elements include the CQUAD4 (quadrilateral) and CTRIA3 (triangular) elements. Two-dimensional elements do not imply that the elements can only be used for a 2-D structure; rather, it can be used for a 1-D, 2-D, or 3-D structure. An example of the use of 2-D elements in a 3-D structure is the use of CQUAD4s for modeling the panels of an automobile.

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). MSC.Nastran contains a set of solutions sequences called cyclic symmetric solution sequences that perform this task automatically (see “[Cyclic Symmetry](#)” on page 559).

System Message. System Messages refer to diagnostics associated with program or system 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 [Degrees-of-Freedom \(DOFs\)](#) and [Rotation](#).

Turnaround Time. The time from when you submit your job to when you obtain results. Turnaround time includes the effect of queuing at your computer site; therefore, it may be longer than the elapsed time. See [CPU Time](#), [Elapsed Time](#), and [I/O Time](#).

Two-Dimensional (2-D) Element. See [Surface Element](#).

U

User Fatal Message (UFM). An MSC.Nastran message describing an error severe enough to cause the program to terminate. See the [MSC.Nastran Reference Manual](#) for a complete listing of all of the UFM.

User Information Message (UIM). An MSC.Nastran message that provides general information. The issuance of an UIM is not necessarily indicative of a problem. MSC.Nastran does not terminate due to an UIM. See the [MSC.Nastran Reference Manual](#) for a complete listing of all of the UIMs.

User Warning Message (UWM). An MSC.Nastran message warning of an atypical situation; the user must determine whether or not a problem exists. In general, MSC.Nastran does not terminate due to an UWM. See the [MSC.Nastran Reference Manual](#) for a complete listing of all of the UWMs.

.usrojb File. A file created by running MSC.Nastran that contains the object file of the user-written DMAPs. By default, the extension .USROBJ is appended to the filename. This file can be deleted, and is deleted by default, if you are not creating your own DMAP.

.ursou File. A file created by running MSC.Nastran that contains the source file of the user-written DMAPs. By default, the extension .USRSOU is appended to the filename. This file can be deleted, and is deleted by default, if you are not creating your own DMAP.

V

von Mises Stress. This is a failure criteria commonly used for ductile materials. This stress value is an invariant quantity that is independent of the element coordinate system used; it is always a positive number. For a general stress state (nonprincipal axes), von Mises stress is given by the following equation:

$$\sigma_{von} = \frac{1}{\sqrt{2}} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{yz})^2 + 6(\tau_{zx})^2 + 6(\tau_{xy})^2]^{1/2}$$

X

.xdb. An optional file created by running MSC.Nastran that contains the graphical database used by MSC / Aries, MSC / XL, and other graphic packages. By default, the extension .xdb is appended to the filename.

APPENDIX

B

Grid Point Weight Generator

- Introduction to Grid Point Weight Generation
- Commonly Used Features
- Example with Direction-Dependent Masses

2.1 Introduction to Grid Point Weight Generation

The grid point weight generator (GPWG) calculates the masses, centers of gravity, and inertias of the mathematical model of the structure. The data are extracted from the mass matrix by using a rigid-body transformation calculation. Computing the mass properties is somewhat complex because a finite element model may have directional mass properties, that is, the mass may differ in each of the three coordinate directions. From a mathematical point of view, the MSC.Nastran mass may have tensor properties similar to the inertia tensor. This complexity is reflected in the GPWG output. All of the transformations used in calculating the mass properties are shown for the general case. Since most models have the same mass in each of the three coordinate directions, the GPWG output provides more information than you generally need.

To avoid unnecessary confusion and at the same time provide the necessary information for the advanced user, the discussion of the GPWG is separated into two sections. In “**Commonly Used Features**” on page 615, a basic discussion is given that should satisfy most users. If you need additional information, read “**Example with Direction-Dependent Masses**” on page 618.

In both sections, a simple model consisting of four concentrated masses is used to demonstrate the GPWG output. In the first section, the mass is the same in each direction. For the second section, the mass is different in each of the three directions.

2.2 Commonly Used Features

To demonstrate the typical output generated by the GPWG, a small model consisting of four concentrated masses as shown in [Figure 2-1](#) is used. This model is typical of most models because the mass is the same in each coordinate direction. The number of masses has been kept small so you can better understand the physics.

Concentrated masses are located at four different grid points. The displacement coordinate system for each of the grid points is the basic coordinate system.

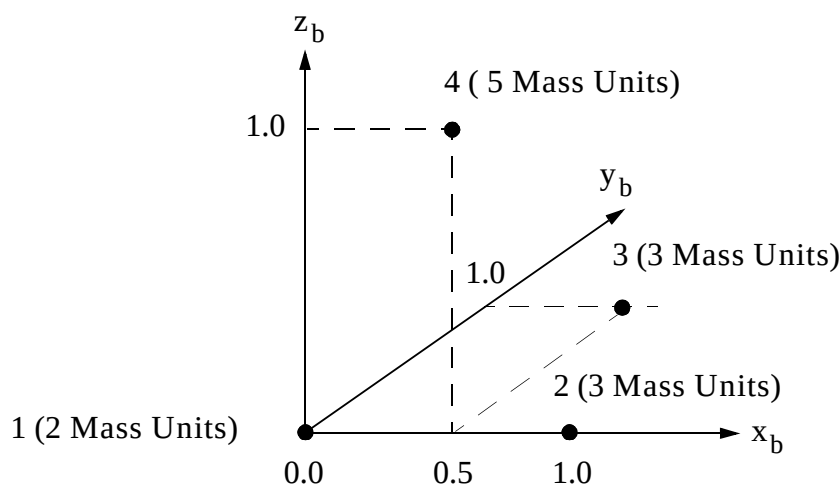


Figure 2-1 Four Concentrated Mass Model

To request the GPWG output, you must add parameter GRDPNT in either the Bulk Data Section or the Case Control Section as follows:

PARAM,GRDPNT,x

If

$x = -1$	GPWG is skipped (default).
$x = 0$	The mass properties are computed relative to the origin of the basic coordinate system.
$x > 0$	The mass properties are computed relative to grid point x . If grid point x does not exist, the properties are computed relative to the basic coordinate system.

For the four masses shown in [Figure 2-1](#), the resulting GPWG output is given in [Figure 2-2](#).

```

      O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
                        REFERENCE POINT =          0
                        M O
*  1.300000E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.000000E+00 -3.000000E+00 *
*  0.000000E+00  1.300000E+01  0.000000E+00 -5.000000E+00  0.000000E+00  7.000000E+00 *
*  0.000000E+00  0.000000E+00  1.300000E+01  3.000000E+00 -7.000000E+00  0.000000E+00 *
*  0.000000E+00 -5.000000E+00  3.000000E+00  8.000000E+00 -1.500000E+00 -2.500000E+00 *
*  5.000000E+00  0.000000E+00 -7.000000E+00 -1.500000E+00  1.000000E+01  0.000000E+00 *
* -3.000000E+00  7.000000E+00  0.000000E+00 -2.500000E+00  0.000000E+00  8.000000E+00 *
      S
      *  1.000000E+00  0.000000E+00  0.000000E+00 *
      *  0.000000E+00  1.000000E+00  0.000000E+00 *
      *  0.000000E+00  0.000000E+00  1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
      X      1.300000E+01      0.000000E+00  2.307692E-01  3.846154E-01
      Y      1.300000E+01      5.384616E-01  0.000000E+00  3.846154E-01
      Z      1.300000E+01      5.384616E-01  2.307692E-01  0.000000E+00
      I(S)
      *  5.384615E+00 -1.153847E-01 -1.923079E-01 *
      * -1.153847E-01  4.307692E+00 -1.153846E+00 *
      * -1.923079E-01 -1.153846E+00  3.538461E+00 *
      I(Q)
      *  5.503882E+00
      *      5.023013E+00
      *      2.703873E+00
      Q
      *  8.702303E-01  4.915230E-01  3.323378E-02 *
      *  3.829170E-01 -7.173043E-01  5.821075E-01 *
      *  3.099580E-01 -4.938418E-01 -8.124324E-01 *

```

Figure 2-2 GPWG Output for the Four Concentrated Mass Model

The [MO] matrix represents the rigid-body mass properties of the structure and is generally not needed for model checkout. This matrix represents an intermediate step in computing the inertia properties of the structure. The [S] matrix should always be equal to the identity matrix when the mass is the same in each coordinate direction, which is the typical case. If this matrix is not the identity matrix, inspect the model for inconsistent masses.

Following the [S] matrix are the mass and center of gravity locations. These are the most commonly used information of the GPWG output. Because the mass may be different in the three translational directions, the mass is printed for every coordinate direction. For the same reason, the center of gravity location is given for each of the three translational masses. If the mass is the same in all directions, a unique center of gravity exists and is located at the x-component of the y (or z) mass, the y-component of the x (or z) mass, and the z-component of the x (or y) mass. If the mass is not the same in all three directions, it is likely due to the CONM1, CMASSi, or DMIG input. If the reference point is specified with PARAM,GRDPNT,0 and [S] is the identity matrix, then the center of gravity location is given in the basic coordinate system. For the example, the mass of the structure is 13.0 and the center of gravity location is (0.5384, 0.2307, 0.3846) in the basic coordinate system.

If a grid point ID is used for the reference point, and [S] is an identity matrix, then the center of gravity location is in a coordinate system parallel to the basic coordinate system with an origin located at the grid point.

If the [S] matrix is equal to the identity matrix, then the [I(S)] matrix represents the inertia matrix of structure for the center of gravity with respect to the basic coordinate system, the [I(Q)] matrix is the corresponding principal moments of inertia matrix, and [Q] represents the transformation from the principal directions to the basic coordinate system.

The following additional comments on the GPWG should be noted.

- The scale factor entered with parameter WTMASS is applied to the assembled element mass before the GPWG. The GPWG module, however, converts mass back to the original input units that existed prior to the scaling effect of the parameter WTMASS. (Note: The parameter WTMASS is not applied to M2GG or M2PP input, but the M2GG mass is assembled into the mass matrix prior to GPWG. Therefore, for GPWG output only, the M2GG mass is scaled by the same parameter as the element mass. M2GG input may be scaled independently using the CM2 parameter.)
- The GPWG is performed on the *g*-size mass matrix, which is the mass matrix prior to the processing of the rigid elements, MPCs, and SPCs.
- The mass at scalar points and fluid-related masses are not included in the GPWG calculation.
- The GPWG for a superelement does not include the mass from upstream superelements. Therefore, the GPWG for the residual structure includes only the mass on the residual points. The center of gravity location is also based on the mass of the current superelement only. (Use WEIGHTCHECK = YES to print the total mass for all Superelements.)
- If a large mass is used for enforced motion, the large mass dominates the structural mass. For model checkout, it is recommended to remove the large mass and constrain the driving point. A static analysis is a convenient way to generate a mass matrix and obtain output from the GPWG.
- The output from the GPWG is for information purposes only and is not used in the analysis.

2.3 Example with Direction-Dependent Masses

In the previous section, the mass was the same in each of the three coordinate directions thereby producing a unique center of gravity location. However, if scalar masses are used, the total mass may have different properties in each direction, and the center of gravity may not be a unique location. This effect is shown in the output by providing the direction and center of gravity for each of the three mass principal components.

When using directional mass, the axes about which the inertia matrix is calculated are referred to as the principal mass axes. The principal mass axes may not necessarily intersect. However, these axes provide uncoupled rotation and translation mass properties. If the structural model is constructed using only physical masses, the three principal mass values printed out are equal, the center of gravity is unique, and the principal mass axes intersect at the center of gravity.

To demonstrate all of the features of the GPWG module, the four-mass sample problem discussed in the previous section is modified so that the mass is not equal in each of the three translational directions (see [Figure 2-3](#)). Furthermore, different displacement coordinate systems are used for the grid points. The displacement coordinate system for grid point 1 is the local rectangular system 1, which is oriented at an angle of 45 degrees (about the z_b axis). The displacement coordinate system for grid point 3 is the local rectangular system 3, which is oriented at an angle of 60 degrees (about the z_b axis). The grid point locations and masses are summarized in [Table 2-1](#).

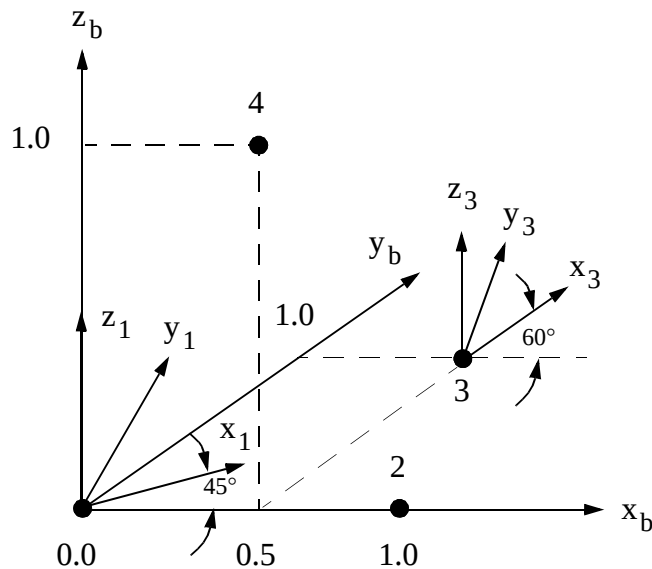


Figure 2-3 Four Concentrated Mass Model.

Table 2-1 Location and Size of Masses

Grid ID	Location Basic System (CP Fields)			Mass Global System (CD Fields)		
	x_b	y_b	z_b	x_{CD}	y_{CD}	z_{CD}
1	0	0	0	2	3	5
2	1	0	0	2	3	5
3	0.5	1	0	2	3	5
4	0.5	0	1	2	3	5

The GPWG output for the four mass model is shown in [Figure 2-4](#).

*** USER WARNING MESSAGE 3042 MODULE = GPWG
INCONSISTENT SCALAR MASSES HAVE BEEN USED. EPSILON/Delta = 3.8054429E-02

OUTPUT FROM GRID POINT WEIGHT GENERATOR

REFERENCE POINT = 1
M 0

```
* 9.250000E+00 -9.330128E-01 0.000000E+00 0.000000E+00 2.000000E+00 -2.966506E+00 *
* -9.330128E-01 1.075000E+01 0.000000E+00 -3.000000E+00 0.000000E+00 6.058013E+00 *
* 0.000000E+00 0.000000E+00 2.000000E+01 5.000000E+00 -1.000000E+01 0.000000E+00 *
* 0.000000E+00 -3.000000E+00 5.000000E+00 8.000000E+00 -2.500000E+00 -1.500000E+00 *
* 2.000000E+00 0.000000E+00 -1.000000E+01 -2.500000E+00 9.500000E+00 0.000000E+00 *
* -2.966506E+00 6.058013E+00 0.000000E+00 -1.500000E+00 0.000000E+00 7.495513E+00 *
```

Rigid-Body Mass
Properties Matrix About
the Reference Point

S

```
* 4.321332E-01 9.018098E-01 0.000000E+00 *
* -9.018098E-01 4.321332E-01 0.000000E+00 *
* 0.000000E+00 0.000000E+00 1.000000E+00 *
```

Transformation from the
Principal Mass to the Basic
Direction

DIRECTION

MASS AXIS SYSTEM (S)	MASS	X-C.G.	Y-C.G.	Z-C.G.
X	1.119709E+01	3.480388E-02	6.023980E-01	2.512494E-01
Y	8.802916E+00	-6.515555E-03	-4.426965E-02	2.484108E-01
Z	2.000000E+01	-9.385824E-03	5.589382E-01	0.000000E+00

Center of Gravity Relative
to the Reference Point in
the Principal Mass Axes
System

I(S)

```
* 4.376953E+00 -8.768300E-01 6.624477E-01 *
* -8.768300E-01 5.623007E+00 -3.419880E-01 *
* 6.624477E-01 -3.419880E-01 3.431904E+00 *
```

Moments of Inertia with
Respect to Principal Mass
Axes for the Center of
Gravity

I(Q)

```
* 4.463246E+00 *
* 6.075616E+00 *
* 2.893001E+00 *
```

Principal Moments of
Inertia

Q

```
* 7.201511E-01 4.586845E-01 5.205678E-01 *
* -3.717336E-01 8.885992E-01 -2.687111E-01 *
* -5.858298E-01 0.000000E+00 8.104341E-01 *
```

Transformation from the
Principal Direction of the
Moment of Inertia Ellipsoid to the
Principal Mass Axes

Figure 2-4 Output from the Grid Point Weight Generator

Before showing how each of the matrices are computed, a few items should be noted for this model:

- User Warning Message 3042 is printed to inform you that inconsistent scalar masses were used. This message occurs because there are different scalar masses in the three components. In general, if you are using structural mass and / or CONM2s, you should not get this message.
- The rigid-body mass matrix $[MO]$ is computed with respect to the reference grid point1 in the basic coordinate system. Grid point 1 is used for this example because PARAM,GNDPNT,1 is entered in the Bulk Data Section.
- The mass and center of gravity location shown are not in the basic coordinate system for this example. The mass and center of gravity are computed and printed in the principal mass coordinate system. This principal mass coordinate system should not be confused with the principal axes discussed in most text books. The principal mass axes in MSC.Nastran are the axes that have no coupling terms between the translational masses (diagonal translational mass matrix). Also, the MSC.Nastran principal mass axes are not the axes of the inertia ellipsoid. The $[S]$ matrix is the transformation from the principal mass direction to the basic coordinate system.

This additional step may sound confusing, but it is necessary. In real structures, the mass of structure is generally the same in all directions, so there is no coupling between the translational mass terms. Since text books are written to solve real structural problems, there is no need to discuss the principal mass axes, as they are called in MSC.Nastran. However, with MSC.Nastran you are not restricted to the same mass in each coordinate direction--a situation that may not be physically realizable but still is quite useful for certain modeling situations. Therefore, the additional step of computing principal mass axes is necessary.

- If your model has the same mass in all coordinate directions, then the matrix is the identity matrix indicating that the principal mass axes is the basic coordinate system. Always check the $[S]$ matrix. If it is not the identity matrix, verify that the mass distribution is correct. Do not use the directional mass and center of gravity location blindly. Remember, these quantities are in the principal mass axes.

To fully understand how the GPWG module works, it is useful to trace the steps MSC.Nastran follows to generate the output shown in [Figure 2-4](#). The following shows the step-by-step procedure for the four mass example.

1. The GPWG module uses the global mass matrix, which is the mass matrix before any constraints are applied. In this example, there are four grids, each with six degrees of freedom, resulting in a total of 24 degrees of freedom in the mass matrix. The $[MJJ]$ matrix is shown in [Figure 2-5](#).

Figure 2-5 Global Mass Matrix

Figure 1 displays a 24x24 grid representing the spatial distribution of the $[MJJ]$ index. The grid is divided into four 6x6 sub-grids, each labeled "Grid 1", "Grid 2", "Grid 3", and "Grid 4". Each sub-grid contains a 3x3 pattern of values: 2, 3, and 5. The values are distributed as follows:

- Grid 1 (Rows 1-6, Columns 1-6):** 2 at (1,1), 3 at (2,2), 5 at (3,3).
- Grid 2 (Rows 7-12, Columns 7-12):** 2 at (7,7), 3 at (8,8), 5 at (9,9).
- Grid 3 (Rows 13-18, Columns 13-18):** 2 at (13,13), 3 at (14,14), 5 at (15,15).
- Grid 4 (Rows 19-24, Columns 19-24):** 2 at (19,19), 3 at (20,20), 5 at (21,21).

All other cells in the grid contain the value 0. The grid is labeled $[MJJ]=$ on the left and has row and column indices from 1 to 24.

The matrix shows the mass contribution for each of the four grid points. Note that the coordinate system associated with rows and columns 1 through 6 (grid point 1) is coordinate system 1, the coordinate system associated with rows and columns 13 through 18 (grid point 3) is coordinate system 3. The remaining rows and columns are in the basic coordinate system.

2. To generate the 6 x 6 rigid-body mass matrix $[MO]$ for the structure, it is necessary to compute the mass matrix in the basic coordinate system relative to the reference point. This computation requires the transformation matrix $[D]$ that relates the rigid-body displacements in the global system u_g to the six unit displacements in the basic coordinate system located at the reference grid point (u_O) as shown in [Eq. 2-1](#).

$$\{u_o\} = [D]\{u_o\} \text{ (Reference Point)} \quad \text{Eq. 2-1}$$

The transformation matrix $[D]$ is assembled from the individual transformation matrices $[d]_i$ computed for each grid point. Each individual transformation $[d]_i$ matrix consists of two transformations: $[Tr]_i$, which relates the location of the grid point to the reference grid point in the basic coordinate system, and $[Ti]_i$, which relates the global coordinate system at the grid point to the basic coordinate system.

