



MSC Nastran 2023.1

Linear Static Analysis User's Guide

Americas

5161 California Ave. Suite 200
University Research Park
Irvine, CA 92617
Telephone: (714) 540-8900
[Email: americas.contact@hexagon.com](mailto:americas.contact@hexagon.com)

Europe, Middle East, Africa

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

Japan

KANDA SQUARE 16F
2-2-1 Kanda Nishikicho, Chiyoda-ku
1-Chome, Shinjuku-Ku
Tokyo 101-0054, Japan
Telephone: (81)(3) 6275 0870
[Email: MSCJ.Market@hexagon.com](mailto:MSCJ.Market@hexagon.com)

Asia-Pacific

100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
[Email: APAC.Contact@hexagon.com](mailto:APAC.Contact@hexagon.com)

Worldwide Web

www.hexagon.com

Support

<https://simcompanion.hexagon.com>

Disclaimer

Hexagon 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. Hexagon 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 © 2023 Hexagon AB and/or its subsidiaries. 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 Hexagon is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from Hexagon suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at [third-party software information](#) (or successor website designated by Hexagon from time to time).

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

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

NASTRAN is a registered trademark of NASA. FLEXIm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. All other trademarks are the property of their respective owners.

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

U.S. Patent 9,361,413

March 16, 2023

NA:V2023.1:Z:Z:DC-LIN-PDF

Documentation Feedback

At Hexagon Manufacturing Intelligence, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, [write to us](#).

Please include the following information with your feedback:

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

You may also provide your feedback about Hexagon Manufacturing Intelligence documentation by taking a short 5-minute [survey](#).

Note:

The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact [Technical Support](#).

Contents

MSC Nastran Linear Static Analysis User's Guide

Preface

List of MSC Nastran Guides	11
Technical Support	12
Training and Internet Resources	13

1 Introduction

Introduction	16
Scope	16
Using the Finite Element Method.....	18
A Simple Problem.....	24

2 Organization of MSC Nastran Files

Introduction to Organization of MSC Nastran Files	32
Parameter Discussion	32
Overview of the MSC Nastran Input File	32
The NASTRAN Statement	35
File Management Section	35
Executive Control Section	35
Case Control Section	35
Bulk Data Section	36
Overview of the MSC Nastran Output Files	38
The Executive Control Section	42
The Case Control Section	44
POST Case Control Command	49
The Bulk Data Section	51

3 Grid Points, Scalar Points, and Coordinate Systems

Introduction to Degrees of Freedom	62
---	-----------



Grid Points	62
Scalar Points	63
Coordinate Systems	64
User-Defined Coordinate Systems	66
CORD1R and CORD2R	66
CORD1C and CORD2C	70
CORD1S and CORD2S	71
The Element and Material Coordinate System	74

4 MSC Nastran Elements

Introduction to MSC Nastran Elements	76
Scalar Elements	76
One-Dimensional Elements	76
Two-Dimensional Elements	77
Three-Dimensional Elements	78
GENEL Element	78
Connector Elements	78
Scalar Elements	79
The CELASI Elements	79
The BUSH Element	82
One-Dimensional Elements	90
The CROD Element	90
The CBAR Element	94
The CBEAM Element	115
Arbitrary Beam Cross-Section (ABCS)	134
Composite Beam - Variational Asymptotic Method (VAM)	140
The CBEND Element	146
CBEAM3 Three-Node Beam Element	150
Two-Dimensional Elements	154
The CQUAD4 and CTRIA3 Elements	155
The CQUAD8 and CTRIA6 Elements	166
Shell Element in the Material Coordinate System	167
The CQUADR and CTRIAR Elements	169
The Shear Panel Element (CSHEAR)	169
Two-Dimensional Crack Tip Element (CRAC2D)	172
Three-Dimensional Elements	175
The CHEXA, CPENTA, CPYRAM and CTETRA Elements	175
The CTRIAX6 Element	179
Three-Dimensional Crack Tip Element (CRAC3D)	180
GENEL Element	184
Connector Elements	188



CWELD Element	189
CAFAST Element	191
CSEAM Element	196
CINTC Element	198

5 Material Properties and Composites

Introduction to Material Properties	204
Isotropic Material (MAT1)	204
Two-Dimensional Anisotropic Material (MAT2)	205
Axisymmetric Solid Orthotropic Material (MAT3)	206
Two-Dimensional Orthotropic Material (MAT8)	207
Three-Dimensional Anisotropic Material (MAT9)	211
The PSHELL Entry	219
The Composite Element (PCOMP or PCOMPG)	220
Equivalent Composite Honeycomb Section	231

6 Constraints

Introduction to Constraints	236
Single-Point Constraints (SPC, SPC1)	236
Automatic Application of Single-Point Constraints (AUTOSPC)	238
Enforced Displacements at Grid Points (SPCD, SPC)	242
Multipoint Constraints (MPC)	243
Nonhomogeneous Multipoint Constraint (MPCY, MPCD)	246

7 Static Loads

Introduction to Static Loads	254
Loads on Grid Points and Scalar Points	254
Distributed Loads on Line Elements	259
PLOAD1	259
PLOADB3	265
Distributed Loads on Surfaces	266
Gravity and Centrifugal Force	271
Gravity (GRAV)	271
Centrifugal (RFORCE)	272
Set Selection of GRAV and RFORCE (GRAVSET, RFORSET)	275



GRAVSET	275
RFORSET	275
Acceleration Loads (ACCEL and ACCEL1)	277
Defining Mass (CMASSi and CONMi)	279
Initial Strains (DEFORM)	283
Thermal Loads (TEMP and TEMPD)	287
Load Combinations	292
Export of Static Loads	296
Fatigue Analysis/Output Request	298
Introduction	298
Fatigue Description	298
Overview of Case Control and Bulk Data:	300
Examples	301
STATSUB and BENDL Support in SOL101 Linear Statics	302
Introduction	302
Bar END Load.	302

8 R-Type Constraint Elements

Methods for Rigid Element Solution	306
Linear Method	306
Lagrange Method.	306
Method Selection	307
Introduction to R-Type Elements.	307
Description of the R-Type Elements	307
The RBAR Element	310
The RBE2 Element	316
The RBE3 Element	320
The RJOINT Element.	329

9 Modeling Guidelines

Introduction to Modeling Guidelines.	334
Choosing the Right Element	334
General Guidelines	335
Zero-Dimensional Elements	335
One-Dimensional Elements	335
Two-Dimensional Elements	335
Shell Normals.	338
Remarks and Recommendations	339



Three-Dimensional Elements	340
R-Type Elements	340
Mesh Density	340
h-elements.....	344
Mesh Transitions	344
Transition from a Coarse Mesh to a Fine Mesh	345
Mesh Transition Between Dissimilar Element Types	356
Shell-to-Solid Transition Element (RSSCON)	358
Grid Point Stresses.....	365
Description of the Grid Point Stress Method	367
Topological Method - Plate Elements and Solid Elements	369
Treatment of Exception Points	372
Grid Point Stress Interface	373
Consistent Loading.....	376
Symmetry	379

10 Model Verification

Introduction to Model Verification.....	386
Preprocessor Checks	386
Shrink Plot	386
Hidden Line/Shaded Plots.....	387
Free Edge/Face.....	388
Zipper Effect.....	390
CBAR/CBEAM Orientation and Offset Check	391
Duplicate Grid Points/Elements.....	391
Properties/Material Plots.....	391
Consistent Plate Normals	392
Strain Energy Output	394
Diagnostic Tools.....	398
Element Summary Output (ELSUM)	398
Element Geometry Checks (GEOMCHECK).....	401
Model Checking Material Properties (MODEL_CHECK).....	409
Weight Checks (PARAM,GRDPNT,WEIGHTCHECK)	412
Mechanisms and Singularities	414
Applied Loads Check	418
Reaction Load.....	420
1g Load	421
Equilibrium Checks	422
Output of Maximum, Minimum Grid Point-Based Responses.....	423
Unconstrained Equilibrium Check	428