The $[Tr]_i$ transformation matrices are first computed by constructing the location vectors in the basic coordinate system T_i for each grid point in the model relative to the reference point as shown in [Eq. 2-2](#).

$$\{r\}_i = \{R\}_i - \{R\}_0 = \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \end{Bmatrix} \quad \text{Eq. 2-2}$$

The location vectors for the example are as follows:

$$\{r\}_1 = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_2 = \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_3 = \begin{Bmatrix} .5 \\ 1 \\ 0 \end{Bmatrix}; \{r\}_4 = \begin{Bmatrix} .5 \\ 0 \\ 1 \end{Bmatrix} \quad \text{Eq. 2-3}$$

Using the location vectors, the grid point transformation matrix $[Tr]_i$ is computed for each grid point by expanding the location vectors to a 3×3 matrix as shown in [Eq. 2-4](#).

$$[Tr]_i = \begin{bmatrix} 0 & r_3 & -r_2 \\ -r_3 & 0 & r_1 \\ r_2 & -r_1 & 0 \end{bmatrix} \quad \text{Eq. 2-4}$$

For the example problem, the grid point transformation matrices are

$$[Tr]_1 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}; [Tr]_2 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & -1 & 0 \end{bmatrix}; [Tr]_3 = \begin{bmatrix} 0 & 0 & -1 \\ 0 & 0 & .5 \\ 1 & -.5 & 0 \end{bmatrix}; [Tr]_4 = \begin{bmatrix} 0 & 1 & 0 \\ -1 & 0 & .5 \\ 0 & -.5 & 0 \end{bmatrix} \quad \text{Eq. 2-5}$$

The coordinate system transformation matrices from the global coordinate system to the basic system coordinates are given by the direction cosine matrices as follows:

$$[Ti]_1 = \begin{bmatrix} \cos(45^\circ) & -\sin(45^\circ) & 0 \\ \sin(45^\circ) & \cos(45^\circ) & 0 \\ 0 & 0 & 1 \end{bmatrix}; [Ti]_2 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \text{Eq. 2-6}$$

$$[Ti]_3 = \begin{bmatrix} \cos(60^\circ) & -\sin(60^\circ) & 0 \\ \sin(60^\circ) & \cos(60^\circ) & 0 \\ 0 & 0 & 1 \end{bmatrix}; [Ti]_2 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The grid point transformation $[Tr]_i$ and the coordinate system transformation $[Ti]_i$ are combined to form the individual grid point transformation matrix $[d]_i$ for each grid point using [Eq. 2-7](#).

$$[d]_i = \begin{bmatrix} Ti^T & | & Ti^T \cdot Tr \\ \hline 0 & | & Ti^T \end{bmatrix} \quad \text{Eq. 2-7}$$

The rows of the each $[d]_i$ form the columns of the global transformation matrix $[D]^T$ as shown in [Eq. 2-8](#).

$$[D]^T = [d_1^T | d_2^T | \dots | d_n^T] \quad \text{Eq. 2-8}$$

Using [Eq. 2-8](#), the global transformation matrix for the example is

$$\begin{bmatrix} .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & .5 & -.866 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ .707 & .707 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & .866 & .5 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 1 & .5 & -.866 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & .707 & .707 & 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & -5 & .866 & .5 & 0 & 1 & 0 & -.5 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 1 & -.067 & 1.116 & 0 & 0 & 0 & 1 & 0 & .5 & 0 \end{bmatrix} \quad \text{Eq. 2-9}$$

Grid 1 Grid 2 Grid 3 Grid 4

Using the global transformation matrix $[D]$, the rigid-body mass matrix about the reference point in the basic coordinate system $[MO]$ is obtained by [Eq. 2-10](#).

$$[MO] = [D]^T [MJJ] [D] \quad \text{Eq. 2-10}$$

For the example, $[MO]$ is determined to be

$$[MO] = \begin{bmatrix} 9.25 & -.933 & 0. & 0. & 2. & -2.967 \\ -.933 & 10.75 & 0. & -3. & 0. & 6.058 \\ 0. & 0 & 20. & 5. & -10. & 0. \\ 0. & -3 & 5. & 8. & -2.5 & -1.5 \\ 2. & 0 & -10. & -2.5 & 9.5 & 0. \\ -2.967 & 6.058 & 0. & -1.5 & 0. & 7.496 \end{bmatrix} \quad \text{Eq. 2-11}$$

Comparing the results shown in [Eq. 2-11](#) to $[MO]$ generated by the GPWG module ([Figure 2-4](#)) shows the matrices to be numerically the same.

3. The next step is to inspect the $[MO]$ to determine whether the basic coordinate system can be used as the principal mass directions. The principal mass axes are axes that have no coupling between the translational mass components. For real structures, there is no coupling in the translational mass terms in the inertia matrix. However, with inconsistent scalar masses (CONM1, CMASSi, or DMIG), you may define any type of mass matrix you desire.

To determine whether coupling exists between the translational mass terms, $[MO]$ is partitioned into four 3×3 matrices as shown in [Eq. 2-12](#).

$$[MO] = \left[\begin{array}{c|c} \bar{M}^t & \bar{M}^{tr} \\ \hline \bar{M}^{rt} & \bar{M}^r \end{array} \right] \quad \text{Eq. 2-12}$$

where the superscripts t and r refer to translation and rotation, respectively.

For this example, the translational mass partition is given by

$$[\bar{M}^t] = \begin{bmatrix} 9.25 & -.933 & 0 \\ -.933 & 10.75 & 0 \\ 0 & 0 & 20. \end{bmatrix} \quad \text{Eq. 2-13}$$

A check is made for coupling as follows:

$$\begin{aligned} \delta &= \sqrt{\sum (\bar{M}_{ij}^t)^2} & i &= j \\ \epsilon &= \sqrt{\sum (\bar{M}_{ij}^t)^2} & i &> j \end{aligned} \quad \text{Eq. 2-14}$$

If ϵ / δ is greater than .001, then excessive coupling exists preventing the basic coordinate system from being used for the principal mass directions and User Warning Message 3042 is printed. For this problem, $\delta = (9.25^2 + 10.75^2 + 20^2)^{.5}$ and $\epsilon = (.933^2 + 0.^2 + 0.^2)^{.5}$. The ratio $\epsilon / \delta = .038$ agrees with the System Warning Message 3042 shown in [Figure 2-4](#).

If needed, the principal mass directions are computed by performing an eigensolution with the translational mass components. The eigenvectors of $[\bar{M}^t]$ are the columns of the transformation matrix $[S]$.

$$[S] = [\{e_1\}, \{e_2\}, \{e_3\}] \quad \text{Eq. 2-15}$$

Using this eigenvector matrix, the partitions of the rigid-body mass matrix with respect to the principal mass direction are computed by [Eq. 2-16](#).

$$\begin{aligned} [\bar{M}^t] &= [S]^T [\bar{M}^t] [S] \\ [\bar{M}^{tr}] &= [S]^T [\bar{M}^{tr}] [S] \\ [\bar{M}^r] &= [S]^T [\bar{M}^r] [S] \end{aligned} \quad \text{Eq. 2-16}$$

The $[S]$ matrix for the example is given by

$$[S] = \begin{bmatrix} .432 & .902 & 0. \\ -.902 & .432 & 0. \\ 0. & 0. & 1. \end{bmatrix} \quad \text{Eq. 2-17}$$

The $[S]$ matrix is printed after the matrix as shown in [Figure 2-4](#). This represents the transformation relating the basic coordinate system to the principal mass axes. Again, if there is no coupling between the translational mass component, which is the case for most problems, the eigensolution is not required, and the $[S]$ matrix is set equal to the identity matrix. This example was selected to demonstrate all of the features of the GPWG module, but it is not a typical problem.

4. The next step is to determine the principal masses and the center of gravity location in the principal mass axes system as shown in [Eq. 2-18](#).

$$\begin{aligned} M_x &= M_{11}^t \\ M_y &= M_{22}^t \\ M_z &= M_{33}^t \end{aligned}$$

The “centers of gravity” are

$$\begin{aligned} X_x &= \frac{M_{11}^{tr}}{M_x}, & Y_x &= \frac{-M_{13}^{tr}}{M_x}, & Z_x &= \frac{M_{12}^{tr}}{M_x} \\ X_y &= \frac{M_{23}^{tr}}{M_y}, & Y_y &= \frac{M_{22}^{tr}}{M_y}, & Z_y &= \frac{-M_{21}^{tr}}{M_y} \\ X_{tr} &= \frac{-M_{32}^{tr}}{M_z}, & Y_z &= \frac{M_{31}^{tr}}{M_z}, & Z_z &= \frac{M_{33}^{tr}}{M_z} \end{aligned} \quad \text{Eq. 2-18}$$

As can be seen, the center of gravity location is not a unique location. The center of gravity location is computed separately for the for x-, y-, and z-directions relative to the principal mass axes. Only if the mass is the same in each direction (which is typical) is there a unique center of gravity location, which is relative to the reference point in the basic coordinate system.

For this example problem, the center of gravity locations are determined to be

Table 2-2 Mass Center of Gravity Locations

Mass Component	Center of Gravity Location in Principal Mass Axes		
	X	Y	Z
x - mass	.035	0.602	0.251
y - mass	-0.007	-0.044	0.248
z - mass	-0.009	0.559	0

The center of gravity location given in **Table 2-2** is the same as shown in **Figure 2-4**.

- Following the center of gravity calculation is the calculation to determine the moments of inertia for the center of gravity with respect to the principal mass axes as shown in **Eq. 2-19**.

$$\begin{aligned}
 I_{11}^{(S)} &= M_{11}^r - M_y Z_y^2 - M_z Y_z^2 \\
 I_{12}^{(S)} &= I_{21}^{(S)} = -M_{12}^r - M_z X_z Y_z \\
 I_{13}^{(S)} &= I_{31}^{(S)} = -M_{13}^r - M_y X_y Y_y \\
 I_{22}^{(S)} &= M_{22}^r - M_z X_z^2 - M_x Z_x^2 \\
 I_{32}^{(S)} &= I_{23}^{(S)} = -M_{23}^r - M_x X_x Z_x \\
 I_{33}^{(S)} &= M_{33}^r - M_x Y_x^2 - M_y X_y^2
 \end{aligned}
 \tag{Eq. 2-19}$$

For the example, the inertia matrix $[I(S)]$ is given by

$$[I(S)] = \begin{bmatrix} 4.377 & -.877 & .662 \\ -.877 & 5.632 & -.342 \\ .662 & -.342 & 3.432 \end{bmatrix}
 \tag{Eq. 2-20}$$

- The final step is to compute the principal moments of inertia and the principal directions of the momental ellipsoid (commonly referred to as the principal axes in text books). An intermediate inertia matrix $[\tilde{I}]$ is generated by reversing the sign on the off diagonal terms of $[I(S)]$. For the inertia matrix given in $[\tilde{I}]$, is given by

$$[\tilde{I}] = \begin{bmatrix} 4.377 & .877 & -.662 \\ .877 & 5.632 & .342 \\ -.662 & .342 & 3.432 \end{bmatrix}
 \tag{Eq. 2-21}$$

An eigensolution is performed on the $[\tilde{I}]$ matrix to determine the principal directions. The resulting eigenvalues are the principal moments of inertia, which are assumed to be the diagonal terms of the principal inertia matrix $[I(Q)]$. The eigenvectors form the columns of the matrix $[Q]$, which is the transformation relating the intermediate inertia matrix $[\tilde{I}]$ to the principal inertia matrix $[I(Q)]$ as shown in **Eq. 2-22**.

$$[I(Q)] = [Q]^T [\tilde{I}] [Q] \quad \text{Eq. 2-22}$$

For the example, $[Q]$ and $[I(Q)]$ are given by

$$[I(Q)] = \begin{bmatrix} 4.463 & 0. & 0. \\ 0. & 6.076 & 0. \\ 0. & 0. & 2.893 \end{bmatrix} \quad \text{Eq. 2-23}$$

$$[Q] = \begin{bmatrix} .720 & .459 & .521 \\ -.372 & .889 & -.269 \\ -.586 & 0 & .810 \end{bmatrix}$$

The matrices $[S]$ and $[Q]$ matrices are the coordinate rotation matrices, which when taken together, relate the principal directions of the momental ellipsoid to the basic coordinate system. The matrices given in **Eq. 2-23** are in agreement with those given in **Figure 2-4**. The example for the four mass model is now complete.

APPENDIX

C

Summary of Basic Case Control Commands

The common Case Control commands used in linear static analysis are summarized below:

DEFORM	Element Deformation Static Load
DISPLACEMENT	Displacement Output Request
DSYM	Dihedral Symmetry Option in Cyclic Symmetry
ECHO	Bulk Data Echo Request
ELFORCE	Element Force Output or Particle Velocity Request
ELSDCON	Element Stress Discontinuity Output Request
ELSTRESS	Element Stress Output Request
ELSUM	Element Summary Request
ESE	Element Strain Energy Output Request
FORCE	Element Force-Output Request
GPFORCE	Grid Point Force Output Request
GPSDCON	Grid Point Stress Discontinuity Output Request
GPSTRAIN	Grid Point Strain Output Request for Printing Only
GPSTRESS	Grid Point Stress Output Request for Printing Only
GROUNDCHECK	Rigid Body Motion Grounding Check
HARMONICS	Harmonic Analysis or Printout Control
HOUTPUT	Harmonic Output Request in Cyclic Symmetry Problems
INCLUDE	Insert External File
K2GG	Direct Input Stiffness Matrix Selection
LABEL	Output Label
LINE	Maximum Lines per Printed Page
LOAD	External Static Load Set Selection
M2GG	Direct Input Mass Matrix Selection
MASTER	Redefine the MASTER Subcase
MAXLINES	Maximum Number of Output Lines
MAXMIN	MAXMIN Survey Output Request
METHOD	Real Eigenvalue Extraction Method Selection
MPC	Multipoint Constraint Set Selection
MPCFORCES	Multipoint Force of Constraint Vector Output
LOAD	Applied Load Output Request
PAGE	Page Eject
PARAM	Parameter Specification

PLOTID	Plotter Identification
REPCASE	Repeat Output Subcase Delimiter
SET	Set Definition
SKIP	Activates or deactivates the execution of subsequent commands
SPC	Single-Point Constraint Set Selection
SPCFORCES	Single-Point Force of Constraint Vector Output
STATSUB	Static Solution Selection for Buckling Analysis
STRAIN	Element Strain Output Request
STRESS	Element Stress Output Request
STRFIELD	Grid Point Stress Output Request
SUBCASE	Subcase Delimiter
SUBCOM	Combination Subcase Delimiter
SUBSEQ	Subcase Sequence Coefficients
SUBTITLE	Output Subtitle
SURFACE	Surface Definition
SYM	Symmetry Subcase Delimiter
SYMCOM	Symmetry Combination Subcase Delimiter
SYMSEQ	Symmetry Sequence Coefficients
TEMPERATURE	Temperature Set Selection
THERMAL	Temperature Output Request
TITLE	Output Title
VOLUME	Volume Definition
WEIGHTCHECK	Rigid Body Mass Reduction Check

Full descriptions of these and other Case Control commands can be found in the “**Case Control Commands**” on page 175 of the *MSC.Nastran Quick Reference Guide*.

APPENDIX

D

Summary of Basic Bulk Data Entries

The common Bulk Data entries used in linear static analysis are summarized below:

\$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
/	Removes entries on restart.
ADUMi	Dummy Element Attributes
ASET	Degrees of Freedom for the a-set
ASET1	Degrees of Freedom for the a-set, Alternate Form of ASET Entry
AXIC	Conical Shell Problem Flag
BAROR	CBAR Entry Default Values
BEAMOR	CBEAM Entry Default Values
CBAR	Simple Beam Element Connection
CBARAO	Auxiliary Output Points Along Bar Element Axis (CBAR Entry)
CBEAM	Beam Element Connection
CBEND	Curved Beam or Pipe Element Connection
CBUSH	Generalized Spring and Damper Structural Element
CDUMi	Dummy Element Connection
CELAS1	Scalar Spring Connection
CELAS2	Scalar Spring Property and Connection
CELAS3	Scalar Spring Connection to Scalar Points Only
CELAS4	Scalar Spring Property and Connection to Scalar Points Only
CHEXA	Six-Sided Solid Element Connection
CMASS1	Scalar Mass Connection
CMASS2	Scalar Mass Property and Connection
CMASS3	Scalar Mass Connection to Scalar Points Only
CMASS4	Scalar Mass Property and Connection to Scalar Points Only
CONM1	Concentrated Mass Element Connection, General Form
CONM2	Concentrated Mass Element Connection, Rigid Body Form
CONROD	Rod Element Property and Connection
CORD1C	Cylindrical Coordinate System Definition, Form 1
CORD1R	Rectangular Coordinate System Definition, Form 1
CORD1S	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
CQUAD8	Curved Quadrilateral Shell Element Connection
CQUADR	Quadrilateral Plate Element Connection
CRAC2D	Two-Dimensional Crack Tip Element
CRAC3D	Three-Dimensional Crack Tip Element
CROD	Rod Element Connection
CSHEAR	Shear Panel Element Connection
CTETRA	Four-Sided Solid Element Connection
CTRIA3	Triangular Plate Element Connection
CTRIA6	Curved Triangular Shell Element Connection
CTRIAR	Triangular Plate Element Connection
CTRIAX	Fully Nonlinear Axisymmetric Element
CTUBE	Tube Element Connection
CVISC	Viscous Damper Connection
CWELD	Weld or Fastener Element Connection
CYAX	Grid Points on Axis Symmetry
CYJOIN	Cyclic Symmetry Boundary Points
CYSUP	Fictitious Supports for Cyclic Symmetry
CYSYM	Cyclic Symmetry Parameters
DEFORM	Static Element Deformation
DMI	Direct Matrix Input
DMIAx	Direct Matrix Input for Axisymmetric Analysis
DMIG	Direct Matrix Input at Points
DMIG,UACCEL	Direct Matrix Input of Enforced Static Acceleration
DTI	Direct Table Input
DTI,INDTA	Stress and/or Force Sort Parameters
ECHOOFF	Deactivate Printed Echo
ECHOON	Activate Printed Echo
EIGB	Buckling Analysis Set
EIGRL	Real Eigenvalue Extraction Data, Lanczos Method
FORCE	Static Force

FORCE1	Static Force, Alternate Form 1
FORCE2	Static Force, Alternate Form 2
FORCEAX	Conical Shell Static Force
GENEL	General Element
GRAV	Acceleration or Gravity Load
GRDSET	Default Options for Fields 3,7,8, and 9 of all GRID Entries
GRIDB	Axisymmetric Grid Point
GRID	Grid Point
INCLUDE	Insert External File
ITER	Options for Iterative Solver
LOAD	Static Load Combination (Superposition)
LOADCYH	Harmonic Load Input for Cyclic Symmetry
LOADCYN	Physical Load Input for Cyclic Symmetry
LOADCYT	Table Load input for Cyclic Symmetry
MAT1	Isotropic Material Property Definition
MAT2	Shell Element Anisotropic Material Property Definition
MAT3	CTRIAX6 Material Property Definition
MAT8	Shell Element Orthotropic Material Property Definition
MAT9	Solid Element Anisotropic Material Property Definition
MATT1	Isotropic Material Temperature Dependence
MATT2	Anisotropic Material Temperature Dependence
MATT3	CTRIAX6 Material Temperature Dependence
MATT9	Solid Element Anisotropic Material Temperature Dependence
MOMAX	Conical Shell Static Moment
MOMENT	Static Moment
MOMENT1	Static Moment, Alternate Form 1
MOMENT2	Static Moment, Alternate Form 2
MPC	Multipoint Constraint
MPCADD	Multipoint Constraint Set Combination
MPCAX	Conical Shell Multipoint Constraint
OMIT	Omitted Degrees of Freedom
OMIT1	Omitted Degrees of Freedom, Alternate Form 1
OMITAX	Omitted Conical Shell Degrees of Freedom

PARAM	Parameter
PBAR	Simple Beam Property
PBARL	Cross-Sectional Beam Element (CBAR Entry) Dimensions
PBCOMP	Beam Property (Alternate Form of PBEAM)
PBEAM	Beam Property
PBEAML	Cross-Sectional Beam Element Dimensions
PBEND	Curved Beam or Pipe Element Property
PBUSH	Nominal Property Values for a Generalized Spring and Damper Structural Element
PCOMP	Layered Composite Element Property
PCONEAX	Conical Shell Element Property
PDUMi	Dummy Element Property
PELAS	Scalar Elastic Property
PLOAD	Static Pressure Load
PLOAD1	Applied Loads on CBAR, CBEAM or CBEND Elements
PLOAD2	Uniform Normal Pressure Load on a Surface Element
PLOAD4	Pressure Loads on Face of Structural Elements
PLOADX1	Pressure Load on Axisymmetric Element
PLOTEL	Dummy Plot Element Definition
PMASS	Scalar Mass Property
POINTAX	Conical Shell Point
PRAC2D	CRAC2D Element Property
PRAC3D	CRAC3D Element Property
PRESAX	Conical Shell Pressure Load
PRESPT	Fluid Pressure Point
PROD	Rod Property
PSHEAR	Shear Panel Property
PSHELL	Shell Element Property
PSOLID	Properties of Solid Elements
PTUBE	Tube Property
PWELD	Weld Element Property
RBAR	Rigid Bar
RBE1	Rigid Body Element, Form 1

RBE2	Rigid Body Element, Form 2
RBE3	Interpolation Constraint Element
RFORCE	Rotational Force
RINGAX	Conical Shell Ring
RROD	Rigid Pin-Ended Element Connection
RSPLINE	Interpolation Constraint Element
RSSCON	Shell-to-Solid Element Connector
RTRPLT	Rigid Triangular Plate
SNORM	Surface Normal Vector at a Grid Point
SPC	Single-Point Constraint
SPC1	Single-Point Constraint, Alternate Form
SPCADD	Single-Point Constraint Set Combination
SPCAX	Conical Shell Single Point Constraint
SPCD	Enforced Motion Value
SPCOFF	Excludes Degrees of Freedom from the AUTOSPC Operation
SPCOFF1	Excludes DOF's from AUTOSPC Processing, Alternate Form
SPOINT	Scalar Point Definition
SUPAX	Conical Shell Fictitious Support
SUPPORT	Fictitious Support
SUPPORT1	Fictitious Support, Alternate Form
TABLEM1	Material Property Table, Form 1
TABLEM2	Material Property Table, Form 2
TABLEM3	Material Property Table, Form 3
TABLEM4	Material Property Table, Form 4
TEMP	Grid Point Temperature Field
TEMPAX	Conical Shell Temperature
TEMPD	Grid Point Temperature Field Default
TEMPP1	Plate Element Temperature Field, Form 1
TEMPRB	One-Dimensional Element Temperature Field

Full descriptions of these and other Bulk Data entries can be found in the “**Bulk Data Entries**” on page 849 of the *MSC.Nastran Quick Reference Guide*.