Unintentional Grounding (GROUNDCHECK)	429
Thermal Equilibrium Check	431
Grid Point Force Balance	431
Stress Error Estimators	436
Grid Point Stress Discontinuities	437
Element Stress Discontinuities	441
Discussion of Error Measures	441
Postprocessor Checks	442
Monitor Points	447
Introduction	447
User Interface	448
Example 1	450
Example 2	452
Guidelines	453
Summary	454

11 Inertia Relief

Introduction to Inertia Relief	456
Guidelines for Inertia Relief	457
Inertia Relief Using PARAM,INREL,-1 (SUPPORT)	458
Inertia Relief Using PARAM,INREL,-2 (no Support)	467

12 Matrix Operations

Introduction to Matrix Operations	470
Set Definition	470
The Global Displacement Set	470
Subsets of the Global Displacement Set	471
Static Condensation (Guyan Reduction)	474
User Interface (ASET, OMIT)	476
Direct Matrix Input	482
DMIG	483
DMIG Bulk Data User Interface	484
DMIG Case Control User Interface	486
Stiffness Matrix Using DMIG	487
Mass Matrix Using DMIG	491
Load Matrix Using DMIG	491



13 Linear Buckling

Introduction to Linear Buckling	496
Finite Element Approach	496
Eigenvalue Extraction Method	501
Inverse Power Method (INV)	501
Enhanced Inverse Power Method (SINV)	502
Lanczos Method	502
Comparison of Methods	502
User Interface	502
MSC Nastran Linear Buckling with Local Exclusion of Differential Stiffness	504
Introduction	504
Benefits	504
Parameter	504
Example	505
Assumptions and Limitations of Linear Buckling Analysis	505
Examples	506
Example 1 -- Classical Euler Beam Buckling	506
Example 2 -- Lateral Buckling	511
Example 3 -- Planar Frame Buckling	516
Example 4 -- Buckling of a Stiffened Panel with Transverse Shear Flexibility	519
Example 5 -- Buckling of a Cylinder Under Uniform Axial Load	523
Example 6 -- Multiple Buckling Analyses in a Single Run	528

14 Restarts

Introduction to Restarts	532
Types of Restarts	532
Structure of the MSC Nastran Input File	533
User Interface for Read-Write Restart	533
Cold Start Run	533
Restart Run	534
Determining the Version for a Restart	539
Read-Only Restarts	540
Mini-Database Restarts	540
Miscellaneous Applications	541

15 Database Management

Introduction to Database Management	544
--	------------



Definitions	544
MSC Nastran Database	544
File Management Section	545
INIT	546
ASSIGN	547
EXPAND	549
RESTART	550
DBCLEAN	550
DBDIR	551
INCLUDE	552
Guidelines for Large Problems	552
BUFFSIZE	555

16 Cyclic Symmetry

Introduction to Cyclic Symmetry.....	558
Description of Geometric Symmetry in MSC Nastran.....	558
Mathematical Description of Structural Symmetry	564
Reflective Symmetry	564
Axisymmetry	567
Rotational Symmetry	569
Dihedral Symmetry	575
Bulk Data Entries for Cyclic Symmetry.....	579
CYJOIN, CYSYM, CYAX, and CYSUP Bulk Data Entries.....	580
LOADCYN, LOADCYH, and LOADCYT Bulk Data Entries	583
Case Control Section for Cyclic Symmetry.....	587
Axisymmetric Example	591

17 Contact in Linear Static Analysis

Introduction	598
Simplified Support Method	598
Example 1	601
Example 2	603
Contact in SOL 101 and Permanent Glued Contact	606
Contact in SOL 101	607
Permanent Glued Contact Modeling in SOL 101	608
Interface to General and Permanent Glued Contact	608
Guidelines and Limitations	615
Example 1: Permanent Glued Contact	618



18	Adaptive Meshing	
	Local Adaptive Mesh Refinement	622
	Introduction	622
19	Nastran Smart Superelement	
	Introduction to Smart Superelement	634
	Benefits	634
	Workflow	635
	The SSE Process and the User Interface	637
	EXTSEOUT Input File	637
	Linear Statics SSE Consumption in a MSC Nastran Assembly Run	638
	Guidelines and Limitation	638
	Example	638
	References	639
20	Grid Point Weight Generator	
	Introduction to Grid Point Weight Generation	642
	Commonly Used Features	642
	Example with Direction-Dependent Masses	644
21	Special Topics	
	Multiple Mass Configuration	656
A	Sample Output Listings	
	Examples List	660
	Glossary	
	References	





Preface

- List of MSC Nastran Guides 11
- Technical Support 12
- Training and Internet Resources 13



List of MSC Nastran Guides

A list of some of the MSC Nastran guides is as follows:

Installation and Release Guides
■ Installation and Operations Guide
■ Release Guide
Reference Guides
■ Quick Reference Guide
■ DMAP Programmer's Guide
■ Reference Guide
■ Utilities Guide
■ Getting Started Guide
■ SOL 400 Getting Started Guide
■ MSC Nastran Error Messages Guide
Demonstration Guides
■ Linear Analysis
■ Implicit Nonlinear (SOL 400)
■ Explicit Nonlinear (SOL 700)
■ MSC Nastran Verification Guide
User's Guides
■ Automated Component Modal Synthesis (ACMS)
■ Access Manual
■ Aeroelastic Analysis
■ Design Sensitivity and Optimization
■ DEMATD
■ Dynamic Analysis
■ Embedded Fatigue
■ Embedded Vibration Fatigue
■ Explicit Nonlinear (SOL 700)
■ High Performance Computing
■ Linear Static Analysis
■ Nonlinear (SOL 400)
■ Numerical Methods
■ Rotordynamics
■ Superelements and Modules
■ Thermal Analysis
■ User Defined Services



You may find any of these documents from Hexagon at:

<https://simcompanion.hexagon.com/customers/s/article/MSC-Nastran-Support-Home-Page>

Technical Support

For technical support phone numbers and contact information, please visit:

<https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304>

Support Center (<http://simcompanion.hexagon.com>)

The SimCompanion link above gives you access to the wealth of resources for Hexagon products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from Hexagon, allows you to manage your support cases, and participate in our discussion forums.

Training and Internet Resources

The Hexagon corporate site has the information on the latest events, products, and services for the CAD/CAE/CAM marketplace.

Design and Engineering e-Learning

The above link will point you to schedule and description of seminars. Following courses are recommended for beginning MSC Nastran users.

NAS120 - Linear Static Analysis using MSC Nastran and Patran

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC Nastran and Patran. MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran will be an integral part of the examples and workshops and will be used to generate and verify illustrative MSC Nastran models, manage analysis submission requests, and visualize results. This seminar provides the foundation required for intermediate and advanced MSC Nastran applications.





1

Introduction

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



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, example problems referenced in this guide are delivered with MSC Nastran system under the `MSC_DOC_DIR/doc/linstat` with the extensions of ".dat". It is recommended you copy any example problem to your local directory, so you can see the files created and you don't inadvertently create files in the delivery directory.

Problem sizes and computer resources have both been expanding rapidly over the years since NASTRAN was initially developed. What was large in the 1960's is tiny today. While the discussions in this guide of large problems, cpu, storage usage and restart capabilities may imply thousands of degrees of freedom, the reality of MSC Nastran today is 100's of million dof problems are possible. The discussions are still valid, just the scale has changed.

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, many have direct links.

This guide contains many highlighted links (in blue) to other MSC Nastran documents and all the documents were delivered together as a collection. If you keep the collection together the links between documents will work. Two suggestions when working with links are 1) "alt ← " returns you back in the window your mouse is in and 2) you can open the other "linked to" document in a new window from an Adobe Reader by choosing **Edit → Preferences → Documents → Open cross-document links in the same window**; then you would uncheck the and select "OK".

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.

[Introduction](#) (Ch. 1) discusses the Finite Element Method and shows a sample analysis. The following seven chapters will take you through choices and formats for building and loading a model.

[Organization of MSC Nastran Files](#) (Ch. 2) discusses the input and output files and sections of a MSC Nastran analysis through the use of small sample problems.



[Grid Points, Scalar Points, and Coordinate Systems](#) (Ch. 3) discusses the building blocks of Finite Elements, the Degrees of Freedom defined by Grid Points and Scalar Points and their coordinate systems.

[MSC Nastran Elements](#) (Ch. 4) discusses 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).