APPENDIX

E

Output Listing

■ Examples List

5.1 Examples List

Examples of most of the printed output formats that are used in linear static analysis are contained in this section. Each example is annotated with comments and identified by page number and title. A summary of these figures is shown in [Listing 5-1](#).

Listing 5-1 List of Figures for Output Formats

Figure 5-1	Forces in Scalar Springs
Figure 5-2	Stresses in Scalar Springs
Figure 5-3	Forces in CROD Elements
Figure 5-4	Stresses in CROD Elements
Figure 5-5	Forces in CBAR Elements
Figure 5-6	Stresses in CBAR Elements
Figure 5-7	Force Distribution in CBAR Elements (CBARAO)
Figure 5-8	Stress Distribution in CBAR Elements (CBARAO)
Figure 5-9	Forces in CBEAM Elements
Figure 5-10	Stresses in CBEAM Elements
Figure 5-11	Forces in CBEND Elements
Figure 5-12	Stresses in CBEND Elements
Figure 5-13	Forces in CBUSH Element
Figure 5-14	Forces Acting on Shear Panel Elements
Figure 5-15	Stresses in Shear Panels
Figure 5-16	Forces in CQUAD4 Elements (Without Corner Option)
Figure 5-17	Stresses in CQUAD4 Elements (Without Corner Option)
Figure 5-18	Strains in CQUAD4 Elements (Without Corner Option)
Figure 5-19	Forces in CQUAD4 Elements (With Corner Option)
Figure 5-20	Stresses in CQUAD4 Elements (With Corner Option)
Figure 5-21	Strains in CQUAD4 Elements (With Corner Option)
Figure 5-22	Forces in CQUAD8 Elements
Figure 5-23	Stresses in CQUAD8 Elements
Figure 5-24	Strains in CQUAD8 Elements
Figure 5-25	Stresses in CHEXA Elements
Figure 5-26	Strains in CHEXA Elements
Figure 5-27	Forces in CWELD Elements
Figure 5-28	Grid Point Force Balance

Figure 5-29	Grid Point Stress Output
Figure 5-30	Grid Point Stress Discontinuities Output
Figure 5-31	Element Stress Discontinuities Output
Figure 5-32	Element Strain Energy
Figure 5-33	Sequence Processor Output
Figure 5-34	Grid Point Singularity Table
Figure 5-35	User Information Message 5293
Figure 5-36	OLOAD Resultant
Figure 5-37	SPCFORCE Resultant
Figure 5-38	Maximum SPCFORCES, DISPLACEMENT, and Applied Loads
Figure 5-39	Displacement Vectors
Figure 5-40	Load Vectors
Figure 5-41	Database Summary Table
Figure 5-42	Restartable Versions

F O R C E S I N S C A L A R S P R I N G S (C E L A S 2)							
ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE
1	-6.366724E+02	2	3.633277E+02	3	-5.537512E+02	4	3.653757E+02

Note: Standard FORCE output, requested in Case Control Section.

See “**Scalar Elements**” on page 69 for the interpretation of the CELASi output.

This output is typical of all scalar elements (CELAS1, CELAS2, CELAS3, CELAS4).

Figure 5-1 Forces in Scalar Springs

STRESSES IN SCALAR SPRINGS (CELAS2)

ELEMENT ID.	STRESS	ELEMENT ID.	STRESS	ELEMENT ID.	STRESS	ELEMENT ID.	STRESS
1	-6.366723E+04	2	3.633277E+04	3	-5.537512E+04	4	3.653757E+04

Note: Standard STRESS output, requested in Case Control Section

Stress output is obtained by multiplying the force in the spring by the stress coefficient entered in field 9 of the CELAS2 entry or field 9 of the PELAS entry.

Figure 5-2 Stresses in Scalar Springs

FORCES IN ROD ELEMENTS (CROD)

ELEMENT ID.	AXIAL FORCE	TORQUE	ELEMENT ID.	AXIAL FORCE	TORQUE
1	2.495156E+03	0	2	2.554449E+03	.0
3	-2.544570E+03	.0	4	-2.504844E+03	.0
5	4.990313E+02	.0			

Note: See “**One-Dimensional Elements**” on page 73 for definition of positive torque (same as CBAR).

Positive axial force is tension.

Figure 5-3 Forces in CROD Elements

STRESSES IN ROD ELEMENTS (CROD)									
ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN
1	6.237891E+02		.0		2	6.386123E+02		.0	
3	-6.361426E+02		.0		4	-6.262109E+02		.0	
5	1.247578E+02		.0						

Axial stress and margin-of-safety due to axial stress only.

Torsional stress and margin-of-safety due to torsion stress only.

Note: Positive stress is tension.

Standard STRESS output, requested in Case Control Section.

Definition of Margin-of-Safety (M.S.):

The M.S. for axial stress is: $M.S. = (ST/SA) - 1.0$ if SA is greater than zero or $M.S. = (-SC/SA) - 1.0$ if SA is less than zero, where ST and SC are the tension and compression allowables given on the MAT1 entry and SA is the axial stress given in the output. M.S. is computed and printed if and only if ST (or SC) is greater than zero and SA is not equal to zero.

The M.S. for torsional stress is: $M.S. = (SS/TAU) - 1.0$, where SS is the shear allowable given on the MAT1 entry and TAU is the torsional stress given in the output. M.S. is computed and printed if and only if SS is greater than zero and TAU is not equal to zero.

Figure 5-4 Stresses in CROD Elements

FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT		END-A	BEND-MOMENT		END-B	- SHEAR -		AXIAL
	PLANE 1	PLANE 2		PLANE 1	PLANE 2		PLANE 1	PLANE 2	FORCE
1	2.586560E+01	-1.519694E+01		1.619619E+00	4.542250E+00		2.168636E-02	-1.765535E-02	3.726698E+03
2	2.586560E+01	1.519694E+01		1.619619E+00	-4.542250E+00		2.168636E-02	1.765535E-02	3.726698E+03
3	-5.172854E+01	.0		-3.240443E+00	.0		-4.336907E-02	.0	-7.453428E+03
									.0

Note: See [Figure 4-9](#) and [Figure 4-10](#) for definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force and torque.

Standard FORCE output, requested in the Case Control Section.

Figure 5-5 Forces in CBAR Elements

STRESSES IN BAR ELEMENTS (CBAR)

ELEMENT ID.	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	8.434532E-01 -2.004801E+00	1.054002E+01 -1.397634E+00	-1.054002E+01 1.397634E+00	-8.434532E-01 2.004801E+00	4.658372E+02	4.763772E+02 4.678420E+02	4.552972E+02 4.638324E+02	-3.2E-02 -2.8E-03
2	-1.054002E+01 1.397634E+00	-8.434532E-01 2.004801E+00	8.434532E-01 -2.004801E+00	1.054002E+01 -1.397634E+00	4.658372E+02	4.763772E+02 4.678420E+02	4.552972E+02 4.638324E+02	
3	9.696071E+00 6.073933E-01	-9.696071E+00 -6.073933E-01	9.696071E+00 6.073933E-01	-9.696071E+00 -6.073933E-01	-9.316785E+02	-9.219824E+02 -9.310710E+02	-9.413745E+02 -9.322859E+02	6.1E-12 6.0E-12

Normal stresses due to bending only at ends A and B, points C, D, E, and F.

Example: SA1 = stress at end A, point C.
SB4 = stress at end B, point F.

Normal stresses due to axial loads only.

Normal stresses due to combined bending and axial loads.
Maximum and minimum values over points C, D, E, and F at ends A and B.

Margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

Note: Positive stress is tension.

Standard STRESS output, requested in Case Control Section.

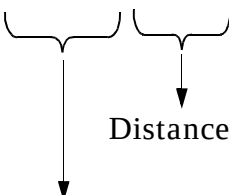
Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output, $S-MAX = \max (SA-MAX, SB-MAX)$. M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 card and S-MIN is the minimum stress given in the output, $S-MIN = \min (SA-MIN, SB-MIN)$. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Figure 5-6 Stresses in CBAR Elements

FORCE DISTRIBUTION IN BAR ELEMENTS (CBAR)								
ELEMENT ID.	STATION (PCT)	BEND-MOMENT		SHEAR FORCE		AXIAL FORCE	TORQUE	
		PLANE 1	PLANE 2	PLANE 1	PLANE 2			
11	.000	.0	1.868856E+06	.0	-7.851453E+03	.0	.0	
11	.200	.0	3.439147E+06	.0	-7.851453E+03	.0	.0	
11	.400	.0	4.887104E+06	.0	-5.331454E+03	.0	.0	
11	.600	.0	5.346728E+06	.0	1.028547E+03	.0	.0	
11	.800	.0	4.358352E+06	.0	9.148546E+03	.0	.0	
11	1.000	.0	2.528644E+06	.0	9.148546E+03	.0	.0	



 Distance from end A as a fraction of the elements length.

 Element ID

Note: Optional output for the CBAR element using the CBARAO entry.

See “**One-Dimensional Elements**” on page 73 for a discussion of plane 1 and 2.

Force output must be requested in Case Control Section.

Figure 5-7 Force Distribution in CBAR Elements (CBARAO)

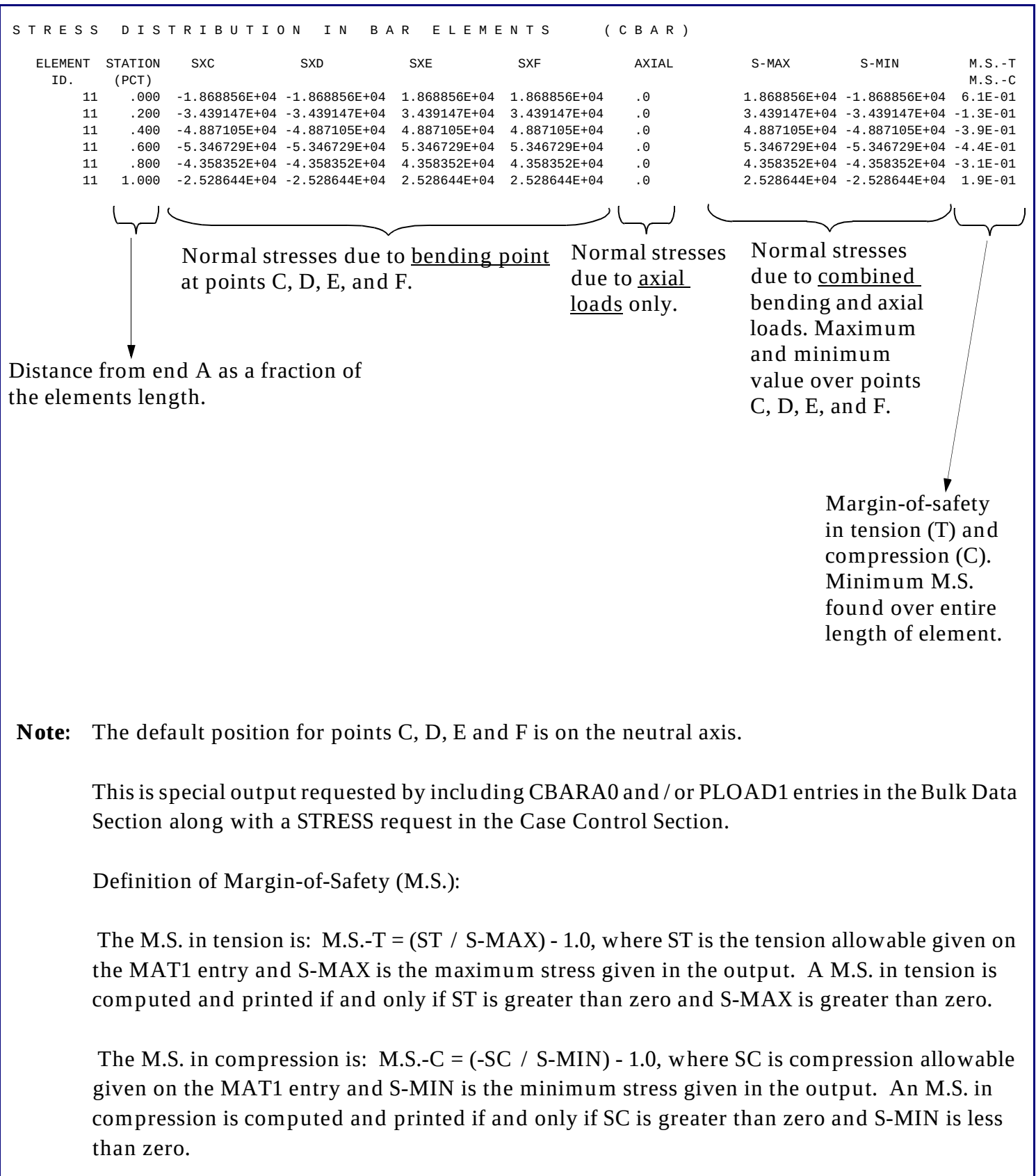


Figure 5-8 Stress Distribution in CBAR Elements (CBARAO)

FORCES IN BEAM ELEMENTS (CBEAM)									
ELEMENT-ID	GRID	STAT DIST/ LENGTH	- BENDING MOMENTS -		- WEB SHEARS -		AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
			PLANE 1	PLANE 2	PLANE 1	PLANE 2			
1									
	1	.000	8.653494E+03	-9.295851E+03	3.013959E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.200	6.791337E+03	-7.331033E+03	2.765989E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.400	4.929180E+03	-5.366216E+03	2.518019E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.600	3.067023E+03	-3.401398E+03	2.270049E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.800	1.204866E+03	-1.436580E+03	2.022079E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	2	1.000	-6.572902E+02	5.282371E+02	1.774109E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17

Distance from end A as a fraction of the element's length.

Grid points at ends A and B.

Part of total torque due to cross-sectional warping.

Element ID

Note: See “**One-Dimensional Elements**” on page 73 for definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force, and torque.

Standard FORCE output, requested in Case Control Section.

Figure 5-9 Forces in CBEAM Elements

STRESSES IN BEAM ELEMENTS (CBEAM)										
ELEMENT-ID	GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
1										
	1	.000	-6.118605E+02	1.554121E+03	6.269609E+02	-1.539021E+03	1.554121E+03	-1.539021E+03		
	0	.200	-6.634866E+02	1.679736E+03	6.816084E+02	-1.661615E+03	1.679736E+03	-1.661615E+03		
	0	.400	-6.934662E+02	1.748845E+03	7.173659E+02	-1.724945E+03	1.748845E+03	-1.724945E+03	-4.3E-01	-4.2E-01
	0	.600	-6.610018E+02	1.649506E+03	6.914890E+02	-1.619019E+03	1.649506E+03	-1.619019E+03		
	0	.800	-4.553899E+02	1.094180E+03	4.965609E+02	-1.053009E+03	1.094180E+03	-1.053009E+03		
	2	1.000	2.408044E+02	-7.437941E+02	-1.804025E+02	8.041960E+02	8.041960E+02	-7.437941E+02		

Element ID

Distance from end A as a fraction of the element's length.

Normal stresses due to combined bending and axial loads at points C, D, E, and F.

Maximum and minimum value over points C, D, E, and F.

Minimum margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

Note: This is standard STRESS output requested in Case Control Section.

Positive stress is tension.

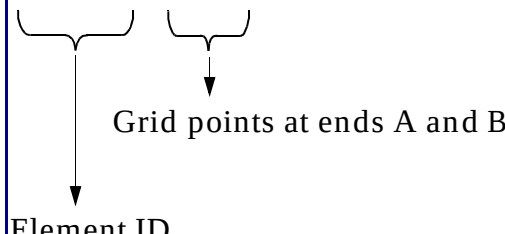
Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output. M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 entry and S-MIN is the minimum stress given in the output. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Figure 5-10 Stresses in CBEAM Elements

FORCES IN BEND ELEMENTS (CBEND)									
ELEMENT-ID	GRID	END	- BENDING MOMENTS -		- SHEARS -		AXIAL		TORQUE
			PLANE 1	PLANE 2	PLANE 1	PLANE 2	FORCE		
991	1	A	-9.507964E+02	-1.118982E+02	-7.252229E+01	4.371233E+00	6.426835E+01		-3.338711E+02
	2	B	-6.589756E+02	-4.884176E+02	6.426834E+01	4.371233E+00	7.252229E+01		2.664446E+02



Note: See “**One-Dimensional Elements**” on page 73 for definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force, and torque.

Standard FORCE output, requested in Case Control Section.

Figure 5-11 Forces in CBEND Elements

STRESSES IN BEND ELEMENTS (CBEND)											
ELEMENT-ID	GRID	END	CIRC. ANG.	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
991	1	A	0	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01		
	2	B	0	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	3.8E-01	

Normal stresses due to combined bending and axial loads at points C, D, E, and F.

Maximum and minimum value over points C, D, E, and F.

Minimum margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

Note: This is standard STRESS output requested in Case Control Section.

Positive stress is tension.

Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output. M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 entry and S-MIN is the minimum stress given in the output. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Figure 5-12 Stresses in CBEND Elements

FORCES IN BUSH ELEMENTS (CBUSH)							
	ELEMENT-ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
0	10000	-6.489702E+00	-5.897736E+00	6.265953E-01	3.955022E+00	-9.723748E-02	1.147097E-02
0	10001	-3.201495E+00	-1.851144E+00	2.724363E-01	8.105669E+00	-1.866592E-01	2.313610E-02
0	10002	9.125268E-01	2.533982E+00	2.790101E-01	8.008885E+00	-1.804384E-01	2.278207E-02
0	10003	2.938818E+00	5.093895E+00	3.161362E-01	7.944401E+00	-1.380170E-01	2.264773E-02
0	10004	3.530923E+00	7.028401E+00	3.253268E-01	7.911948E+00	-8.659782E-02	2.265486E-02
0	10005	6.848238E+00	4.654799E+00	3.589581E-01	7.842767E+00	-2.292452E-02	2.255161E-02
0	10006	6.788301E+00	3.347241E+00	3.445033E-01	7.912354E+00	4.136241E-02	2.287587E-02
0	10007	4.012078E+00	2.324608E+00	3.328075E-01	7.965162E+00	1.027839E-01	2.302680E-02
0	10008	5.789543E-01	-4.556357E-01	3.059701E-01	8.035738E+00	1.431401E-01	2.305412E-02
0	10009	-4.496515E+00	-5.173875E+00	2.923176E-01	8.129048E+00	1.063870E-01	2.287990E-02
0	10010	-7.422128E+00	-7.604536E+00	5.459386E-01	3.958032E+00	-3.207400E-02	1.047710E-02

Note: The forces are in the CBUSH element coordinate system.

Figure 5-13 Forces in CBUSH Element

FORCES ACTING ON SHEAR PANEL ELEMENTS (CSHEAR)									
ELEMENT ID	===== POINT 1 =====			===== POINT 2 =====			===== POINT 3 =====		
	F-FROM-4	F-FROM-2	F-FROM-1	F-FROM-3	F-FROM-2	F-FROM-4	F-FROM-3	F-FROM-1	
	KICK-1	SHEAR-12	KICK-2	SHEAR-23	KICK-3	SHEAR-34	KICK-4	SHEAR-41	
11	-1.49620E+00	-1.49620E+00	1.49620E+00	1.49620E+00	-1.49620E+00	-1.49620E+00	1.49620E+00	1.49620E+00	
	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01	
12	-1.28268E+00	-1.28268E+00	1.28268E+00	1.28268E+00	-1.28268E+00	-1.28268E+00	1.28268E+00	1.28268E+00	
	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01	

Note: See “**Two-Dimensional Elements**” on page 127 for definitions of element coordinate system and force components.

Standard FORCE output, requested in Case Control Section.

Figure 5-14 Forces Acting on Shear Panel Elements

STRESSES IN SHEAR PANELS (CSHEAR)							
ELEMENT ID.	MAX SHEAR	AVG SHEAR	SAFETY MARGIN	ELEMENT ID.	MAX SHEAR	AVG SHEAR	SAFETY MARGIN
11	2.992391E+00	-2.992391E+00	1.5E+00	12	2.565355E+00	-2.565355E+00	

Maximum and average shear stresses and margin-of-safety due to shear only.

Note: Standard STRESS output, requested in Case Control Section.

Definition of Margin-of-Safety (M.S.):

The M.S. is: $M.S. = (SS / MAX\ SHEAR) - 1.0$, where SS is the shear allowable given on the MAT1 entry and MAX SHEAR is the maximum shear stress given in the output. M.S. is computed and printed if and only if SS is greater than zero and MAX SHEAR is not equal to zero.

Figure 5-15 Stresses in Shear Panels

FORCES IN QUADRILATERAL ELEMENTS (QUAD4)								
ELEMENT ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -	
	FX	FY	FGY	MX	MY	MXY	QX	QY
1	-2.349886E+00	-3.348329E-01	-2.992391E-01	-1.250000E+01	-2.983451E+00	-5.573193E-01	-5.000000E-01	2.352497E-02
2	-1.593343E+00	2.805547E-02	-2.565355E-01	-7.500000E+00	-5.912674E-01	-1.125781E-01	-5.000000E-01	3.125528E-02
3	-8.648885E-01	2.233189E-02	-4.740243E-01	-2.500000E+00	6.004672E-01	3.312782E-01	-5.000000E-01	1.284359E-02
4	1.349886E+00	1.656714E-01	-4.015217E-01	-1.250000E+01	-2.983451E+00	5.573193E-01	-5.000000E-01	-2.352497E-02
5	5.933433E-01	-8.414105E-02	-4.869289E-01	-7.500000E+00	-5.912674E-01	1.125781E-01	-5.000000E-01	-3.125528E-02
6	-1.351115E-01	-4.106952E-03	-5.259757E-01	-2.500000E+00	6.004672E-01	-3.312782E-01	-5.000000E-01	-1.284359E-02

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

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

Transverse shear forces (force / length).

Note: Positive force is tension.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system and definition of positive shears and moments.

This output is typical for all shell elements.

Standard FORCE output, requested in Case Control Section.

Figure 5-16 Forces in CQUAD4 Elements (Without Corner Option)

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	
1	-5.000000E-02	-7.523499E+03	-1.793419E+03	-3.373840E+02	-86.6419	-1.773622E+03	-7.543295E+03	6.831404E+03
	5.000000E-02	7.476501E+03	1.786722E+03	3.313992E+02	3.3222	7.495738E+03	1.767485E+03	6.786861E+03
2	-5.000000E-02	-4.515933E+03	-3.544799E+02	-7.011219E+01	-89.0350	-3.532990E+02	-4.517114E+03	4.351235E+03
	5.000000E-02	4.484066E+03	3.550410E+02	6.498148E+01	.9014	4.485089E+03	3.540186E+02	4.318975E+03
3	-5.000000E-02	-1.508649E+03	3.605036E+02	1.940267E+02	84.1357	3.804320E+02	-1.528577E+03	1.750085E+03
	5.000000E-02	1.491351E+03	-3.600570E+02	-2.035071E+02	-6.1993	1.513457E+03	-3.821626E+02	1.736371E+03

Stresses in element coordinate system.

Angle of principal stress in element coordinate system.

Major and minor principal stresses.

Maximum shear stress or Hencky-von Mises stress.

Fiber distance Z1 and Z2 specified on the PSHELL property entry

Note: This is standard STRESS output requested in Case Control Section.

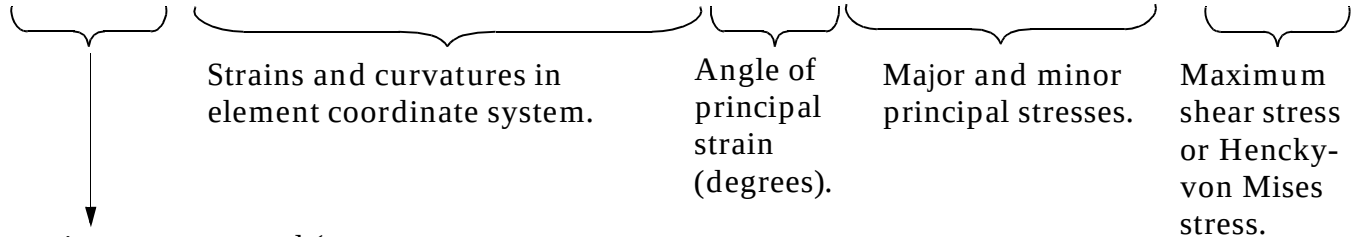
This output is typical for all plate elements.

See “[Two-Dimensional Elements](#)” on page 127 for definition of element coordinate system.

Figure 5-17 Stresses in CQUAD4 Elements (Without Corner Option)

STRAINS IN QUADRILATERAL ELEMENTS (QUAD4)

ELEMENT ID.	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)		VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR MINOR	
1	.0	-2.249436E-06	3.701331E-07	-7.780218E-07	-81.7292	4.266812E-07 -2.305984E-06	1.697520E-06
	-1.000000E+00	-1.392596E-02	9.198587E-04	-1.738836E-03	-86.6598	9.706012E-04 -1.397670E-02	9.657605E-03
2	.0	-1.601760E-06	5.060585E-07	-6.669924E-07	-81.2203	5.575652E-07 -1.653267E-06	1.327650E-06
	-1.000000E+00	-8.787143E-03	1.990479E-03	-3.512435E-04	-89.0667	1.993340E-03 -8.790004E-03	6.625173E-03
3	.0	-8.715881E-07	2.817985E-07	-1.232463E-06	-66.5508	5.490937E-07 -1.138883E-06	9.941859E-07
	-1.000000E+00	-3.216168E-03	1.620561E-03	1.033588E-03	83.9688	1.675163E-03 -3.270770E-03	2.904646E-03



If fiber strains are requested (see Note 2) the fiber distances Z1 and Z2 will appear here. If strain-curvature data is requested, this entry will appear as shown.

Note: Strain output is available for plate elements and is requested using the Case Control command STRAIN.

Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.

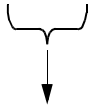
This output is typical for all plate elements.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.

Figure 5-18 Strains in CQUAD4 Elements (Without Corner Option)

FORCES IN QUADRILATERAL ELEMENTS (QUAD4)											OPTION = BILIN
ELEMENT ID	GRID-ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -			
		FX	FY	FX	MY	MX	MY	QX	QY		
1	CEN/4	-2.349886E+00	-3.198563E-01	-5.233257E-01	-1.250000E+01	-2.983451E+00	-5.573193E-01	-5.000000E-01	2.352497E-02		
	1	-4.448889E+00	-1.334667E+00	3.150085E-01	-1.434321E+01	-4.302963E+00	-6.366503E-01	-5.000000E-01	2.352497E-02		
	2	-4.194971E+00	-4.882719E-01	-1.065422E+00	-9.086258E+00	-9.787024E-01	-1.686622E+00	-5.000000E-01	2.352497E-02		
	6	-2.508836E-01	6.949542E-01	-1.361660E+00	-1.040833E+01	-1.803477E+00	-5.073156E-01	-5.000000E-01	2.352497E-02		
	5	-5.048020E-01	-1.514406E-01	1.877032E-02	-1.616221E+01	-4.848663E+00	5.426558E-01	-5.000000E-01	2.352497E-02		
2	CEN/4	-1.593343E+00	6.351992E-02	-4.918268E-01	-7.500000E+00	-5.912674E-01	-1.125781E-01	-5.000000E-01	3.125528E-02		
	2	-2.714164E+00	-4.402977E-02	-1.611118E-01	-1.134483E+01	-1.656274E+00	-1.138151E+00	-5.000000E-01	3.125528E-02		
	3	-2.864952E+00	-5.466592E-01	-9.984620E-01	-6.194139E+00	-3.528372E-03	7.047635E-01	-5.000000E-01	3.125528E-02		
	7	-4.725232E-01	1.710696E-01	-8.225417E-01	-3.723025E+00	5.971142E-01	8.980479E-01	-5.000000E-01	3.125528E-02		
	6	-3.217343E-01	6.736990E-01	1.480843E-02	-8.738006E+00	-1.302381E+00	-9.448670E-01	-5.000000E-01	3.125528E-02		



Denotes CEN / 4 for the center stress or the grid point ID for a corner stress.

Note: Positive force is tension.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system and definition of positive shears and moments.

This output is typical for all shell elements.

Standard FORCE output, requested in Case Control Section.

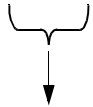
Force output obtained when using the FORCE(CORNER) in Case Control Section.

The transverse shear forces (QX and QY) are computed at the center only, and those same values are displayed at the corners.

Figure 5-19 Forces in CQUAD4 Elements (With Corner Option)

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON MISES
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	
1	CEN/4	-5.000000E-02	-7.523499E+03	-1.793269E+03	-3.396249E+02	-86.6199	-1.773210E+03	-7.543558E+03	6.831780E+03
		5.000000E-02	7.476501E+03	1.786872E+03	3.291584E+02	3.3000	7.495480E+03	1.767893E+03	6.786491E+03
1	1	-5.000000E-02	-8.650413E+03	-2.595124E+03	-3.788401E+02	-86.4339	-2.571515E+03	-8.674022E+03	7.716604E+03
		5.000000E-02	8.561436E+03	2.568431E+03	3.851403E+02	3.6620	8.586085E+03	2.543781E+03	7.638754E+03
2	2	-5.000000E-02	-5.493704E+03	-5.921041E+02	-1.022627E+03	-78.6755	-3.873087E+02	-5.698500E+03	5.515055E+03
		5.000000E-02	5.409805E+03	5.823387E+02	1.001319E+03	11.2654	5.609259E+03	3.828847E+02	5.427954E+03
6	6	-5.000000E-02	-6.247504E+03	-1.075137E+03	-3.180060E+02	-86.4949	-1.055658E+03	-6.266982E+03	5.811513E+03
		5.000000E-02	6.242486E+03	1.089036E+03	2.907728E+02	3.2192	6.258840E+03	1.072681E+03	5.797412E+03
5	5	-5.000000E-02	-9.702373E+03	-2.910712E+03	3.257812E+02	87.2600	-2.895121E+03	-9.717964E+03	8.642099E+03
		5.000000E-02	9.692277E+03	2.907683E+03	-3.254057E+02	-2.7397	9.707849E+03	2.892112E+03	8.633103E+03



Denotes CEN/4 for the center stress or the grid point ID for a corner stress.

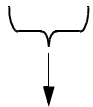
Note: This is standard STRESS output requested in Case Control Section.

Stress output obtained using the STRESS(CORNER) command in the Case Control Section.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system.

Figure 5-20 Stresses in CQUAD4 Elements (With Corner Option)

S T R A I N S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)										OPTION = BILIN
ELEMENT ID	GRID-ID	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)				VON MISES
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
1	CEN/4	.0 -1.000000E+00	-2.253929E-06 -1.392596E-02	3.851096E-07 9.198587E-04	-1.360647E-06 -1.738836E-03	-76.3625 -86.6598	5.501681E-07 9.706012E-04	-2.418988E-06 -1.397670E-02	1.823920E-06 9.657605E-03	
	1	.0 -1.000000E+00	-4.048489E-06 -1.566278E-02	.0 .0	8.190221E-07 -1.986349E-03	84.2816 -86.3862	4.100733E-08 6.272583E-05	-4.089496E-06 -1.572551E-02	2.740102E-06 1.050464E-02	
	2	.0 -1.000000E+00	-4.048489E-06 -1.055118E-02	7.702193E-07 2.096610E-03	-2.770097E-06 -5.262259E-03	-75.0535 -78.7048	1.139956E-06 2.622131E-03	-4.418226E-06 -1.107670E-02	3.389972E-06 8.396119E-03	
	6	.0 -1.000000E+00	-4.593699E-07 -1.184074E-02	7.702193E-07 1.582825E-03	-3.540316E-06 -1.582825E-03	-54.5763 -86.6375	2.029306E-06 1.629323E-03	-1.718457E-06 -1.188724E-02	2.166252E-06 8.520021E-03	
	5	.0 -1.000000E+00	-4.593699E-07 -1.764913E-02	.0 .0	4.880284E-08 1.693086E-03	86.9679 87.2602	1.292550E-09 4.051156E-05	-4.606624E-07 -1.768964E-02	3.075400E-07 1.180662E-02	



Denotes CEN / 4 for the center strain and curvatures and a grid point ID for a corner strain or curvature.

Note: Corner strain output is available for the plate elements and is requested using the Case Control command STRAIN(CORNER).

Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.

This output is typical for all plate elements.

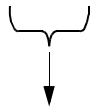
See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.

Figure 5-21 Strains in CQUAD4 Elements (With Corner Option)

FORCES IN QUADRILATERAL ELEMENTS (QUAD8)

ELEMENT		- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -	
ID	GRID-ID	FX	FY	FXY	MX	MY	MXY	QX	QY
1	CEN/4	-2.197888E+00	-3.254840E-01	-5.232677E-01	-1.250000E+01	-3.691373E+00	-3.563170E-02	-5.205615E+00	8.254652E-02
	1	-4.108907E+00	-1.232672E+00	-1.282016E+00	-1.483570E+01	-4.450711E+00	-5.782321E-02	-5.205615E+00	8.254652E-02
	2	-3.888765E+00	-4.988650E-01	-2.135196E-02	-1.014609E+01	-2.877548E+00	-5.853834E-02	-5.205615E+00	8.254652E-02
	6	-2.868679E-01	5.817043E-01	2.354806E-01	-9.853907E+00	-2.887942E+00	-1.344019E-02	-5.205615E+00	8.254652E-02
	5	-5.070101E-01	-1.521030E-01	-1.025183E+00	-1.516430E+01	-4.549290E+00	-1.272506E-02	-5.205615E+00	8.254652E-02
2	CEN/4	-1.499658E+00	4.452617E-02	-4.931346E-01	-7.500000E+00	-2.112700E+00	-1.628812E-02	-1.779801E+00	2.273268E-02
	2	-2.536411E+00	-9.315875E-02	-8.093295E-01	-9.912405E+00	-2.807442E+00	-2.389918E-02	-1.779801E+00	2.273268E-02
	3	-2.650702E+00	-4.741282E-01	-4.360040E-02	-5.061935E+00	-1.334132E+00	-2.409612E-02	-1.779801E+00	2.273268E-02
	7	-4.629044E-01	1.822111E-01	-1.769397E-01	-4.938065E+00	-1.351179E+00	-8.677053E-03	-1.779801E+00	2.273268E-02
	6	-3.486136E-01	5.631805E-01	-9.426688E-01	-1.008759E+01	-2.958048E+00	-8.480112E-03	-1.779801E+00	2.273268E-02



Denotes CEN/4 for the center forces or the grid point ID for a corner force.

Note: Positive force is tension.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system and definition of positive shears and moments.

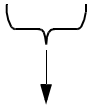
This output is typical for all shell elements.

Standard FORCE output, requested in Case Control Section.

Figure 5-22 Forces in CQUAD8 Elements

STRESSES IN QUADRILATERAL ELEMENTS (QUAD8)

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	-7.521979E+03	-2.218078E+03	-2.661170E+01	-89.7125	-2.217945E+03	-7.522112E+03	6.694611E+03
		5.000000E-02	7.478021E+03	2.211569E+03	1.614634E+01	.1757	7.478070E+03	2.211519E+03	6.653905E+03
	1	-5.000000E-02	-8.942510E+03	-2.682753E+03	-4.751408E+01	-89.5651	-2.682393E+03	-8.942870E+03	7.948703E+03
		5.000000E-02	8.860332E+03	2.658100E+03	2.187377E+01	.2021	8.860409E+03	2.658022E+03	7.875326E+03
	2	-5.000000E-02	-6.126543E+03	-1.731518E+03	-3.533652E+01	-89.5394	-1.731234E+03	-6.126827E+03	5.470668E+03
		5.000000E-02	6.048768E+03	1.721540E+03	3.490948E+01	.4622	6.049049E+03	1.721259E+03	5.398310E+03
	6	-5.000000E-02	-5.915213E+03	-1.726948E+03	-5.709310E+00	-89.9219	-1.726940E+03	-5.915221E+03	5.268484E+03
		5.000000E-02	5.909476E+03	1.738582E+03	1.041892E+01	.1431	5.909501E+03	1.738556E+03	5.260303E+03
	5	-5.000000E-02	-9.103648E+03	-2.731095E+03	-1.788687E+01	-89.8392	-2.731045E+03	-9.103699E+03	8.091559E+03
		5.000000E-02	9.093509E+03	2.728053E+03	-2.616797E+00	-.0236	9.093510E+03	2.728052E+03	8.082488E+03



Denotes CEN/4 for the center stress or a grid point ID for a corner stress.

Note: This is standard STRESS output requested in Case Control Section.

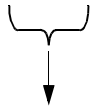
This output is typical for all plate elements.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system.

Figure 5-23 Stresses in CQUAD8 Elements

STRAINS IN QUADRILATERAL ELEMENTS (QUAD8)

ELEMENT ID	GRID-ID	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL ANGLE	STRAINS (ZERO SHEAR)			VON MISES
			NORMAL-X	NORMAL-Y	SHEAR-XY		MAJOR	MINOR		
1	CEN/4	.0	-2.100242E-06	3.338823E-07	-1.360496E-06	-75.3990	5.110861E-07	-2.277446E-06	1.714246E-06	
		-1.000000E+00	-1.367111E-02	7.035311E-05	-1.111709E-04	-89.7682	7.057795E-05	-1.367133E-02	9.137837E-03	
	1	.0	-3.739106E-06	5.151080E-22	-3.333241E-06	-69.1422	6.350135E-07	-4.374119E-06	3.149165E-06	
		-1.000000E+00	-1.620059E-02	2.948739E-20	-1.804084E-04	-89.6810	5.022378E-07	-1.620109E-02	1.080089E-02	
	2	.0	-3.739106E-06	6.677646E-07	-5.551511E-08	-89.6391	6.679394E-07	-3.739280E-06	2.742746E-06	
		-1.000000E+00	-1.113939E-02	1.995355E-04	-1.826396E-04	-89.5386	2.002709E-04	-1.114013E-02	7.494402E-03	
	6	.0	-4.613792E-07	6.677646E-07	6.122495E-07	75.7662	7.454184E-07	-5.390330E-07	7.447625E-07	
		-1.000000E+00	-1.078503E-02	8.187693E-05	-4.193340E-05	-89.8895	8.191738E-05	-1.078507E-02	7.217507E-03	
	5	.0	-4.613792E-07	-1.419557E-22	-2.665477E-06	-49.9101	1.121867E-06	-1.583246E-06	1.569352E-06	
		-1.000000E+00	-1.655941E-02	2.215808E-22	-3.970219E-05	-89.9313	2.379707E-08	-1.655944E-02	1.103963E-02	



Denotes CEN / 4 for the center strain and curvature and a grid point ID for a corner strain and curvature.

Note: Strain output is available for plate elements and is requested using the Case Control command STRAIN.

Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.

This output is typical for all plate elements.

See “**Two-Dimensional Elements**” on page 127 for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.

Figure 5-24 Strains in CQUAD8 Elements

Location of Stresses
 GRID = Stresses at center and vertex points
 GAUSS = Stresses at center and Gauss points

Stress coordinate system definition
 0 = Basic Coordinate System
 -1 = Element Coordinate System
 +X = Material Coordinate System
 Defined on PSOLID entry

Number of active grid points for the element

STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)													
ELEMENT-ID	CORNER GRID-ID	-----CENTER AND CORNER POINT STRESSES-----						DIR.	COSINES	MEAN			
		NORMAL		SHEAR		PRINCIPAL		-A-	-B-	-C-	PRESSURE	VON MISES	
1	1 GRID	CS		8 GP									
	CENTER	X	-1.559055E+01	XY	-4.613456E-01	A	1.642750E+01	LX	.01	1.00	.03	-6.818652E-02	2.776379E+01
		Y	-6.143418E-01	YZ	-4.613456E-01	B	-1.560962E+01	LY	-.03	.03	-1.00		
		Z	1.640945E+01	ZX	4.094507E-01	C	-6.133184E-01	LZ	1.00	-.01	-.03		
	51	X	7.173539E+01	XY	5.611941E+00	A	2.073488E+02	LX	-.59	.81	-.03	-8.944785E+01	1.848900E+02
		Y	6.159913E+01	YZ	-1.855275E+00	B	-6.360914E-01	LY	-.03	-.06	-1.00		
		Z	1.350090E+02	ZX	-9.888012E+01	C	6.163087E+01	LZ	.81	.59	-.06		
	55	X	6.940665E+01	XY	4.212266E+00	A	1.836086E+02	LX	-.61	.79	-.02	-8.151105E+01	1.614635E+02
		Y	5.992439E+01	YZ	-6.248875E-01	B	9.289488E-01	LY	-.02	-.05	-1.00		
		Z	1.152021E+02	ZX	-8.830753E+01	C	5.999560E+01	LZ	.79	.61	-.05		

Center and corner point stresses in the stress coordinate system.

Directional cosines of the principal stress with respect to the stress coordinate system.

Hencky-von Mises stress or octahedral shear stress.

Note: This output is typical for the CHEXA, CPENTA, and CTETRA elements.

Standard STRESS output, requested in Case Control Section.