[Material Properties and Composites](#) (Ch. 5) discusses MSC Nastran support for isotropic, orthotropic and anisotropic materials.

[Constraints](#) (Ch. 6) discusses how to specifying the appropriate boundary conditions for your model to prevent any rigid body motion of your structure.

[Static Loads](#) (Ch. 7) discusses the static loads available when performing a linear static analysis

[R-Type Constraint Elements](#) (Ch. 8) discusses elements that imposes fixed constraints between components of motion at the grid points or scalar points to which they are connected.

[Modeling Guidelines](#) (Ch. 9) focuses on modeling guidelines and techniques such as element choice, density and transition.

[Model Verification](#) (Ch. 10) discusses the method used for model verification and how to improve the quality of a model.

[Inertia Relief](#) (Ch. 11) discusses a more advanced technique to allow simulation of unconstrained structures, which would normally be singular. This can be very useful for modeling aircraft in flight or satellites in space.

[Matrix Operations](#) (Ch. 12) discusses the matrix sets and operations involved in a structural analysis.

[Linear Buckling](#) (Ch. 13) 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.

The latter part of this guide focuses on modeling techniques and model verification. [Restarts](#) (Ch. 14) and [Database Management](#) (Ch. 15) 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.

[Cyclic Symmetry](#) (Ch. 16) has an expanded description on how to exploit symmetry in a structure to reduce problem size.

[Contact in Linear Static Analysis](#) (Ch. 17) discusses the use of linear and glued contact in MSC Nastran.

[Adaptive Meshing](#) (Ch. 18) discusses techniques of creating local area meshing to improve modeling ease and accuracy.

[Grid Point Weight Generator](#) (Ch. 20) is an in depth discussion of the grid point weight generator output.

[Sample Output Listings](#) (App. A) contains examples of most printed output formats in Linear Static Analysis.



Using the Finite Element Method

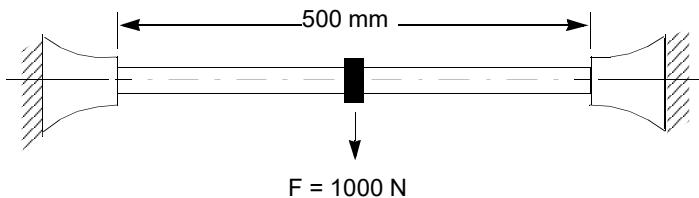
Before discussing the specifics of the finite element method using MSC Nastran, it is useful to examine how you might use the finite element method to solve an engineering problem. The discussion in this section is quite general and is not meant to show you the specifics of generating and running a finite 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 Dynamic Analysis 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.



Material - Aluminum
Tube OD - 20 mm
Tube ID - 19 mm



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](#).



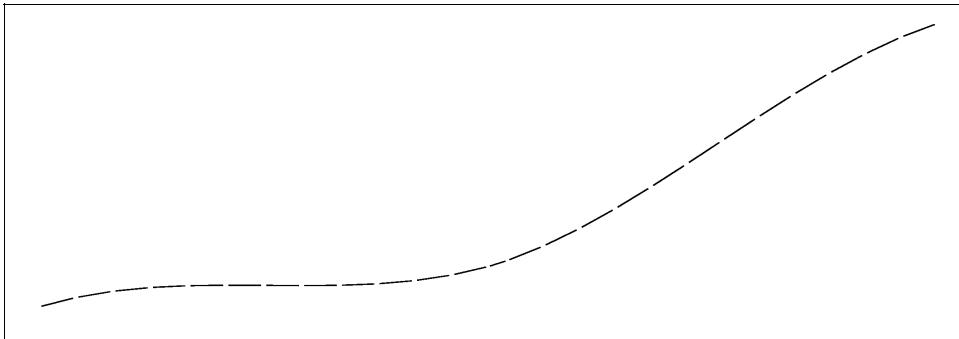


Figure 1-2 Deformed Shape of the Tube Structure Modeled with CBAR Elements

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/mm² 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.



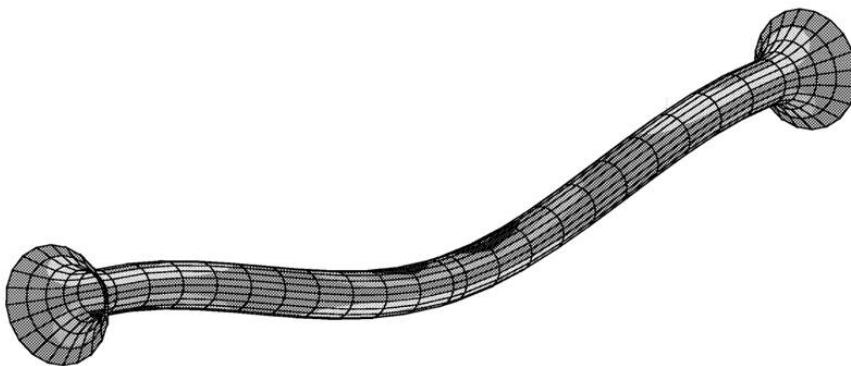


Figure 1-3 Deformed Shape of Coarse CQUAD4 Element Model

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 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.

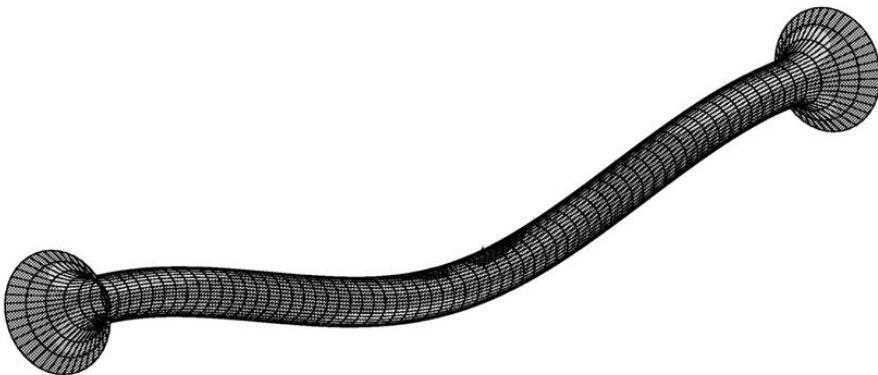


Figure 1-4 Deformed Shape of Fine CQUAD4 Element Model

[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](#).

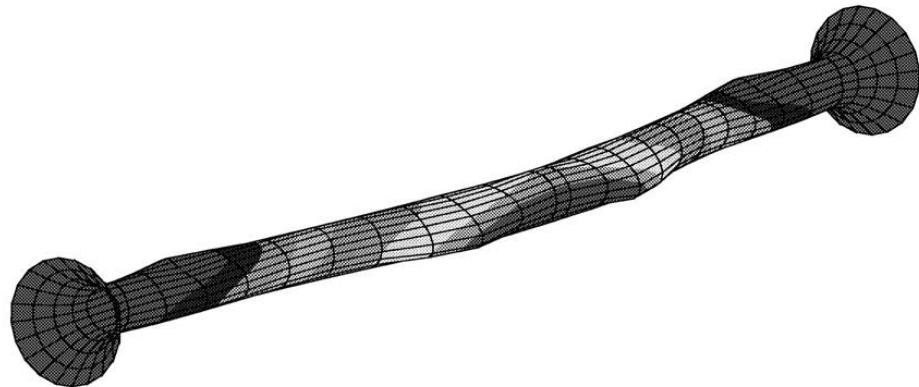


Figure 1-5 Buckled Shape Using the Coarse CQUAD4 Model

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](#).



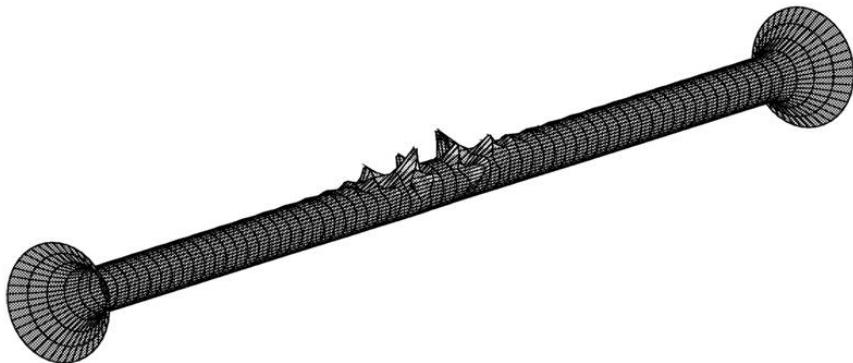


Figure 1-6 Buckled Shape Using the Fine CQUAD4 Model

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 [CHAPTER](#).