The convention for the principal stresses are such that ""($A \geq C \geq B$). In the case where the principal stresses are equal, the directional cosines are not unique, and the values of zero are output for the directional cosines.

Figure 5-25 Stresses in CHEXA Elements

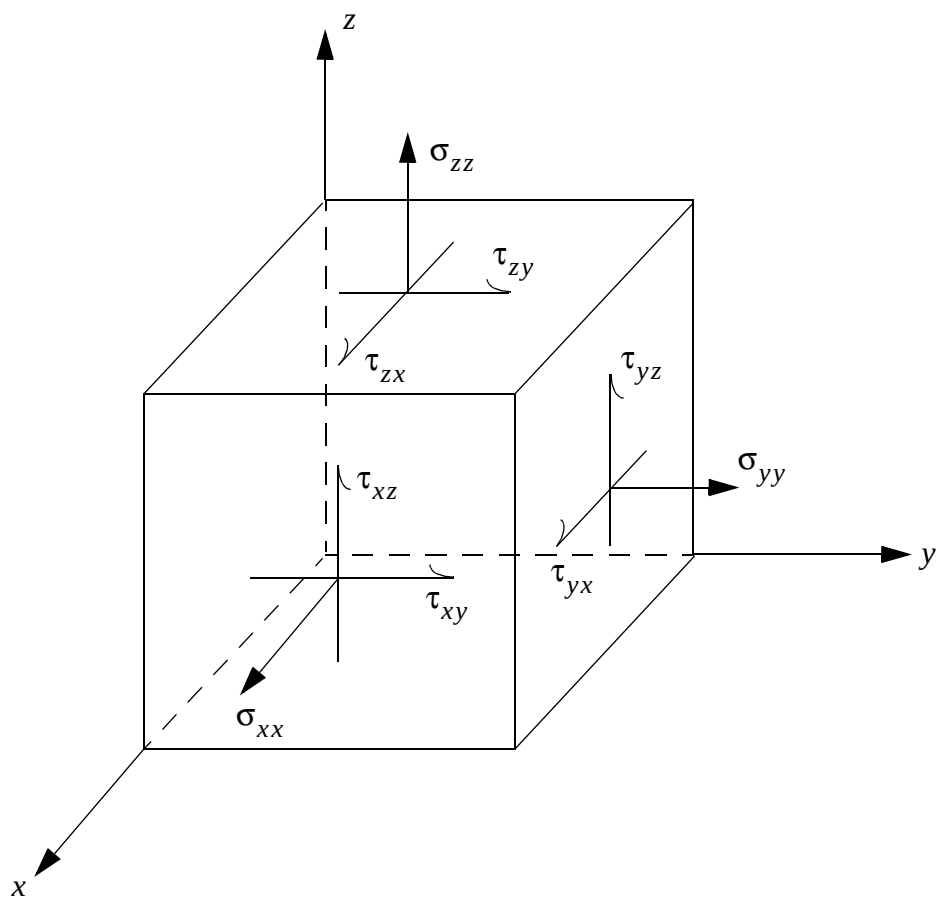
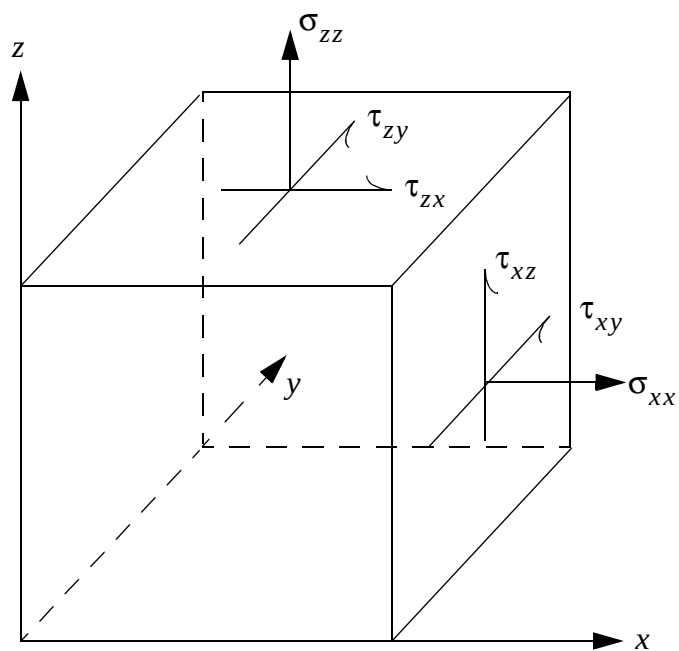


Figure E-25 Stresses in CHEXA Elements(continued)

```

      S T R A I N S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   (   H E X A   )

ELEMENT-ID  CORNER  -----CENTER AND CORNER POINT  STRAINS-----  DIR.  COSINES  MEAN  VON
1  GRID-ID  NORMAL  SHEAR  PRINCIPAL  -A- -B- -C-  PRESSURE  MISES

      1GRID CS  8 GP
CENTER X -3.315102E-06 XY 4.167708E-08 A 2.637197E-06 LX .01 1.00 -.01 1.582972E-07 3.456435E-06
      Y 2.051620E-07 YZ -9.459202E-08 B -3.316474E-06 LY -.02 -.01 -1.00
      Z 2.635048E-06 ZX 1.718858E-07 C 2.043855E-07 LZ 1.00 -.01 -.02
51 X 2.388123E-06 XY -2.219673E-06 A 1.648163E-05 LX -.57 .82 .01 -3.979110E-06 1.271542E-05
      Y -1.419658E-07 YZ 2.219673E-06 B -4.278821E-06 LY .09 .07 -.99
      Z 9.691174E-06 ZX -1.925896E-05 C -2.654761E-07 LZ .82 .56 .12
55 X 3.108307E-06 XY -2.219673E-06 A 1.451382E-05 LX -.61 .79 -.02 -3.651943E-06 1.105123E-05
      Y 1.419658E-07 YZ 2.219673E-06 B -3.543073E-06 LY .11 .06 -.99
      Z 7.705556E-06 ZX -1.727543E-05 C -1.491768E-08 LZ .79 .60 .12

```

Note: This output is typical for the CHEXA, CPENTA, and CTETRA elements.

Standard STRAIN output, requested in Case Control Section.

Figure 5-26 Strains in CHEXA Elements

```

1  WELDED SHELL MODEL, POINT-TO-POINT OPTION              JULY 25, 2001  MSC.NASTRAN  4/ 9/01  PAGE  21
0
                                SUBCASE 1

      F O R C E S   I N   W E L D   E L E M E N T S   (   C W E L D   )

ELEMENT  BEND-MOMENT END-A  BEND-MOMENT END-B  - SHEAR -  AXIAL
ID  PLANE 1 (MZ)  PLANE 2 (MY)  PLANE 1 (MZ)  PLANE 2 (MY)  PLANE 1 (FY)  PLANE 2 (FZ)  FORCE FX  TORQUE MX
109 -2.135744E+02  1.586358E+02  7.728133E+01  1.390482E+01  -2.908656E+02  -1.447359E+02  -8.345229E+01  0.0
110 2.135769E+02  1.586297E+02  -7.728750E+01  1.390158E+01  2.908643E+02  -1.447281E+02  -8.346558E+01  0.0
111 -2.363556E+02  -3.536731E+02  -2.396099E+02  -2.361487E+02  3.254667E+00  1.175355E+02  -1.198793E+01  0.0
112 2.363570E+02  -3.536719E+02  2.396101E+02  -2.361489E+02  -3.253387E+00  1.175340E+02  -1.198794E+01  0.0

```

Figure 5-27 Forces in CWELD Elements

SUBCASE 1

GRID POINT FORCE BALANCE

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	5.000000E+03	4.990313E+02	.0	.0	.0	.0
1	3	ROD	-2.495156E+03	-4.990313E+02	.0	.0	.0	.0
1	4	ROD	-2.504844E+03	.0	.0	.0	.0	.0
1		*TOTALS*	.0	.0	.0	.0	.0	.0
2		F-OF-SPC	-5.000000E+03	5.009687E+02	.0	.0	.0	.0
2	1	ROD	2.495156E+03	.0	.0	.0	.0	.0
2	2	ROD	2.504844E+03	-5.009687E+02	.0	.0	.0	.0
2		*TOTALS*	.0	.0	.0	.0	.0	.0
3	1	ROD	-2.495156E+03	.0	.0	.0	.0	.0
3	3	ROD	2.495156E+03	4.990313E+02	.0	.0	.0	.0
3	5	ROD	.0	-4.990313E+02	.0	.0	.0	.0
3		*TOTALS*	-4.547474E-13	-1.347189E-11	.0	.0	.0	.0

Forces and moments acting on the grid point from each source in the global coordinate system.

Note: Only the forces due to elements, SPCs and applied loads are considered. The totals will be nonzero due to round-off error, or to the fact that forces due to MPCs, rigid elements, GENELs, DMIG, or inertia loads (in dynamic analysis) are not included.

Standard GPFORCE output, requested in Case Control Section.

Figure 5-28 Grid Point Force Balance

STRESSES AT GRID POINTS - - SURFACE 91										
GRID ID	ELEMENT ID	STRESSES IN ELEMENT SYSTEM				REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0				
		SURFACE X-AXIS X NORMAL(Z-AXIS) Z				PRINCIPAL STRESSES				
		FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	SHEAR	VON MISES
20	0	Z1	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		Z2	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		MID	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
21	0	Z1	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		Z2	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		MID	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
60	0	Z1	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		Z2	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		MID	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
61	0	Z1	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		Z2	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		MID	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03

Note: Stress surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Stress volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

Figure 5-29 Grid Point Stress Output

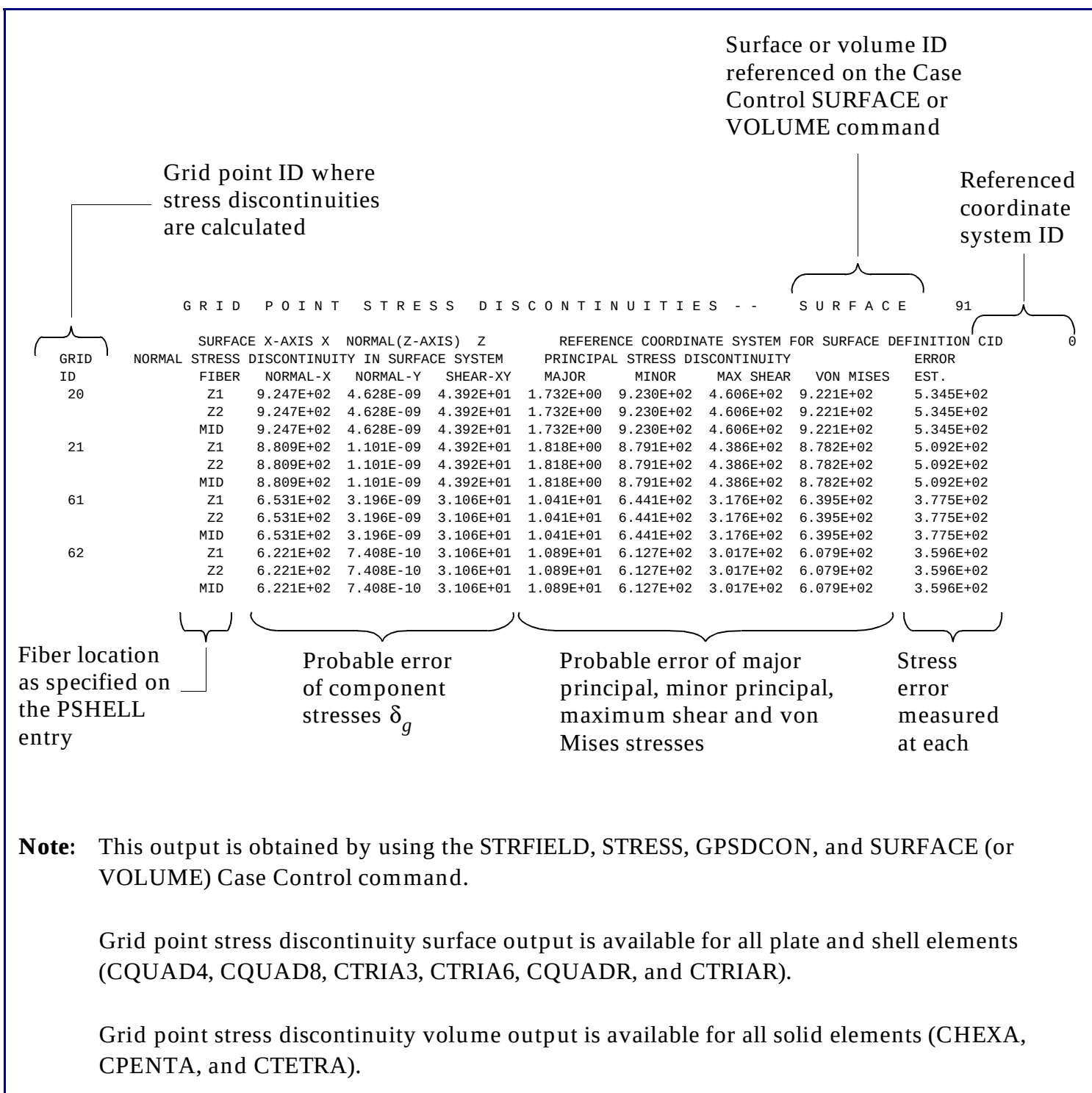


Figure 5-30 Grid Point Stress Discontinuities Output

Element ID where element stress discontinuities are calculated		Surface or volume ID referenced on the Case Control SURFACE or VOLUME command										Referenced coordinate system ID
		ELEMENT STRESS DISCONTINUITIES - - SURFACE 91										0
ELEMENT ID	ELEMENT TYPE	SURFACE FIBER	X-AXIS X NORMAL-X	Y NORMAL-Y	Z DISCONTINUITY IN SURFACE SHEAR-XY	REFERENCE COORDINATE SYSTEM MAJOR	PRINCIPAL STRESS MINOR	STRESS DISCONTINUITY MAX SHEAR	VON MISES	ERROR EST.	CID	
19	QUAD4	Z1	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02		
		Z2	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02		
		MID	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02		
20	QUAD4	Z1	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02		
		Z2	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02		
		MID	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02		
59	QUAD4	Z1	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02		
		Z2	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02		
		MID	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02		
60	QUAD4	Z1	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02		
		Z2	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02		
		MID	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02		

Fiber location as specified on the PSHELL entry

Probable error of component stresses δ_e

Probable error of major principal, minor principal, maximum shear and von Mises stresses

Stress error measured at each

Fiber location as specified on the PSHELL entry

Probable error of component stresses δ_e

Probable error of major principal, minor principal, maximum shear and von Mises stresses

Stress error measured at each

Note: This output is obtained by using the STRFIELD, STRESS, ELSDCON, and SURFACE (or VOLUME) Case Control commands.

Element stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Element stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

Figure 5-31 Element Stress Discontinuities Output

Total strain energy in requested SET (SET -1 denotes ALL)

SUBCASE 2

E L E M E N T S T R A I N E N E R G I E S

ELEMENT-TYPE = ROD		* TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM	=	7.578219E+00
SUBCASE	2	TOTAL ENERGY OF ALL ELEMENTS IN SET	-1 =	7.578219E+00

ELEMENT-ID	STRAIN-ENERGY	PERCENT OF TOTAL	STRAIN-ENERGY-DENSITY
1	5.742940E+00	75.7822	2.392892E-03
2	6.220422E-01	8.2083	2.541511E-04
3	6.220422E-01	8.2083	2.541511E-04
4	5.865024E-01	7.7393	2.443760E-04
5	4.692019E-03	.0619	9.775041E-06
TYPE = ROD	SUBTOTAL	7.578219E+00	100.0000

Total strain energy for all elements of this type in the requested set

Strain energy in each element

Note: This output is incorrect in the presence of thermal loads or element deformations.

Standard output, requested with Case Control command ESE.

Figure 5-32 Element Strain Energy

SEQUENCE PROCESSOR OUTPUT

THERE ARE 20 POINTS DIVIDED INTO 1 GROUP(S).

CONNECTION DATA

ELEMENT TYPE	NUMBER	ASSEMBLY TIME(SEC)
HEXA	4	.07

TOTAL MATRIX ASSEMBLY TIME FOR 4 ELEMENTS IS .07 SECONDS.

ORIGINAL PERFORMANCE DATA

SUPER(GROUP) ID	NO. GRIDS	AV. CONNECTIVITY	C-AVERAGE	C-RMS	C-MAXIMUM	P-GROUPS	P-AVERAGE	DECOMP TIME(SEC) (6.0 DOF/GRID)
0	20	10.40	6.70	7.20	11	0	.00	.017

RESEQUENCED PERFORMANCE DATA

SUPER(GROUP) ID	NO. GRIDS	AV. CONNECTIVITY	C-AVERAGE	C-RMS	C-MAXIMUM	P-GROUPS	P-AVERAGE	DECOMP TIME(SEC) (6.0 DOF/GRID)	METHOD
0	20	10.40	5.70	6.02	8	0	.00	.012	ACTIVE

Average, RMS, and
maximum number of active

Resequencer method
selected

Note: The average and RMS values depend on the manner in which the grid points are sequenced.

Further information can be found in the [MSC.Nastran Numerical Methods User's Guide](#).

This output is available only if the sequencer is turned on (param,oldseq,x).

Figure 5-33 Sequence Processor Output

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET		NEW USET			
				EXCLUSIVE	UNION	EXCLUSIVE	UNION		
53	G	4	0.00E+00	B	F	SB	S	*	
53	G	5	0.00E+00	B	F	SB	S	*	
53	G	6	0.00E+00	B	F	SB	S	*	
54	G	4	0.00E+00	B	F	SB	S	*	
54	G	5	0.00E+00	B	F	SB	S	*	
54	G	6	0.00E+00	B	F	SB	S	*	
57	G	4	0.00E+00	B	F	SB	S	*	
57	G	5	0.00E+00	B	F	SB	S	*	
57	G	6	0.00E+00	B	F	SB	S	*	
58	G	4	0.00E+00	B	F	SB	S	*	
58	G	5	0.00E+00	B	F	SB	S	*	
58	G	6	0.00E+00	B	F	SB	S	*	

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET		NEW USET			
				EXCLUSIVE	UNION	EXCLUSIVE	UNION		
53	G	4	0.00E+00	B	F	SB	S	*	
53	G	5	0.00E+00	B	F	SB	S	*	
53	G	6	0.00E+00	B	F	SB	S	*	
54	G	4	0.00E+00	B	F	SB	S	*	
54	G	5	0.00E+00	B	F	SB	S	*	
54	G	6	0.00E+00	B	F	SB	S	*	
57	G	4	0.00E+00	B	F	SB	S	*	
57	G	5	0.00E+00	B	F	SB	S	*	
57	G	6	0.00E+00	B	F	SB	S	*	
58	G	4	0.00E+00	B	F	SB	S	*	
58	G	5	0.00E+00	B	F	SB	S	*	
58	G	6	0.00E+00	B	F	SB	S	*	

Note: This is automatically output from the Grid Point Singularity Processor (GPSP1) and is obtained anytime a stiffness ratio is less than the parameter EPZERO (default = 10⁻⁸).

See “[Automatic Application of Single-Point Constraints \(AUTOSPC\)](#)” on page 262 for a discussion of automatic single point constraints.

Figure 5-34 Grid Point Singularity Table

*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

LOAD SEQ. NO.	EPSILON	EXTERNAL WORK	EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
1	-4.5888836E-15	6.4518387E+01	
2	-2.8991481E-15	7.5782189E+00	

Residual vector error $\epsilon = \frac{\{u_l\}^T \{\delta P_l\}}{\{u_l\}^T \{P_l\}}$

Work performed by the applied loads $= \frac{1}{2} P_l u_l$

$$\epsilon = \frac{\{u_l\}^T \{\delta P_l\}}{\{u_l\}^T \{P_l\}}$$

where:

$\{P_l\}$ = Load on the l -set degrees of freedom

$\{u_l\}$ = Displacement of the l -set degrees of freedom

$[K_{ll}]$ = Stiffness matrix of the l -set

$$\{\delta P_l\} = [K_{ll}]\{u_l\} - \{P_l\}$$

Note: Small epsilons are due to machine roundoff are acceptable. Large epsilons indicate a potential modeling error.

One line of output is printed for each static loading condition.

Figure 5-35 User Information Message 5293

SUBCASE/ DAREA ID		LOAD TYPE	T1	T2	T3	RESULTANT		
						R1	R2	R3
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	3.307500E+02	0.000000E+00	-8.268750E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-8.268750E+02	0.000000E+00

Subcase number

Resultant of the applied loads about the point specified on the “PARAM,GRDPNT,x”. The default is at the origin of the basic coordinate system.

Note: One line is printed for each subcase.

See “[Overview of the MSC.Nastran Output Files](#)” on page 27 for an example.

This output can be obtained by adding PARAM,PRTRESLT,YES in the bulk data file.

Figure 5-36 OLOAD Resultant

SUBCASE/ DAREA ID		LOAD TYPE	T1	T2	T3	SPCFORCE RESULTANT		
						R1	R2	R3
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	-1.000000E+02	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	5.000000E+02	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	-1.000000E+02	0.000000E+00	5.000000E+02	0.000000E+00
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	-3.307500E+02	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	8.268750E+02	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	-3.307500E+02	0.000000E+00	8.268750E+02	0.000000E+00

Subcase number

Resultant of the single point constraint force about the point specified on the “PARAM,GRDPNT,x”. The default is at the origin of the basic coordinate system.

Note: One line is printed for each subcase.

This output can be obtained by adding PARAM,PRTRESLT,YES in the bulk data file.

Figure 5-37 SPCFORCE Resultant

MAXIMUM SPCFORCES						
	T1	T2	T3	R1	R2	R3
1	5.0000000E+03	5.0096872E+02	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	2.0000000E+03	9.6871246E+01	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

MAXIMUM DISPLACEMENTS						
	T1	T2	T3	R1	R2	R3
1	1.2524217E-02	1.2903677E-01	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	7.5782188E-03	2.5048435E-02	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

MAXIMUM APPLIED LOADS						
	T1	T2	T3	R1	R2	R3
1	0.0000000E+00	1.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	2.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

1. Quantities are transformed to the basic coordinate system.
2. Absolute value of transformed quantities are taken.
3. Largest value for each component is found and printed.

Subcase number

Note: The largest magnitude of quantities transformed into the basic coordinate system is printed. The maximums shown for T1 may not be at the same grid point as T2, etc.

One line is printed for each subcase.

This output can be obtained by adding PARAM,PRTMAXIM,YES in the bulk data file.

Figure 5-38 Maximum SPCFORCES, DISPLACEMENT, and Applied Loads

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	
2	G	.0	.0	.0	.0	.0	.0	
3	G	7.578219E-03	-2.504843E-02	.0	.0	.0	.0	
4	G	-2.421781E-03	-2.495156E-02	.0	.0	.0	.0	

Displacements expressed in the global coordinate system

G for grid point, S for scalar point

Note: Standard DISPLACEMENT output for a static analysis, requested in Case Control Section.

See “[Coordinate Systems](#)” on page 52 for definition of the global coordinate system.

See “[Scalar Points](#)” on page 51 and “[Coordinate Systems](#)” on page 52 for an explanation of the displacement components T1, T2, T3, R1, R2, R3.

Figure 5-39 Displacement Vectors

L O A D V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
4	G	.0	-1.000000E+03	.0	.0	.0	.0	

Applied loads and moments expressed in the global coordinate system

Note: Standard OLOAD output, requested in Case Control Section.

Figure 5-40 Load Vectors

*** TOTAL MEMORY AND DISK USAGE STATISTICS ***										Maximum disk space allocated in GINO blocks										Maximum disk space used in GINO blocks										Disk spaced used in megabytes										Amount of I/O transferred between each DBset and memory									
+----- SPARSE SOLUTION MODULES -----+										+----- MAXIMUM DISK USAGE -----+										+-----+ +-----+ +-----+ +-----+										+-----+ +-----+ +-----+ +-----+										+-----+ +-----+ +-----+ +-----+									
HIWATER		DAY_TIME		SUB_DMAP		DMAP				HIWATER		DAY_TIME		SUB_DMAP		DMAP				HIWATER		DAY_TIME		SUB_DMAP		DMAP				HIWATER		DAY_TIME		SUB_DMAP		DMAP													
(WORDS)				NAME		MODULE				(MB)				NAME		MODULE				(BLOCKS)				NAME		MODULE				(BLOCKS)				NAME		MODULE													
408922		09:56:52		SEKRRS		18		DCMP		2.266		09:56:54		SESTATIC		186		EXIT																															
*** DATABASE USAGE STATISTICS ***																																																	
+----- LOGICAL DBSETS -----+										+----- DBSET FILES -----+																																							
DBSET		ALLOCATED		BLOCKSIZE		USED		USED		FILE		ALLOCATED		HIWATER		HIWATER		I/O		TRANSFERRED																													
		(BLOCKS)		(WORDS)		(BLOCKS)		%				(BLOCKS)		(BLOCKS)		(MB)				(GB)																													
MASTER		5000		2048		149		2.98		MASTER		5000		149		1.164				.010																													
DBALL		250000		2048		9		.00		DBALL		250000		9		.070				.000																													
OBJJSCR		5000		2048		130		2.60		OBJJSCR		5000		130		1.016				.003																													
SCRATCH		500100		2048		20		.00		(MEMFILE		100		62		.484				.000)																													
										SCRATCH		250000		1		.008				.000																													
										SCR300		250000		1		.008				.000																													

PROJECT VERSION DIRECTORY PRINT						
PROJECT_ID	ASSIGNED INT. VALUE		VERSION_ID	CREATION TIME		
" B L A N K "	1		** 1	3/ 1/93	10:44.42	
			** 2	3/ 1/93	10:45.37	
			3	3/ 1/93	10:46. 5	
			** 4	3/ 1/93	10:46.38	
			** 5	3/ 1/93	10:47. 2	
			** 6	3/ 1/93	10:47.25	
			7	3/ 1/93	10:47.49	

There were seven versions created in the database.
 Only Versions 3 and 7 are, however, restartable.
 The other versions denoted by a “**” are not
 restartable.

Figure 5-42 Restartable Versions

APPENDIX

F

Parameters

Each MSC.Nastran solution sequence is designed with a specific set of default values and solution paths. In the majority of the cases, the default values are the ones best suited for your model. There are occasions, however, when you may want to deviate from the default values. Several methods are available in MSC.Nastran to perform this task: DMAP (Direct Matrix Abstraction Program) modules, NASTRAN statements, and Parameters. Their functions, in general, do not overlap each other.

The use of parameters is the most straightforward method, and it is the only topic that will be covered in this section. Refer to the *MSC.Nastran Reference Manual* for a discussion of the other methods. Depending on the parameter, it can be specified either as a PARAM Bulk Data entry and /or a PARAM Case Control command. The format of each is as follows:

PARAM,xname,xvalue

where xname and xvalue are the name and value of the PARAMeter, respectively.

PARAM,WTMASS,0.00259

The terms of the structural mass matrix are multiplied by the value of this parameter when they are generated.

A complete description of all the parameters is available in “**Parameters**” on page 601 of the *MSC.Nastran Quick Reference Guide*.

APPENDIX

G

Resource Estimation

When running large problems, it is essential that you have some idea of the resource requirements--both disk space and computer time--prior to engaging in the actual analysis. A resource usage guide, using empirical data, was written to provide a tool for users in estimating the resource requirement. See the ***MSC.Nastran Resource Usage Guide*** for further details on this subject. Furthermore, a utility program called ESTIMATE provides a set of recommended resource values for running a particular job. See the ***MSC.Nastran 2003 Installation and Operations Guide*** for further details.

APPENDIX

H

Generating the Cylinder in MSC.Patran

■ Procedures

8.1 Procedures

This section summarizes the procedure for generating and interpreting the cylindrical buckling problem shown in “[Example 5 -- Buckling of a Cylinder Under Uniform Axial Load](#)” on page 514 of [Examples](#).

Opening a New Database

1. Create a new database called **buckcy20r.db**.

File/New Database...

New Database Name:

buckcy20r

OK

In the New Model Preference form set the following:

Tolerance:

◆ Default

Analysis Code:

MSC.Nastran

Analysis Type:

Structural

OK

Using the Viewing/Angles Option

2. Use the **Viewing/Angles...** option to change the view of the model.

Viewing/ Angles...

Method

◆ Model Absolute

Angles

-67., 0, 34.

Apply

Cancel

Create Named Views

- Use the **Viewing/Named View Options...** to create a customized view setting.

**Viewing/ Named View
Options...**

Create View...

Create New View

buck_iso

Apply

Close

- We will need one other customized view setting.

Viewing/Angles...

Method

◆ Model Absolute

Angles

-90., 0, 0.

Apply

Cancel

**Viewing/Named View
Options...**

Create View...

Create New View

buck_side

Apply

Applying Named Views

Also, change the view of the model.

Select Named View

buck_iso

Close

Create a Point and a Coordinate System

- Create a new point and use this point to define the origin of a local cylindrical coordinate system.

◆ Geometry

Action:

Create

Object:

Point

Method:

XYZ

Point Coordinates List

[0, 0, 0]

Apply

Action:

Create

Object:

Coord

Method:

Enter

Type:

Cylindrical

Rotation Parameters...

Angle of Rotation

0.0

Angle of Rotation

0.0

Angle of Rotation

0.0

OK

Origin:

[0, 0, 0]

Apply

Create Point Translate Option

- Create another point by translating Point 1 a radial distance of 10 inches. Note that the reference coordinate frame is the local cylindrical system.

Action:

Transform

Object:	Point
Method:	Translate
Refer. Coordinate Frame	Coord 1
Translation Vector	<10, 0, 0>
☒ Auto Execute	
Point List	Point 1

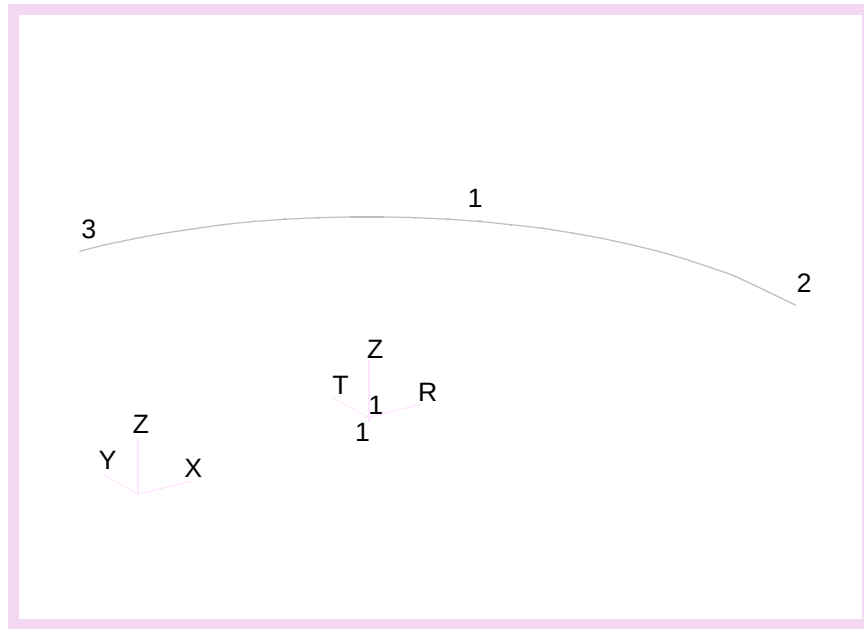
There is no need to click on the **Apply** button since the **Auto Execute** button is toggled on.

Create Curve Revolve Option

- Use Point 2 to create a curve by sweeping it 90 degrees about the Z-axis of the local coordinate system.

Action:	Create
Object:	Curve
Method:	Revolve
Refer. Coordinate Frame	Coord 1
Axis	{[0, 0, 0] , [0, 0, 1]}
Total Angle	90.0
☒ Auto Execute	
Point List	Point 2

Your model should appear as follows:



Create Surface Extrude Option

- Generate one-quarter of the cylinder wall by extruding the curve created in the previous operation.

Action:	Create
Object:	Surface
Method:	Extrude
Refer. Coordinate Frame	Coord 1
Translation Vector	<0, 0, 20>
☒ Auto Execute	
Curve List	Curve 1

Create Surface Rotate Option

- Now complete the model by rotating this surface about the local Z-axis.

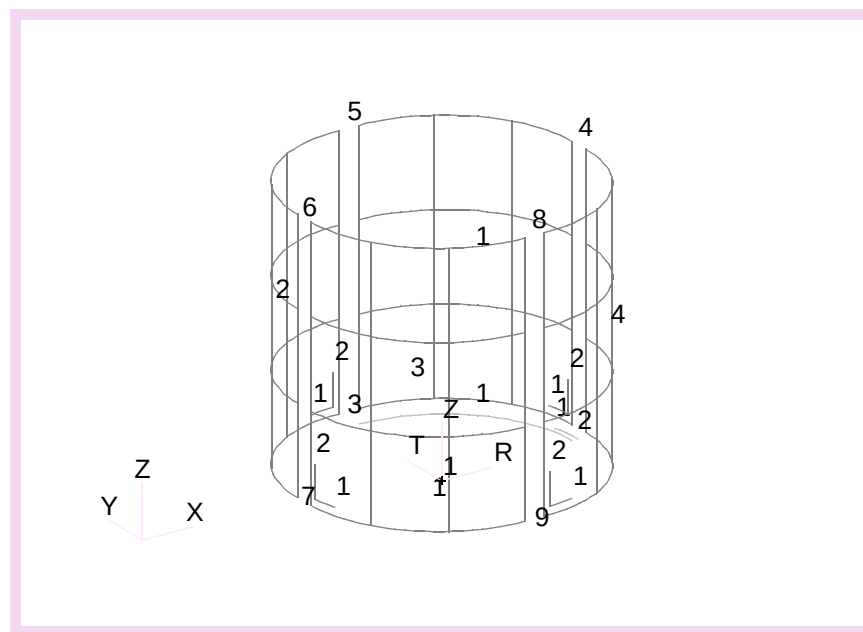
Action:	Transform
Object:	Surface
Method:	Rotate

Refer. Coordinate Frame	Coord 1
Axis	{[0, 0, 0], [0, 0, 1]}
Total Angle	90.0
Repeat Count	3
☒ Auto Execute	
Surface List	Surface 1

- For clarity, shrink the geometry by 10% to see the surface boundaries and turn on the parametric directions using the **Display/Geometry...** option.

Display/Geometry...	
Geometric Shrink	0.10
☒ Show Parametric Directions	
Apply	
Cancel	

Your model should appear as follows:



Changing the Graphics Preferences

- Before you mesh the model, plant mesh seeds to define the size of the elements. Note that the mesh seeds will have precedence over the **Global Edge Length** parameter in the **Create/Mesh** form. You may wish to make the nodes that will be generated during the meshing operation more visible. Do this by changing the node size using the **Display/Finite Elements...** option.

Display/Finite Elements...	
Node Sizes	5
Apply	
Cancel	

Create Mesh Seed

- Create 18 elements along the bottom of the surfaces and 20 elements along the height.

◆ Finite Elements

Action: **Create**

Object: **Mesh Seed**

Type: **Uniform**

Number = **18**

Curve List **Surface 1:4.4**

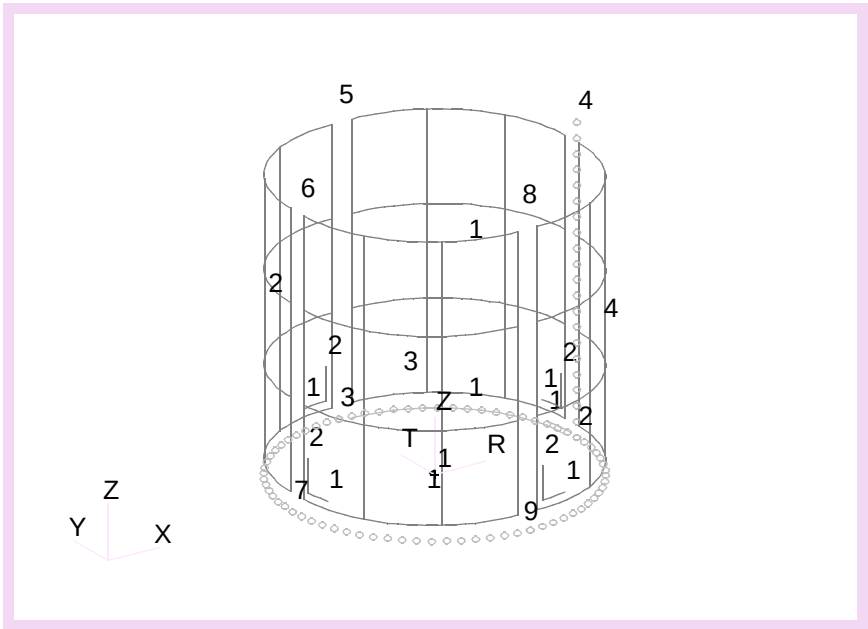
Apply

Number = **20**

Curve List **Surface 1.1**

Apply

The mesh seeds should appear as follows:



5. Disable all entity labels using **Display/Entity Color/Label/Render...** option. For clarity, you may also consider disabling the display of parametric directions.

Display/Entity
Color/Label/Render...

Entity Types

Hide All Entity Labels

Apply

Cancel

- Generate an analysis model by meshing the geometry model.

Note: The node locations are to be specified with respect to our local cylindrical system (i.e. in $r-\theta-z$ coordinates). Hence, we control this by designating the local coordinate system as the Reference Coordinate frame.

However, it is desired to have our displacement results be generated with respect to our Global coordinate system. Therefore, our Analysis Coordinate Frame references the Global coordinate system.

Create Mesh Using QUAD4 Elements

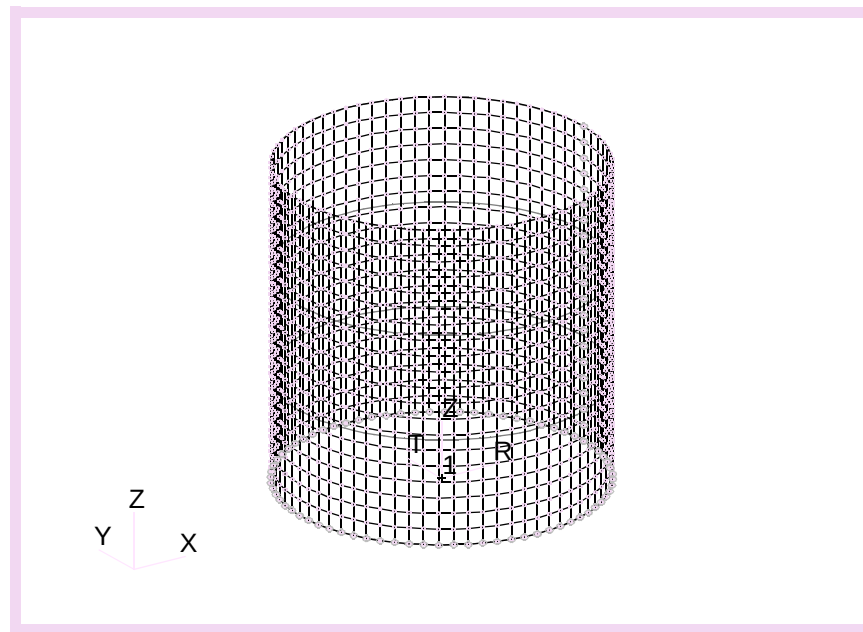
◆ Finite Elements

Action:

Create

Object:	Mesh
Type:	Surface
Element Topology	Quad4
Mesher	◆ IsoMesh
Node Coordinate Frames	
Analysis Coordinate Frame	Coord 0
Refer. Coordinate Frame	Coord 1
OK	
Surface List	Surface 1:4
Apply	

You model should appear as follows:



Equivalence Method

Remember to equivalence the model to remove duplicate nodes at common surface edges.

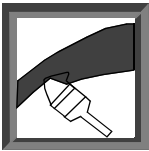
◆ Finite Elements	
Action:	Equivalence
Object:	All

Method:

Tolerance Cube

Apply

Refresh the display when needed using the brush icon on the Top Menu Bar.



Using the View/Named View Options

- 6. Use the **Viewing/Named View Options...** to change the view of the model.

Viewing/Named View Options...

Select Named View

default_view

Close

- Erase All FEM for clarity using the option in **Display/Plot/Erase...**

Create a Node

- Create a new node at Z=20 inches. The compressive load will be applied to this node. The Analysis & Reference coordinate frames are set in accordance with the previous step.

◆ Finite Elements

Action:

Create

Object:

Node

Method:

Edit

Node ID List

5000

Analysis Coordinate Frame

Coord 0

Refer. Coordinate Frame

Coord 1

☐ Associate with Geometry

Node Location List

[0, 0, 20]

Apply

- For clarity, set Geometric Shrink to zero and remove Parametric Direction labels.

Display/Geometry...

Geometric Shrink:

0.0

☐ Show Parametric Direction

Apply

Cancel

- Turn on node labels.

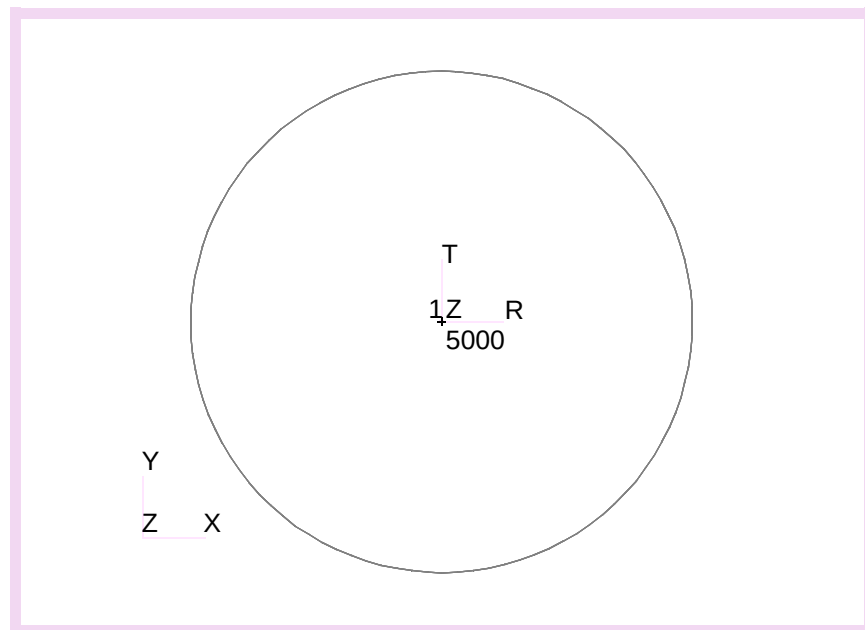
**Display/Entity
Color/Label/Render...**

Node:

■ Label

Apply

The model should appear as follows:



7. Next, create an RBE3 element to distribute the load from the load application node to the cylinder walls.

- For clarity:

Hide All Entity Labels

Plot All Posted FEM

Erase All Geometry

Create and Post a New Group

- Use your mouse to place the top row of elements and their top most nodes into a new group called rbe3_region. Make this group current and post only this group to the display.