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.

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 [Equation \(1-1\)](#) is solved as follows:

$$[K]\{u\} = \{p\} \quad (1-1)$$

where:

$[K]$ = system stiffness

$\{u\}$ = grid point displacements

$\{p\}$ = applied load vector

The unknowns in [Equation \(1-1\)](#) 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.



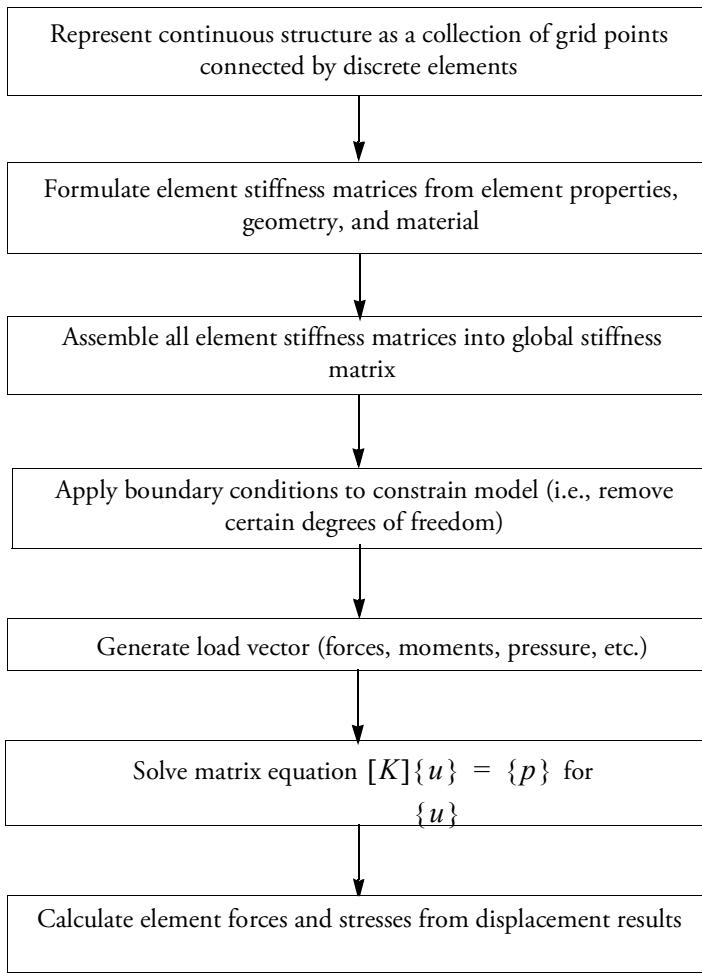


Figure 1-7 Linear Static Analysis in MSC Nastran

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.



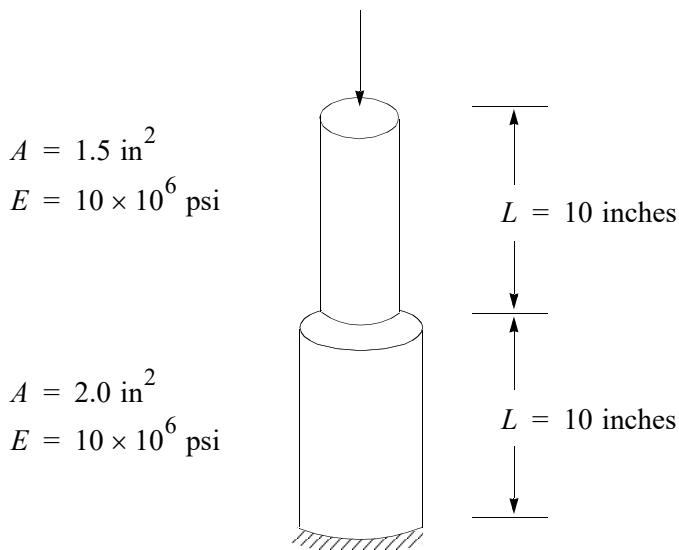


Figure 1-8 Two-Member Structure Under Axial Load

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](#).

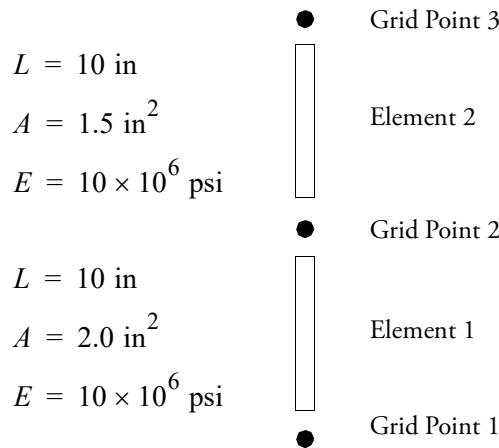


Figure 1-9 Exploded View of the Finite Element Model

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 [Equation \(1-2\)](#).

$$K_{el} = \begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \quad (1-2)$$

Using the cross-sectional area A , Young's modulus E , and the length L for each element in [Equation \(1-2\)](#), the element stiffness matrices are given by

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

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 [Equation \(1-4\)](#).

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

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

4. Apply the fixed boundary condition to grid point T, 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 [Equation \(1-5\)](#). Partitioning the matrix is achieved by removing rows and columns from one matrix to create a smaller matrix.

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

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 [Equation \(1-6\)](#).

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

6. Solve the matrix equation.

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

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

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

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 [Equation \(1-8\)](#).

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

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

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



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

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

(1-9)

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

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

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. (See MSC_DOC_DIR/doc/linstat/intro.dat.)



2

Organization of MSC Nastran Files

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



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 files can be found on the delivery media, in the `MSC_DOC_DIR/doc/linstat` directory.

Parameter Discussion

Parameters are used throughout MSC Nastran to provide the user with controls over many aspects of the solution. They are powerful tools for setting corporate, section, or user defaults. Each solution sequence is designed with a specific set of default values and solution paths. In the majority of the cases, the default values have been found well suited for most models. There are occasions, however, when you may want to deviate from the default values.

There are multiple ways of setting parameters for MSC Nastran. For the NASTRAN Statement they are set by the `rc` file and the execution statement. (See the [Configuring the Runtime Environment](#) (Ch. A) in the *MSC Nastran Installation and Operations Guide* and the [\(Ch. 1\) in the MSC Nastran Quick Reference Guide](#)). For the Case Control and Bulk Data Section the selection is extensive and controls for a parameter can come from each section. (See [Parameters](#) in the *MSC Nastran Quick Reference Guide*). For the advanced users, there is also the DMAP (Direct Matrix Abstraction Program) module language. (See the *MSC Nastran DMAP Programmer's Guide*)

The majority of the parameters referenced in this guide are Case Control or Bulk Data parameters. These will be the ones controlling default tolerances or controls for the type and amount of output. Depending on the parameter, it can be specified either as a PARAM Bulk Data entry and/or a PARAM Case Control command. (See [Parameter Applicability Tables](#) in the *MSC Nastran Quick Reference Guide*) The format of each is as follows: `PARAM,xname,xvalue` where `xname` and `xvalue` are the name and value of the PARAMeter, respectively, such as `PARAM,WTMASS,0.00259`, for the terms of the structural mass matrix to be multiplied by the value of this parameter when they are generated. Or `PARAM,POST,-1` to output data for the Patran Post Processor.

Overview of the MSC Nastran Input File

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](#).