Viewing/Named View Options...

Select Named View

Close

buck_iso

Preferences/Picking...

Single Picking

Rectangle / Polygon Picking

Close

◆ Entity

◆ Enclose Centroid

Group/Create...

New Group Name

Make Current

Unpost All Other Groups

Entity Selection

Apply

Cancel

rbe3_region

Select the top row of nodes (Z=20) and associated elements using the mouse

- Reset the viewing angles to an isometric view.

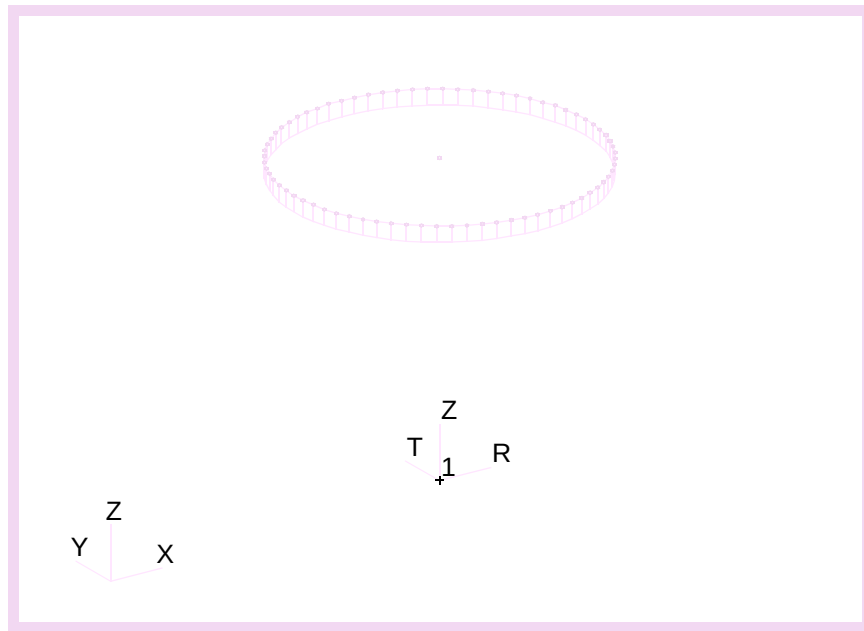
View/Named View Options...

Select Named View

Close

buck_iso

At this point, your model should appear as follows:



Create an RBE3 Element

- Generate the RBE3 element. Hint: select your independent nodes by using the polygon pick feature of your mouse (Ctrl-mouse click).

◆ Finite Elements

Action:

Create

Object:

MPC

Type:

RBE3

Define Terms...

◆ Create Dependent

☐ Auto Execute

Node List

Node 5000

DOFs

UX,UY,UZ,RX,RY,RZ

Apply

◆ Create Independent

Node List

Select all remaining
nodes

DOFs

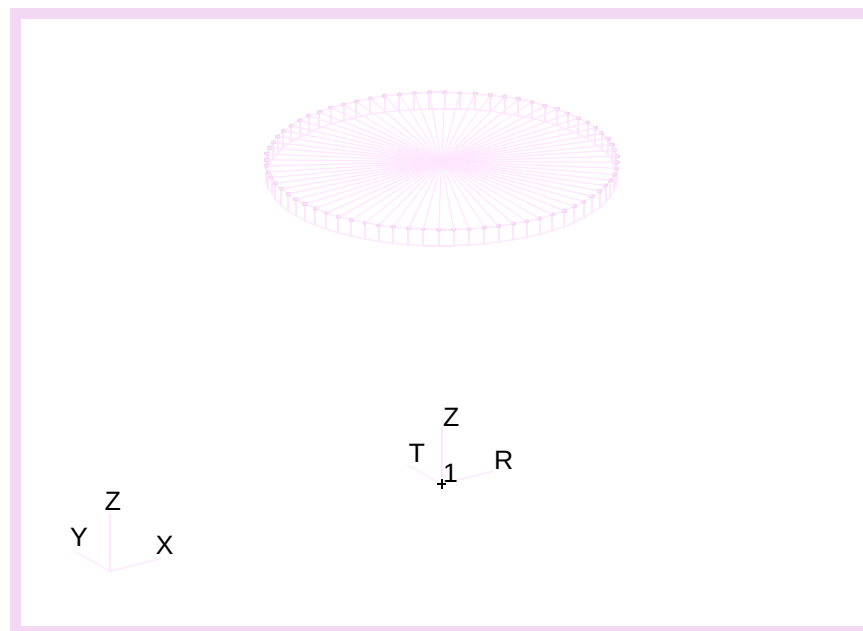
UX,UY,UZ

Apply

Cancel

Apply

This operation should yield the following display:



- Create a new group consists of all the geometry called **geometry_only** and make it current.

Group/Create...

New Group Name:

geometry_only

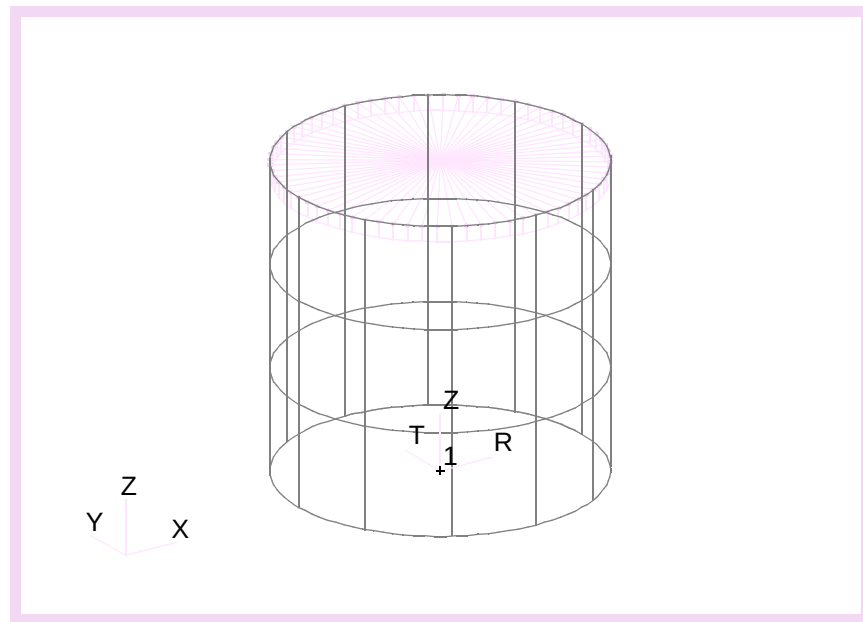
☒ Make Current☐ Unpost all other groups

Group Contents:

Add All Geometry

Apply

The view of the group is shown here (note that rbe3_region group is still posted):



8. Define a material using the specified Modulus of Elasticity, Poisson ratio & density.

◆ Materials

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name

aluminum

Input Properties

Constitutive Model:

Linear Elastic

Elastic Modulus:

1.0e7

Poisson Ratio:

.3

Apply

In the Current Constitutive *Models*, you will see **Linear Elastic - [,,,] - [Active]**.
Click on Cancel to close the form.

Cancel

Create Element Properties

- Now, reference the material you just defined when you specify element properties for your analysis model. Remember to use the specified cylinder thickness.

◆ Properties

Action:	Create
Object:	2D
Method:	Shell
Property Set Name	cylinder_prop
Input Properties...	
Material Name	m:alum
Thickness	0.03
OK	
Select Members	Surface 1:4
Add	
Apply	

- Define the boundary constraints for the model. Refresh the display as needed before continuing.

Create Two Displacement Constraints

- Create displacement constraints and apply it to the model. Recall that the top and base of the cylinder have different boundary conditions. We will apply these constraints to the geometry model instead of the analysis model. First, we define the base constraints:

◆ Load / BCs

Action:	Create
Object:	Displacement
Method:	Nodal

New Set Name	base
Input Data...	
Translation <T1 T2 T3>	<0, 0, 0>
OK	

Select Application Region	
Geometry Filter	◆ Geometry
Select Geometry Entities	Surface:1:4.4
Add	
OK	
Apply	

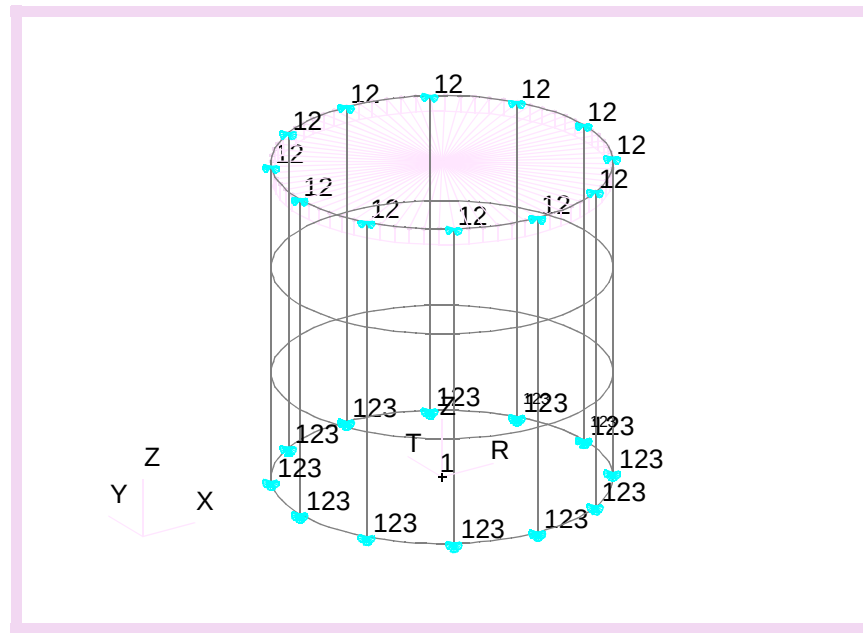
- Next, we define the rim constraints:

◆ Load /BCs	
Action:	Create
Object:	Displacement
Method:	Nodal

New Set Name	top
Input Data	
Translation < T1 T2 T3 >	< 0, 0, >
OK	

Select Application Region...	
Geometry Filter	◆ Geometry
Select Geometry Entities	Surface 1:4.2
Add	
OK	
Apply	

The displacement constraints should appear as follows:



11. Select the broom icon on the Top Menu Bar to reset the display. Enable node labels by using the option in Display / Finite Elements. Also erase all nodes (except for Node 5000) and RBE3s by using Display / Plot / Erase...

Hint: Use the Select menu to filter out all unnecessary entities.

Create Applied Force

- Apply the compressive force along the local Z-axis of Node 5000.

◆ Load / BCs

Action:

Create

Object:

Force

Method:

Nodal

New Set Name

load

Input Data...

Force < F1 F2 F3 >

< , , -100000 >

OK

Select Application Region

Geometry Filter

◆ FEM

Select Nodes

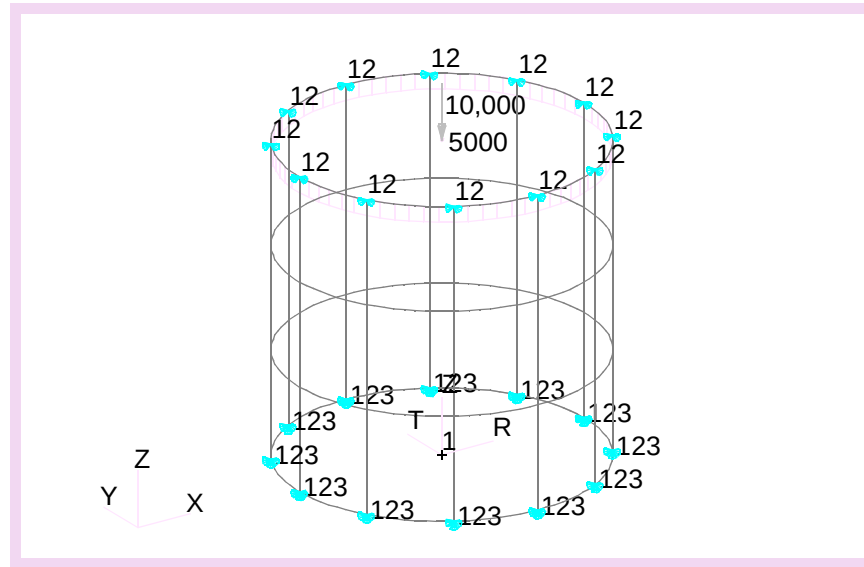
Node 5000

Add

OK

Apply

The load and boundary constraints should appear as follows:



Create a New Group

- For clarity, create a new group called **fem_only** that consists only of analysis model entities.

Group/Create...

New Group Name

fem_only

☒ Make Current

☒ Unpost All Other Groups

Group Contents

Add All FEM

Apply

Cancel

- Since the boundary & load sets were applied to the geometry model, the display of the load sets will only appear on the geometry model. This can be changed using **Display/Load/BC/Elem. Props...** option.

Display/Load/BC/Elem.
Props...

■ Show on FEM Only

Apply

- Also, for clarity, you may wish to disable the display of vector values.

Display/Load/BC/Elem. Props...

Properties

Vector...

Vector Control

☐ Show LBC/EL Prop Values

Apply

Cancel

Turn off all node labels as well.

Display/FiniteElements...

Node:

Apply

☐ Label

Plotting the Load/BCs Markers

- Display all of the loads & boundary markers on the analysis model.

◆ Load /BCs

Action:

Plot Markers

Select all the **Load/BC** sets in the Assigned Load /BCs Sets box by highlighting all of them. Post the markers onto the current group.

Assigned Load /BCs Sets

Displ_base
 Displ_top
 Force_load

Select Groups

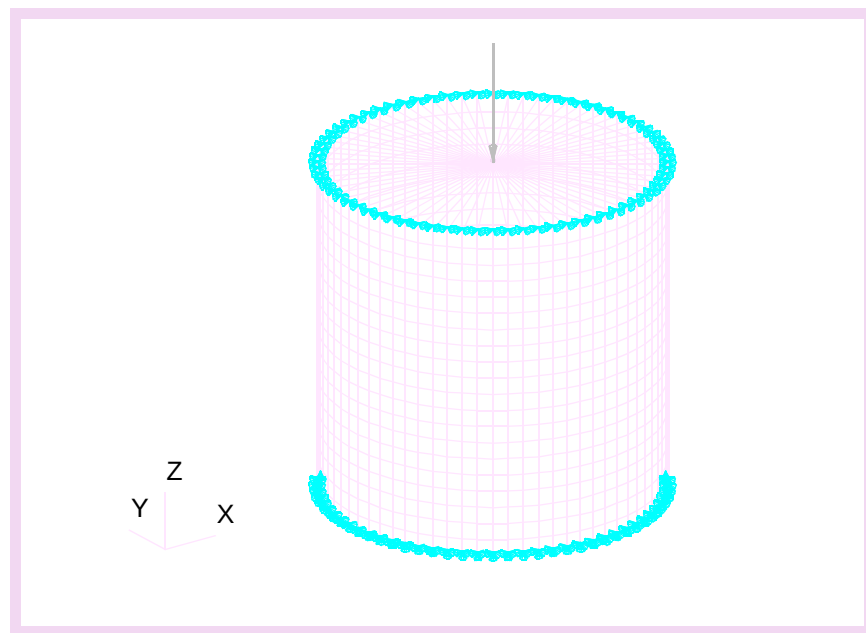
fem_only

Apply

13. Change the Render Style of your model from Wireframe to Hidden Line.

Display/Entity Color/Label/Render...	
Render Style	Hidden Line
Apply	
Cancel	

Your model should appear as follows:



14. Now you are ready to generate an input file for analysis.

Click on the **Analysis** radio button on the Top Menu Bar and complete the entries as shown here.

◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name

cylinder

Solution Type...	
Solution Type:	
Solution Parameters	◆ Buckling
■ Database Run	
■ Automatic Constraints	
Data Deck Echo:	Sorted
Plate RZ Stiffness Factor=	100.
Wt.-Mass Conversion =	0.00259 (English units)
Eigenvalue Extraction	
Number of Desired Roots =	2
OK	
OK	
OK	
Apply	

An input file named **cylinder.bdf** will be generated. The process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the MSC.Patran Heartbeat turns green.

15. If all is well, you can then submit the input file to MSC.Nastran for analysis. To do this, find an available xterm window and at the prompt enter:

nastran cylinder.bdf scr=yes

- When the run is completed, edit the **cylinder.f06** file and search for the word FATAL. If none exists, search for the word WARNING. Determine whether or not existing WARNING messages indicate modeling errors.
- While still editing **cylinder.f06**, search for the word:

R E A L (spaces are necessary)

What are the eigenvalues associated with the 2 buckling modes for our structure?

mode 1 = _____

mode 2 = _____

What is the critical buckling load for this structure?

Pcr = _____

16. Proceed with the Reverse Translation process, that is, importing the **cylinder.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ Analysis

Action:

Attach XDB

Object:

Result Entities

Method:

Local

Select Results File...

Selected Results File

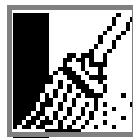
Select the desired .xdb file

Apply

When the translation is complete and the Heartbeat turns green, click the *Results* button.

Choose the desired result case in the Select Result Cases list and select the result(s) in the Select Fringe Result list and / or in the Select Deformation Result list. And click Apply to view the result(s) in the viewport.

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the broom icon.



Quit MSC.Patran when you have completed this exercise.

APPENDIX

I

References

1. R. H. MacNeal and M. A. Gockel (eds.), *MSC/NASTRAN Handbook for Linear Analysis*, Version 64, The MacNeal-Schwendler Corporation, Los Angeles, California, 1985.
2. R. H. Gallagher, *Finite Element Analysis Fundamentals*, Prentice-Hall, Inc., New Jersey, 1975.
3. S. P. Timoshenko and J. N. Goodier, *Theory of Elasticity*, McGraw-Hill Book Company, Third Edition, 1970.
4. R. S. Lahey, M. P. Miller, and M. A. Reymond (eds.), *MSC/NASTRAN Reference Manual*, The MacNeal-Schwendler Corporation, Los Angeles, California,.
5. *MSC/NASTRAN Application Manual*, Version 67, The MacNeal-Schwendler Corporation, Los Angeles, California, June 1991.
6. K. Kilroy, *MSC.Nastran Quick Reference Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.
7. K. Ranger, *MSC.Nastran Common Questions and Answers*, The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.
8. *MSC/NASTRAN Release Notes for Version 67.5*, The MacNeal-Schwendler Corporation, Los Angeles, California, 1993.
9. S. P. Timoshenko and J. M. Gere, *Theory of Elastic Stability*, Engineering Societies Monograph Series, Second Edition, 1961.
10. L. Komzsik, *MSC.Nastran Numerical Methods User's Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.
11. *MSC/NASTRAN Release Notes for Version 68*, The MacNeal-Schwendler Corporation, Los Angeles, California, 1994.
12. M. A. Reymond, *MSC.Nastran 2005 DMAP Programmer's Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California.
13. R. J. Roark and W. C. Young, *Formulas for Stress and Strain*, McGraw-Hill, Fifth Edition, 1975.
14. D. T. Greenwood, *Principles of Dynamics*, Prentice-Hall, 1965.
15. "A Proposed Standard Set of Problems to Test Finite Element Approach," *MSC/NASTRAN Application Notes*, 1984.
16. *MSC.Nastran 2001 Installation and Operations Guide*.
17. Grant G. Sitton, *MSC.Nastran Basic Dynamics User's Guide*, Version 69, The MacNeal-Schwendler Corporation, Los Angeles, California.
18. R. H. MacNeal (ed.), *The NASTRAN Theoretical Manual*, December 1972.
19. S. H. Lee (ed.), *MSC/NASTRAN Handbook for Nonlinear Analysis*, Version 67, The MacNeal-Schwendler Corporation, Los Angeles, California.
20. R. T. Jones, *Mechanics of Composite Materials*, McGraw-Hill, 1975.
21. B. Walton, *Getting Started with MSC.Nastran User's Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.