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

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

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

$$\nu = 0.3$$

○ Elements

□ Grid points

● Pin joints

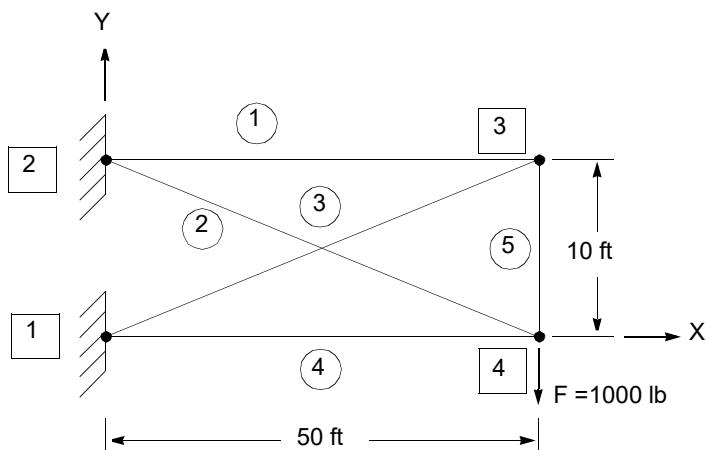


Figure 2-1 Truss Example

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](#). (See `MSC_DOC_DIR/doc/linstat/truss1.dat`.) 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](#).



Listing 2-1 Truss Example Input File

<pre> \$ FILENAME - TRUSS1.DAT \$ ID LINEAR,TRUSS1 SOL 101 TIME 2 CEND </pre>	Executive Control Section
<pre> 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 </pre>	Case Control Section
<pre> 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>	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.

Following are descriptions of these sections.

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 BUFSIZE for the I/O. For a description of the NASTRAN statement options, refer to the [Executing MSC Nastran](#) in the *MSC Nastran Quick Reference Guide*. Some of the NASTRAN statement options are discussed in [Database Management](#).

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](#) and [Database Management](#) 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 (Case) = ALL**, **FORCE (Case) = ALL**, and **STRESS (Case) = ALL** commands. By default, these output requests are printed to the .f06 output file, which is discussed in [Overview of the MSC Nastran Input File](#). 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 selected 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.



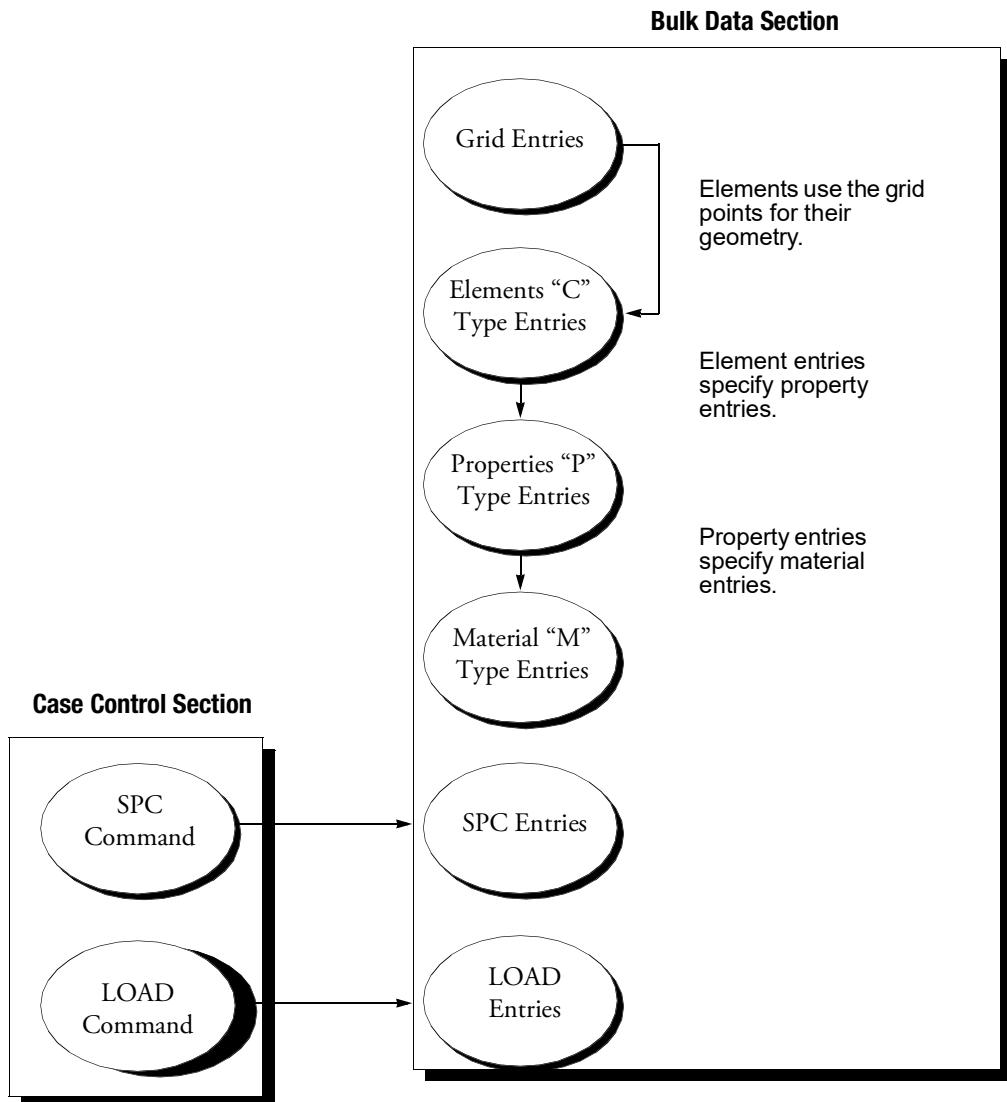


Figure 2-2 MSC Nastran Input File Structure

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 in², and the torsional stiffness coefficient of 1.27 in⁴. 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](#); 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. See [MSC_DOC_DIR/doc/linstat/truss1.dat](#).

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. [Table 2-1](#) provides summary descriptions of all the potential files generated by MSC Nastran (assuming 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.



Table 2-1 Files Resulting from Submitting the Job Named run1.dat

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.IPPDAT	The file containing the Bulk Data entries for IFPStar restart processing (SYSTEM(444)=1)
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 Patran for postprocessing of the results.
run1.h5	Results database in HDF5 format.
run1.sts	Contains nonlinear statistics, if Linear Contact is utilized.
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. Many of the .f06 output formats are shown in the [Sample Output Listings](#) (App. A) with notes, so it is a good referenced for understanding your output requests.

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 Patran graphics pre- and postprocessors. It is requested by a PARAM,POST entry in the Bulk Data Section. The binary file run1.h5 is the results database in HDF5 format that is requested using the MDLPRM, HDF5,1 bulk data entry.

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](#), and [Reference 18](#).

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 might want to 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. Without deleting, these files are reused and can increase in size or have multiple copies created.



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 (minus the page numbers and date) 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](#).



```

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

ID LINEAR,TRUSS1
SOL 101
TIME 2
CEND

C A S E   C O N T R O L   E C H O
COMMAND
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 ENTRY COUNT =      33

      S O R T E D   B U L K   D A T A   E C H O
ENTRY
COUNT    . 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 ..
1-     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

      M O D E L   S U M M A R Y
NUMBER OF GRID   POINTS =      4
NUMBER OF CROD   ELEMENTS =      5

*** USER INFORMATION MESSAGE 7310 (VECPRN)
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
      OLOAD   RESULTANT

SUBCASE/
DAREA ID   LOAD   T1   T2   T3   R1   R2   R3
1   FX   0.000000E+00   ----   ----   0.000000E+00   0.000000E+00
      FY   ----   -1.000000E+03   ----   0.000000E+00   0.000000E+00   -6.000000E+05
      FZ   ----   ----   0.000000E+00   0.000000E+00   0.000000E+00   ----
      MX   ----   ----   ----   0.000000E+00   ----   ----
      MY   ----   ----   ----   ----   0.000000E+00   ----
      MZ   ----   ----   ----   ----   ----   0.000000E+00
TOTALS   0.000000E+00   -1.000000E+03   0.000000E+00   0.000000E+00   0.000000E+00   -6.000000E+05

*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)
THE DECOMPOSITION OF KLL  YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF  1.300101E+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   0.000000E+00   6.4518387E+01

      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   1.247578E-02   -1.285377E-01   0.0   0.0   0.0   0.0
4   G   -1.252422E-02   -1.290368E-01   0.0   0.0   0.0   0.0

      F O R C E S   I N   R O D   E L E M E N T S   ( C R O D )
ELEMENT   AXIAL   ELEMENT   AXIAL
ID.   FORCE   TORQUE   ID.   FORCE   TORQUE
1   2.495156E+03   0.0   2   2.554449E+03   0.0
3   -2.544570E+03   0.0   4   -2.504844E+03   0.0
5   4.990313E+02   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   AXIAL   SAFETY   TORSIONAL   SAFETY
ID.   STRESS   MARGIN   STRESS   MARGIN   ELEMENT   AXIAL   SAFETY
1   6.237891E+02   0.0   2   6.386123E+02   0.0
3   -6.361426E+02   0.0   4   -6.262109E+02   0.0
5   1.247578E+02   0.0

```



Figure 2-3 truss1.f06 Output File.