22. M. S. Chainyk, *MSC.Nastran Thermal Analysis User's Guide*, The MacNeal-Schwendler Corporation, Los Angeles, California, 2001.
23. R. H. MacNeal, *Finite Elements: Their Design and Performance*.
24. *MSC/NASTRAN Release Guide for Version 69*, The MacNeal-Schwendler Corporation, Los Angeles, California, 1996.
25. *MSC.Nastran Resource Usage Guide*, The MacNeal-Schwendler Corporation, Los Angeles, California.
26. *MSC/NASTRAN Release Guide for Version 69.1*, The MacNeal-Schwendler Corporation, Los Angeles, California, 1996.
27. *MSC/NASTRAN Linear Static Analysis User's Guide for Version 69+*, The MacNeal-Schwendler Corporation, Los Angeles, California, 1997

I N D E X

MSC.Nastran Linear Static Analysis User's Guide

Symbols

.op2, 142

.xdb, 142

A

ADUM8, 145

ADUM9, 153

Anisotropic material, 178, 181, 187, 188, 200

Antisymmetric, 364, 366

ASCII, 28, 548

ASET, 463

a-set, 458, 462, 464, 466, 467, 468

ASET1, 463, 464, 468, 470

ASSIGN, 526, 547

Automatic restart, 526, 539

AUTOSPC, 133, 149, 263, 264

Axisymmetric, 594

Axisymmetric analysis, 178, 182, 560, 562, 565, 570, 600

B

Backup structure, 471

BAR, 389

Basic coordinate system, 52, 359

BCD, 41

BEAM, 389

Beam cross section library, 87

BEGIN BULK, 24

Bending, 197

Block, 543, 546, 556

BOTH, 39

Boundary conditions, 259, 269, 560, 569

Brick element, see also CHEXA

Buckling, 8, 486, 489, 490, 491, 497, 498, 507, 508, 510, 511, 513, 514, 517

BUFFSIZE, 23, 525, 543, 556

Bulk Data entries, 470, 494

summary list of, 634

Bulk Data Section, 41, 246, 386, 472, 494

Bytes, 545

C

Cantilever beam, 380

Case Control commands

summary list of, 630

Case Control Section, 23, 26, 34, 246, 292, 383, 386, 395, 401, 402, 474, 481, 491, 589

CBAR, 6, 66, 77, 165, 223, 249, 313, 314, 327, 371, 375, 396, 401, 413, 415, 497, 499, 503

element coordinate system, 82

orientation vector, 80

CBARAO, 226

CBEAM, 66, 103, 223, 249, 313, 314, 375, 497, 499, 503

CBEND, 66, 123, 223, 497

CBUSH, 66, 158, 165

CBUSH elements, 156

CD, 49, 50, 52, 265

CD, see also Coordinate system

CELAS1, 66

CELAS2, 66

CELAS3, 66

CELAS4, 66

CELASi, 69

CEND, 23, 32

Centrifugal force, 234

CHEXA, 68, 148, 187, 189, 318, 327, 350, 356, 361, 372, 373, 391, 497, 531

element coordinate system, 150

Circumferential stress, 320

CMASS1, 66

CMASS2, 66

CMASS3, 66

CMASS4, 66

CMASSi, 236

Coldstart, 525, 527

Column, 456, 477

Comments (\$), 32

Composite element, 199, 208

Compressive load, 499, 510, 511, 514

Computer graphics, 371
 Computer speed, 522
 Concatenate, 548
 Concentrated load, 224
 Concentrated masses, see also CMASS
 CONMi, 236, 396
 CONROD, 77, 249, 497
 Consistent, 376, 396, 432, 457
 Consistent load, 360, 361
 Constraint equation, 286
 contact, 275
 Continuation identifier, 42
 Continuation lines, 42
 Contour plot, 349
 Coordinate system, 52, 357, 568, 574
 CORD1C, 58
 CORD1R, 54, 55
 CORD1S, 59
 CORD2C, 58, 61
 CORD2R, 54
 CORD2S, 59
 coordinate system, 405
 Corner, 352, 354
 Corner output, 138, 324, 351, 355, 356
 BILIN, 324
 CORNER, 324
 CUBIC, 324
 SGAGE, 324
 CP, 49, 52
 CP, see also Coordinate system
 CPENTA, 68, 148, 187, 318, 350, 356, 361, 391, 497
 CPU time, 556
 CQUAD4, 7, 9, 25, 67, 127, 140, 218, 230, 249, 314, 315, 320, 323, 324, 327, 331, 350, 353, 355, 356, 360, 371, 375, 377, 383, 389, 413, 415, 425, 427, 428, 430, 432, 497, 531
 element coordinate system, 129
 CQUAD8, 67, 127, 139, 140, 249, 314, 315, 320, 323, 324, 327, 350, 353, 355, 356, 360, 497
 element coordinate system, 140
 CQUADR, 67, 127, 142, 316, 350, 360, 389
 CRAC2D, 68, 145
 CRAC3D, 68, 153
 CROD, 25, 66, 73, 249, 313, 497
 element coordinate system, 74
 Cross-sectional properties, 87
 CSHEAR, 67, 143, 230, 497
 CTETRA, 148, 350, 356, 391, 497
 CTRIA3, 67, 127, 140, 218, 230, 249, 314, 327, 350, 353, 356, 360, 383, 390, 497

CTRIA6, 67, 127, 139, 140, 249, 314, 350, 353, 355, 356, 360, 497
 CTRIAR, 67, 127, 142, 316, 350, 360, 390
 CTRIAX6, 68, 152, 182
 CTUBE, 77, 249, 497
 Curvatures, 135
 CWELD, 165, 168, 169, 170, 171, 174
 CYAX, 582, 584
 Cyclic symmetry, 363, 368, 582, 589
 CYJOIN, 582
 Cylinder, 514
 Cylindrical coordinate system, 52, 61
 Cylindrical coordinate system, see also CORD1C
 Cylindrical coordinate system, see also CORD2C
 CYSUP, 582, 585
 CYSYM, 582, 583

D

Data block, 543
 Database, 522, 525, 526, 529, 535, 539, 550, 553, 554
 DATAREC, 411
 DBALL, 543, 544, 546, 549, 553
 DBCLEAN, 535, 550, 551
 DBset, 543, 544, 546, 548, 553
 Debugging, 386
 DECOMP, 398
 DECOMP, see also Decomposition
 Decomposition, 522
 DEFORM, 218, 241, 245, 246, 491, 589
 Degrees of freedom, 11, 464
 Degrees of freedom, see also DOFs
 DELETE, 547
 Density, 396
 Dependent degree of freedom, 286
 Determinant, 489
 DIAG, 32, 33
 Diagnostic tools, 370, 386
 Differential stiffness, 487, 490, 497
 Dihedral symmetry, 561, 565, 577
 Direct matrix input, 456
 Disk space, 522
 Displacement, 39, 428
 Dissimilar element types, 336
 Distributed loads, 230
 DMAP, 546, 547
 DMI, 413
 DMIG, 276, 412, 438, 471, 472, 477, 478, 481,

484
DOFs, 324, 336, 338, 472
DSYM, 589, 593
Dynamic analysis, 4

E

ECHO, 39
 BOTH, 39
 NONE, 39
 PUNCH, 41
 SORT, 39
 UNSORT, 39
ECHOOFF, 32, 33
ECHOON, 32, 33
Edge, 352, 373
Edge loads, 220
EIGB, 494, 511
Eigenvalue, 263, 490, 493, 499, 511, 517
Eigenvalue extraction method, 493
 enhanced inverse power, 493
 Givens, 493
 Householder, 493
 inverse power, 493
 Lanczos, 493
 modified Givens, 493
 modified Householder, 493
Eigenvector, 493
EIGRL, 494, 501, 504, 511
Elapsed, 539
Elastic, 497
Elastic stability, 486
Element coordinate system, 62
element geometry checks, 387
Element stiffness matrix, 53
Element strain energy, 379, 382, 383
Element stress, 419
ELEMID, 169
ELSDCON, 426
ELSTRESS, 201
ELSTRESS, see also STRESS
ELSUM, 387
ENDDATA, 24
ENDJOB, 535
Enforced displacements, 267
Enhanced inverse power, 493
Epsilon, 30, 399, 438
EPZERO, 263
Equilibrium, 403, 457, 486, 488
Equivalent section, 208
Error estimate, 424, 425

ESE, 381
ESE, see also Element strain energy
Estimate, 494
Executive Control Section, 23, 32, 357, 386
EXPAND, 548, 549
External programs, 484
External work, 30

F

Face, 373
Failure index table, 201
File Management Section (FMS), 23, 542, 545
Files
 dat, 27, 28
 DBALL, 27, 28, 543, 544, 546, 549, 553
 f04, 27, 28
 f06, 27, 28
 log, 27, 28
 MASTER, 27, 28, 543, 544, 553, 554
 op2, 27
 pch, 27
 plt, 27
Finite element, 9
Finite element method, 4
Finite element model, 48
Flexibility, 471
FMS, 528, 545, 547, 550, 552, 553
FMS, see also File Management Section
FORCE, 39, 218, 219, 246, 586
FORCE1, 218, 219, 246
FORCE2, 218, 219, 246
FORTRAN, 548
Fourier series, 570
Free, 373
 edge, 373
 face, 373
f-set, 458
Fundamental region, 562, 567

G

GAP, 275
GENEL, 160, 413, 471

GEOMCHECK, 388, 392, 393
 BAR_OFF, 392
 BEAM_OFF, 392
 HEX_AR, 392
 HEX_DETJ, 392
 HEX_EPLR, 392
 HEX_WARP, 392
 PEN_AR, 392
 PEN_DETJ, 392
 PEN_EPLR, 392
 PEN_WARP, 392
 Q4_IAMAX, 392
 Q4_IAMIN, 392
 Q4_SKEW, 392
 Q4_TAPER, 392
 Q4_WARP, 392
 T3_IAMAX, 392
 T3_SKEW, 392
 TET_AR, 392
 TET_DETG, 392
 TET_DETJ, 392
 TET_EPLR, 392
 Geometric, 351
 Geometry, 487, 488
 GINO blocks, 545, 554
 Givens, 493
 Global coordinate system, 412
 Global stiffness matrix, 11, 53
 GPFORCE, 305, 412
 GPSDCON, 426
 GPSP, see also Grid Point Singularity Table
 GPSTRESS, 350, 357, 358, 359, 426
 GPWG, 396, 397, 443
 GRAV, 218, 234, 246, 247, 586, 588
 Gravity loads, 234, 401
 GRDSET, 262
 GRID, 41, 49, 262
 Grid point, 11, 48, 49
 Grid point loads, 360
 Grid Point Singularity Table, 263
 Grid point stress, 350, 353, 357, 419
 Grid point weight generator, 614
 GPWG, 614
 GRIDID, 169
 GROUNDCHECK, 392, 410, 411
 g-set, 457, 458
 g-size, 565
 Guyan reduction, 458, 462, 467, 468, 469

H

HARMONICS, 589, 593
 Hencky-von Mises, 331, 350, 351
 Hencky-von Mises, see also von Mises stress
 HEXA, 389
 Hidden line, 372
 Honeycomb, 208, 209
 Honeycomb plate, 202
 Householder, 493
 HOUTPUT, 589

I

I/O time, 556
 ID, 32
 Include, 470
 Independent degrees of freedom, 286, 301
 Inertia properties, 375
 Inertia relief, 437, 438
 INIT, 547
 Initial strains, 241
 Input file, 21
 Input file structure, 524
 INPUTT4, 471
 Instability, 486
 Interior, 352, 355
 Inverse iteration, 493
 Inverse power, 493
 Isotropic material, 178, 179, 200

K

K2GG, 474, 477, 481
 K6ROT, 133
 KEEP, 536, 550
 Kilobytes, 545
 Kilobytes, see also Bytes
 Kilowords, 545
 Kilowords, see also Words

L

Lamina, 200
 Lanczos, 493, 494, 499, 503, 517
 Lateral buckling, 503
 Linear buckling, 486
 Linear stiffness, 487, 488
 Linearly varying forces, 223
 LOAD, 25, 245, 246, 269, 472, 586, 589

Load paths, 5
 LOADCYH, 582, 585
 LOADCYN, 582, 585
 LOADCYT, 582, 585, 588
 LOADSET, 588
 Logical name, 543
 Log-key, 548
 l-set, 458, 460

M

M2GG, 397, 474, 481
 Mass, 236, 472
 grid point weight generator, 614
 MASTER, 543, 544, 553, 554
 MAT1, 26, 179, 197, 397
 MAT2, 181, 199, 208
 MAT3, 182
 MAT8, 183, 208
 MAT9, 187, 188
 Material coordinate system, 62
 MATi, 396
 Matrix, 456
 Matrix operation, 456
 Matrix partitioning, 456
 maximum, 404
 MAXMIN, 404, 405
 MCID, 140, 207
 Mechanism, 397
 Megabytes, 545, 553
 Megabytes, see also Bytes
 Megawords, 545
 Membrane, 197
 Membrane stiffness, 511
 Memory, 494
 Mesh, 327
 Mesh density, 7, 320
 METHOD, 491, 494
 MID1, 197
 MID2, 197
 MID3, 197
 MID4, 197
 Mini-database restarts, 538
 minimum, 404
 Modified Givens, 493
 Modified Householder, 493
 MOMENT, 218, 222, 246, 586
 MOMENT1, 218, 222, 246
 MOMENT2, 218, 222, 246
 MPC, 165, 259, 263, 271, 278, 287, 291, 319,
 464, 589

MSC.Patran, 95, 97, 517
 beam cross-section library support, 95
 MSET, 168, 169, 170
 m-set, 272, 273, 457, 458, 460
 MSGLIMIT, 389
 MSGTYPE, 389
 Multiple-point constraints, see also MPC
 Multi-point constraint, 457

N

NASTRAN statement, 23
 Negative eigenvalue, 504
 Neutral axis, 104
 NOKEEP, 535, 550
 Non-colinear, 338
 NONE, 39
 Nonlinear, 5
 Nonlinear analysis, 138
 Nonlinear material, 497
 Normal, 375, 400
 Normal loads, 220
 Normal pressure, 232
 NOUTPUT, 589, 593
 n-set, 457, 458
 NSM, 403

O

Offset, 81, 497
 Offset beams, 400
 OLOAD, 221, 400, 401, 402, 444
 OLOAD RESULTANT, 30, 443
 OMID, 141, 142
 OMIT, 463, 465, 585
 OMIT1, 463, 465, 468, 470
 One-dimensional elements, 66, 73, 314
 Orientation vectors, 375
 Orthotropic material, 178, 182, 183, 200
 o-set, 458, 462, 464, 466, 468
 OUTPUT, 357
 Output coordinate system, 53
 OUTPUT(POST), 426

P

P2G, 474, 481

PARAM, 275, 276, 278, 341, 343, 436, 437, 438
 AUTOSPC, 398, 458
 CDITER, 275, 276, 278
 CDPCH, 276
 CDPRT, 276
 CDSHUT, 276
 GRDPNT, 400, 402, 436, 437, 438, 615
 INREL, 436, 438
 K6ROT, 174, 497
 MAXRATIO, 398
 SEP1XOVR, 341
 TOLRSC, 341, 343
 USETPRT, 461
 WTMASS, 236, 396, 617
 patch-to-patch, 165, 166, 171, 175
 PBAR, 403
 PBARL, 87, 88
 PBEAM, 105
 PBEAML, 113
 Cross sectional dimensions, 113
 PCOMP, 181, 199, 205, 208
 PENTA, 389
 Pin flags, 294
 Planar, 487
 Planar frame buckling, 508
 Plane of symmetry, 363
 Plate, 349, 351
 PLOAD, 218, 230, 246
 PLOAD1, 123, 218, 223
 PLOAD2, 218, 230, 246
 PLOAD4, 218, 230, 246, 248, 361
 PLOADX, 246
 Ply coordinate system, 207
 point-to-patch, 167
 point-to-point, 168, 171
 Postprocessor, 403, 428
 Postprocessor checks, 370
 Preprocessor checks, 370
 Preprocessors, 371, 373
 Pressure load, 361, 375, 377
 Principal mass axes, 618
 Principal stiffness, 398
 Principal stress, 130, 432
 PROD, 25, 73
 Project, 535
 Projected loads, 224, 227
 project-ID, 551
 proj-ID, 550, 551
 PS, 49
 PSHEAR, 143
 PSHELL, 197, 199, 208
 PSOLID, 151

PUNCH, 41
 PWELD, 165, 168
 Pyramid element, see also CTETRA

Q

QUAD4, 389
 QUADR, 389

R

RBAR, 165, 286, 287, 290, 319, 458
 RBE1, 286, 287
 RBE2, 165, 286, 287, 297, 319, 338
 RBE3, 165, 286, 288, 301, 337, 338, 511, 517
 Reaction, 403
 Reaction loads, 228, 402
 Rectangular matrices, 474
 REFC, 303
 Reference degree of freedom, 437
 reference frame, 405
 REFGRID, 303
 Reflective symmetry, 560, 561, 565, 567
 Releases, 98
 REPCASE, 37
 Restart, 522, 525, 528
 KEEP, 536, 550
 NOKEEP, 535, 550
 Restartable versions, 535
 RFORCE, 218, 234, 246, 586, 588
 Rigid body modes, 458
 Roots, 494, 504
 Rotational, 336, 338
 Rotational symmetry, 560, 561, 565, 572
 Row, 456, 477
 RROD, 286, 287
 r-set, 437, 458, 460
 RSPLINE, 286, 288, 328, 330
 RSSCON, 288, 338, 339, 342
 RSSCON connector element, 338
 RTHRESH, 412
 RTRPLT, 286, 287
 R-type elements, 271, 319

S

sb-set, 460
 Scalar elements, 66, 69
 Scalar point, 48, 51
 SCR300, 553, 555

SCRATCH, 544, 553, 555
 SCRATCH DBset, 546
 Sequencer, 399
 SET, 36, 49, 358, 401, 402, 457
 a-set, 458, 462, 464, 466, 467, 468
 f-set, 458
 g-set, 458
 l-set, 458, 460
 m-set, 272, 273, 457, 458, 460
 n-set, 457, 458
 o-set, 458, 462, 464, 466, 468
 r-set, 437, 458, 460
 sb-set, 460
 sg-set, 458
 s-set, 458, 460
 Shaded plots, 372
 Shear center, 104
 Shear stress, 130
 Shell, 351
 Shell normals, 316
 Shell-to-solid element connector (RSSCON), 338
 Shrink, 371, 372
 Sine wave, 497, 517
 Single-point constraints, see also SPC
 Singularity, 397
 SINV, 494, 511
 SLOAD, 222, 246
 SNORM, 174
 SOL, 32
 101, 357
 105, 497
 106, 497, 514
 Solid, 338, 349
 Solid elements, 361, 371
 SORT, 39
 SPC, 25, 50, 263, 267, 271, 305, 412, 458, 468, 491, 589
 SPCD, 246, 261, 267, 586
 SPCFORCE, 402
 SPCi, 464
 SPCi, see also SPC
 Spherical coordinate system, see also CORD1S
 Spherical coordinate system, see also CORD2S
 SPOINT, 51, 456
 Spring, 379
 s-set, 458, 460
 SSSAlter, 410
 Static analysis, 4, 11
 Static condensation, 456, 458, 462

Stiffened panel, 510
 Stiffness, 336, 338, 398, 457, 471, 472
 Stiffness matrix, 522
 strain, 135, 136
 Strain energy, 438
 Strain energy output, 370
 Strain free, 412
 STRESS, 39, 130, 357
 Stress contour plots, 415, 428
 Stress discontinuity, 419
 Stress error estimates, 370
 Stress gradients, 320
 Stress invariants, 418
 Stress recovery location, 86
 STRFIELD, 357
 Structured solution sequence, 33
 Sturm sequence, 493
 SUBCASE, 34, 367, 404
 SUBCOM, 38, 39, 249, 367, 404, 590
 SUBSEQ, 39, 249, 367, 590
 Superelement, 368, 531
 SUPORT, 276, 278, 437, 438, 439, 458
 SURFACE, 358, 423, 425
 SYMCOM, 560
 Symmetric matrices, 474
 Symmetry, 86, 260, 363, 364, 560
 SYMSEQ, 560

T

TABLEDi, 588
 TEMP, 246, 249, 547
 TEMP(LOAD), 245, 246, 491, 589
 TEMPD, 246, 249
 TEMPP1, 246, 249
 TEMPP3, 246
 TEMPRB, 246, 249
 TETRA, 389
 Thermal coefficients of expansion, 412
 Thermal loads, 249
 THETA, 140
 Three-dimensional, 178, 182
 Three-dimensional elements, 68, 318
 THRESH, 411
 TIME, 32, 33
 Topological, 351, 352
 Tracking method, 493
 Transition, 327, 328, 331, 337
 Translational symmetry, 562, 565
 Transverse shear, 183, 197, 318, 513
 Transverse shear stiffness, 78, 104

TRIA3, 389
TRIAR, 389
Two-dimensional, 178
Two-dimensional elements, 67, 127, 314

U

UACCEL, 438, 449
Units, 396
UNIX, 525, 526
UNSORT, 39
URACCEL, 449

V

Version, 550
Version number, 526
version-ID, 550, 551
VOLUME, 358, 423
von Mises stress, 9, 130, 431, 432
von Mises stress, see also Hencky-von Mises stress

W

Wallclock, 539
Warping, 78, 121
Wedge element, see also CPENTA
Weighting factors, 302
Wireframe, 372
Words, 545, 556

Y

Young's modulus (E), 396

Z

Zero-dimensional elements, 313
Zipper effect, 374