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](#).

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 $1/2 \{P_l\}^T [U_l]$. 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](#).

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 can be used to help identify a job; this statement is optional.

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 or name of the solution sequence. Linear static analysis--the main subject of this guide--is the Solution Sequence, SOL 101 or SOL SESTATIC. The Solution Sequences are numbered in the range of 101 through 400. Linear static analysis can also be selected with the SOL 400 statement and the addition of the Case Control command ANALYSIS=STATICS. For this guide the examples will use SOL 101.

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 to 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.

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 statement **DIAG** in the *MSC Nastran Quick Reference Guide* 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](#) in the *MSC Nastran Quick Reference Guide*.

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 \(Case\)](#) 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 Input File](#), a single subcase with ID 1 is assumed.

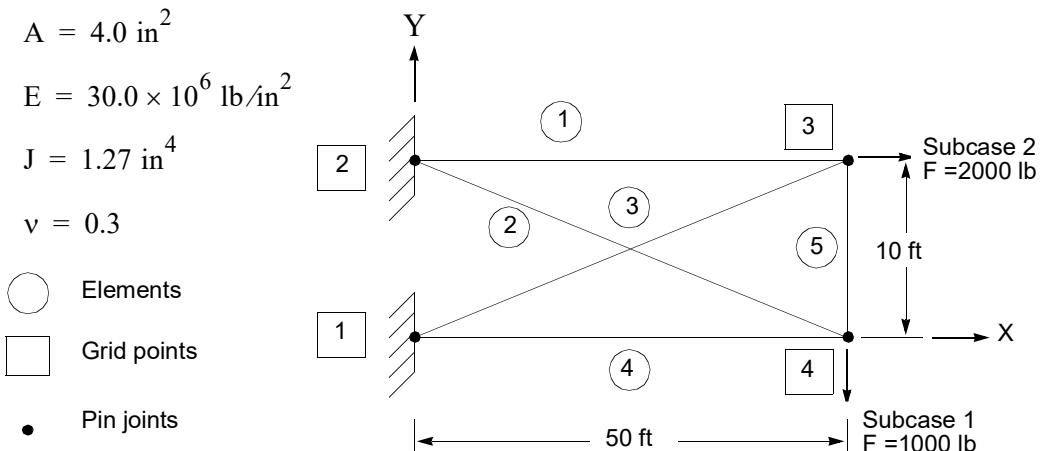


Figure 2-4 Truss Example with Two Loading Conditions.

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.



The Case Control Section up to the Bulk Data Section, meeting these requirements, is shown in [Listing 2-2](#). (See MSC_DOC_DIR/doc/linstat/truss2.dat)

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
$ Same as truss1.dat shown earlier
$
```

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 \(Case\)](#) 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](#).

(See MSC_DOC_DIR/doc/linstat/truss3.dat.)

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 [Listing 2-2](#). Suppose you want to separate the results into three separate sections in the .f06 output file. The case control input file is shown in [Listing 2-4](#). (See `MSC_DOC_DIR/doc/linstat/truss4.dat`.) 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 [Listing 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 case control input file shown in [Listing 2-5](#) demonstrates this feature.
(See `MSC_DOC_DIR/doc/linstat/truss5.dat`.)

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 input file shown in [Listing 2-5](#), the SUBCOM 3 output is as follows:



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

The output for SUBCOM 4 is as follows:

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

The coefficients 1.0, and 0.5 for Subcom 3 are input on a SUBSEQ or SUBSEQ1 command. These coefficients used in the SUBSEQ or SUBSEQ1 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.

The alternative is to use the SUBSEQ1 command which is easier to use, especially if one has a large number of subcases. In which case, you only need to enter the SUBCASE ID and the factor.

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 \(Case\)](#) 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.

POST Case Control Command

The [POST \(Case\)](#) Case Control command provides some subcase-level control over the amount of output data stored on the postprocessing file.

Finite element analysis of large structural components often have many different loading scenarios in the simulation, possibly with several thousand different load cases. During this process, recovery and postprocessing of large amounts of response data usually occurs. Many analysis organizations use commercially available pre and postprocessors such as Patran to graphically display results data. These programs typically read the results data from a FORTRAN file generated by the OUTPUT2 module in MSC Nastran. This file, by default, contains all of the output requested by the case control data recovery requests in each and every subcase. The POST Command can eliminate the output data for a particular subcase from the data file that is not needed by the postprocessor.

The results output from large simulations can quickly consume enormous amounts of computer disk storage space. Not only does MSC Nastran store the output for use by the postprocessing program, the postprocessor itself can use large quantities of disk space when it creates different views of the data for use in a graphical display environment. If the postprocessor program does not require results from one (or more) subcase(s), both computer time and disk space are wasted reading and storing the unnecessary data. Use of the POST Case Control command can be used to eliminate the unwanted results data from the OUTPUT2 file. This can substantially reduce the file processing time and disk space used by the postprocessor.

Another benefit of the POST Case Control command is that it allows MSC Nastran to write data into output files by specifying their file names instead of FORTRAN unit reference numbers. This capability removes the limitation imposed by the MSC Nastran file management system on the number of ASSIGN statements. The user is now able to write solution data into as many output files, as desired.

The amount of data recovered for each subcase is dependent upon the Case Control commands. If a request is placed above all subcases, that request applies to all of the subcases. Data are generated and produced one time for all subcases by examining the case control requests in each subcase. Each separate type of response is stored in its own data block. For example, displacement results are computed and stored separately from element stress results. These data blocks will include results for any subcase that contains a data recovery request with a print, plot or punch destination. The data blocks are then written to an external file by the OUTPUT2 module, unless the POST Command is used to eliminate specific Data requests.

The POST Case Control command provides control over the contents of the output file produced by the MSC Nastran program when a PARAM,[POST](#) Bulk Data entry is present in the input. It does not affect the



contents of the results data blocks, only the amount of data that is transferred from those data blocks to the output file. The POST command is examined for the output requested in each subcase. Only that output requested by the POST command is placed on the output file for the subcase. Note that the data must be made available by the presence of a data recovery command in the subcase. For example, the POST command cannot output stress data if no stress data have been requested.

The general format of the POST command is:

```
POST { TOFILE } { furn } [ppname][oplist]
      { TOCASE } { filename }
```

Example

This sample shows the usage of the POST Case Control command to eliminate unwanted output from being placed on the postprocessor data file generated by MSC Nastran. The model data is not important for this example. For example, consider the case of a static simulation performed on a model requiring three load cases. The required output is displacements at three grid points in the model. Furthermore, the output for the second load case must also include punched displacement data for all grid points. Displacement results for subcases 100 and 300 are to be placed on the default OUTPUT2 file for use by the Patran postprocessor. No output for subcase 200 is required for postprocessing by Patran. The subcase structure might look something like:

```
SET 1000 = 10, 11, 12
DISP = 1000
$
POSTTofile 12 $ use defaults: Patran
$
SUBCASE 100
LOAD = 100
$
SUBCASE 200
LOAD = 200
DISP(PUNCH) = ALL
POST NODISP $ stop any displacement output from going to POST file
$
SUBCASE 300
LOAD = 300
```

In this example, the presence of the POST command above all of the subcases indicates that all output requested (DISP for SET 1000 is the only output requested and generated) is to be stored on the default OUTPUT2 fortran unit for use by Patran. The POST command in SUBCASE 200 with the NODISP option prevents the large output produced by the DISP(PUNCH)=ALL request in SUBCASE 200 from being placed on the OUTPUT2 file.

A complete description of all of the Case Control commands is available in the [MSC Nastran Quick Reference Guide](#). See the [Case Control Applicability Tables](#) in the *MSC Nastran Quick Reference Guide* for commands used in linear static analysis.



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, 531](#).

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](#).

1	2	3	4	5	6	7	8	9	10
8	8	8	8	8	8	8	8	8	8



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; in general, 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. As long as less than 80 columns are used, the free field format can have more than 8 data fields and connections will be automatic.



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 generally 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](#).



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.

The majority of 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.



3

Grid Points, Scalar Points, and Coordinate Systems

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



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 is devoted to the several types of coordinate systems available in MSC Nastran.

Grid Points

The Bulk Data entry **GRID** 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:

GRID

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 u₁, u₂, u₃, θ₁, θ₂ and θ₃ or as T₁, T₂, T₃, R₁, R₂, and R₃. T₁, T₂, and T₃ are the three orthogonal components of translation parallel to the 1, 2, and 3 directions of the grid point's displacement coordinate system, respectively. R₁, R₂, and R₃ 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](#). 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. Note, the units of translation are the same as the units of length of the model and the rotations are in units of radians.

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](#).

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 Bulk Data entry **SPOINT** in the *MSC Nastran Quick Reference Guide* 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](#).

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](#), 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](#), 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.

To aid the output interpretation, the user defined output coordinate system ID, on field 7 (CD) of GRID entry, can be printed along with the responses for most vector requests by adding CID, such as DISP(CID)=ALL to the request. The CID will only be printed in the .f06 output file, all other output files are unchanged. A CID keyword in the output request(s) need(s) only to appear once in the Case Control Section. It will affect ALL grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce

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](#). 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

Defining a rectangular coordinate with the Bulk Data entry [CORD1R](#) in the *MSC Nastran Quick Reference Guide* 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	G1B	G2B	G3B	

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

The Bulk Data entry [CORD2R](#) in the *MSC Nastran Quick Reference Guide* 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.



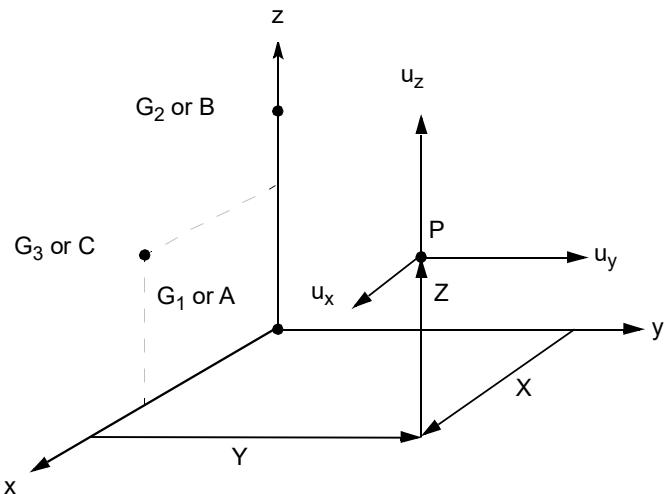


Figure 3-1 CORD1R and CORD2R Definitions

To better understand the use of a CORD1R entry, consider the truss structure introduced in [Organization of MSC Nastran Files](#). 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

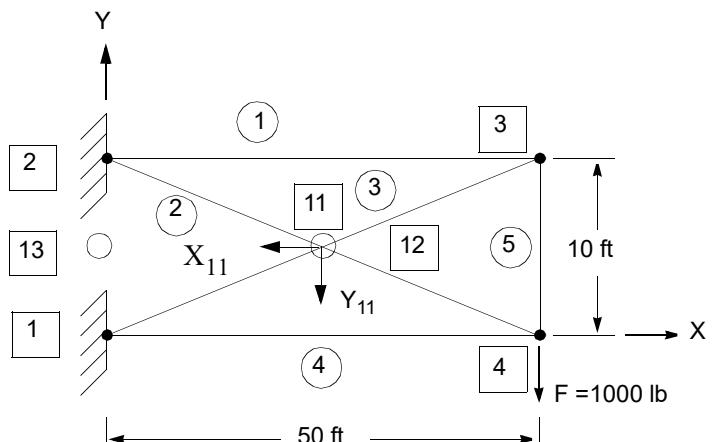


Figure 3-2 Grid Point Locations Using a Local Coordinate System

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.

Part of the corresponding input file for the structure is given in [Listing 3-1](#). (See [MSC_DOC_DIR/doc/linstat/truss6.dat](#)) 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  

$
```

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 [Listing 2-2](#). Since the CD field of grid points 1, 2, 3, and 4 are the same in both files, the displacement 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.

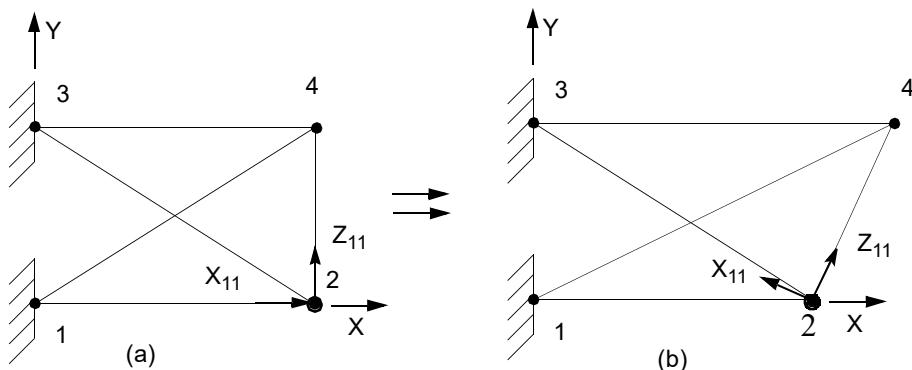


Figure 3-3 **Model Change**

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

The Bulk Data entry [CORD1C](#) in the *MSC Nastran Quick Reference Guide* 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 Bulk Data entry **CORD2C** in the *MSC Nastran Quick Reference Guide* 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.

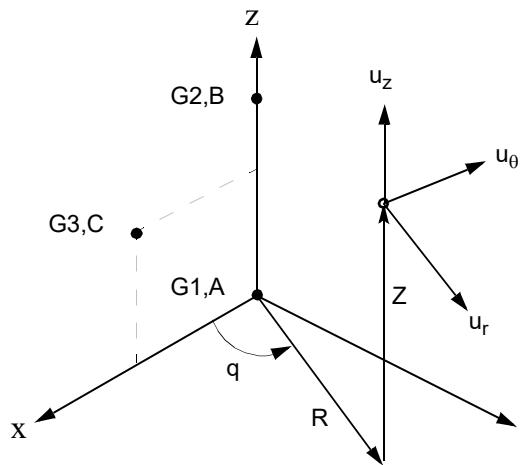


Figure 3-4

CORD1C and CORD2C Definitions

CORD1S and CORD2S

The Bulk Data entry **CORD1S** in the *MSC Nastran Quick Reference Guide* 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 Bulk Data entry **CORD2S** in the *MSC Nastran Quick Reference Guide* 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.



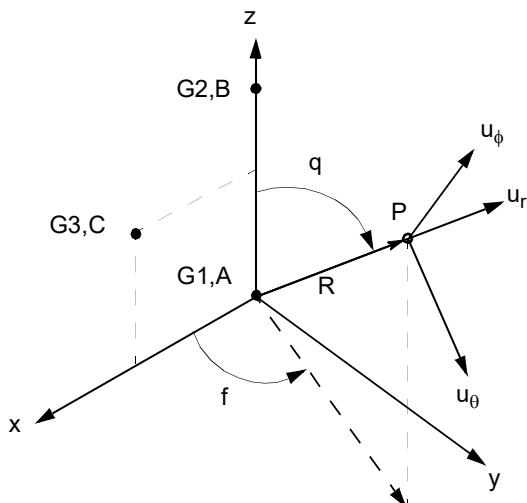


Figure 3-5 CORD1S and CORD2S Definitions

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

Example

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.

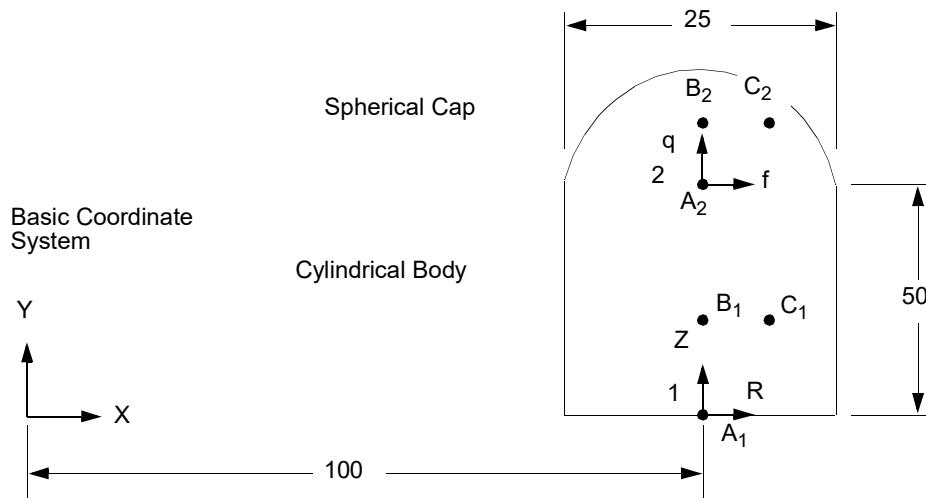


Figure 3-6 Silo Model



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 A₁, 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 A₁ to point B₁, which is located at (100,1,0). The Z-axis along with point C₁, 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 A₂ at (0,0,50). This location is with respect to coordinate system 1. The $\theta = 0$ axis is the line extending from A₂ to B₂, which is located at (0,0,51). Finally, the $\phi = 0$ plane is located by the $\phi = 0$ axis and point C₂, 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](#).

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](#) for a discussion of material coordinate systems.



4

MSC Nastran Elements

- Introduction to MSC Nastran Elements
- Scalar Elements
- One-Dimensional Elements
- Two-Dimensional Elements
- Three-Dimensional Elements
- GENEL Element
- Connector Elements



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), three-dimensional (3-D), general and connectors. Before discussing the elements, it is useful to define each of the 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](#) and [Grid Points, Scalar Points, and Coordinate Systems](#), 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

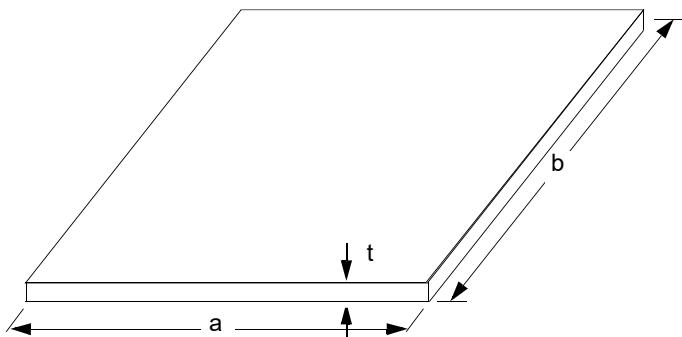
A one-dimensional element is one in which the properties of the element are defined along a line or curve. Typical applications for the one-dimensional element include truss structures, beams, stiffeners, and many others. In fact, the truss structure used in [Organization of MSC Nastran Files](#) and [Grid Points, Scalar Points, and Coordinate Systems](#) 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.
- CBEAM3 - A three noded curved element for modeling with higher order elements.



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 in [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, CPYRAM and CTETRA - General-purpose solid elements. This family of elements is recommended for most solid model applications.
- The CTRIAxis6 - 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.

GENEL Element

The GENEL element is a way to input our own element, usually based on test data. It 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 used to describe a substructure that has an arbitrary number of connection grid points or scalar points.

Connector Elements

MSC Nastran offers 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. The Connector family of elements enables you to connect surfaces with differing mesh densities and using a subset of spot weld elements you can connect more than one element per surface. The Connector elements are general in purpose and always satisfies the condition of rigid body invariance. The Connector elements discussed in this chapter include:

- CWELD - which defines a weld or fastener connecting two surface patches or points
- CFAST - which defines a fastener with material orientation connecting two surface patches
- CSEAM -which defines a seam line to connect two surface patches
- CINCT - which connects dissimilar meshes along the edges of finite element mesh subdomains

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](#).



Scalar Elements

The CELASi 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 (CELAS*i*, *i* = 1–4) and concentrated masses (CMASS*i*, *i* = 1–4) are useful. There are four types of scalar springs and mass definitions. The formats of the CELAS*i* entries, i.e., **CELAS1** in the *MSC Nastran Quick Reference Guide* (elastic springs) are as follows:

CELAS*i*

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](#). 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:

PELAS

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

Field	Contents
PID _i	Property identification number.
K _i	Elastic property value.
G _{Ei}	Damping coefficient.
S _i	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.

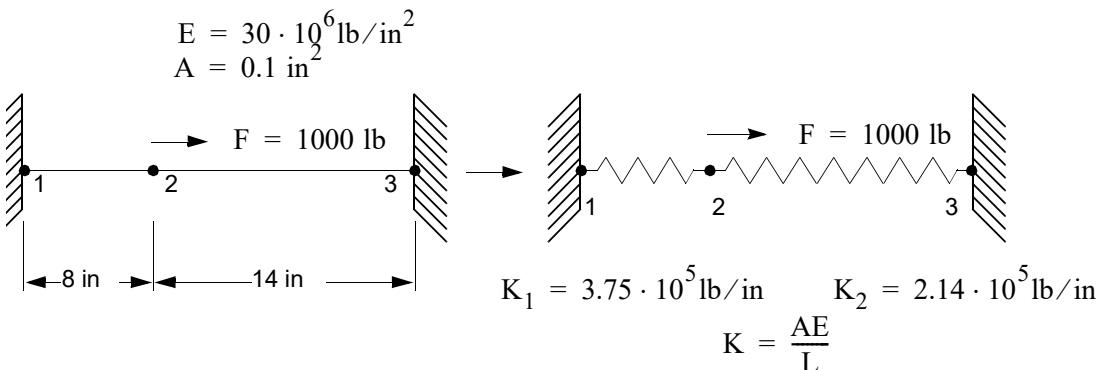


Figure 4-2 Equivalent Spring Model

The input file for the structure shown is given in [Listing 4-1](#). (See MSC_DOC_DIR/doc/linstat/spring1.dat)

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](#).

DISPLACEMENT VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	S	0.0	1.697793E-03	0.0					
<hr/>									
ELEMENT ID.	FORCE	FORCES IN	SCALAR FORCE	SPRINGS FORCE	(ELEMENT ID.)	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 [Equation \(4-1\)](#).

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

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 [Equation \(4-1\)](#).

The same structure modeled with two CROD elements is in the input file rod1.dat. (See [MSC_DOC_DIR/doc/linstat/rod1.dat](#))

The BUSH Element

The CBUSH element is a generalized spring-damper element. Unlike the CELASi, elements the CBUSH is a structural scalar element. It connects two non coincident grid points, or two coincident grid points or one grid point and has an associated PBUSH entry. Also if, CELASi elements are used and the geometry is not aligned properly, internal constraints may be induced. The CBUSH contains all the features of the CELASi elements plus avoiding the internal constraint problem.

The format of the Bulk Data entry [CBUSH](#) is as follows:

CBUSH

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example 1: Noncoincident grid points.

CBUSH	39	6	1	100	75				
-------	----	---	---	-----	----	--	--	--	--

Example 2: GB not specified.

CBUSH	39	6	1					0	
-------	----	---	---	--	--	--	--	---	--

Example 3: Coincident grid points (GA ≠ GB).



CBUSH	39	6	1					6	
-------	----	---	---	--	--	--	--	---	--

Example 4: Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Field	Contents
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PBUSH entry. ($\text{Integer} > 0$; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark 6. ($\text{Integer} > 0$)
Xi	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector \vec{v} using grid point GO. Direction of \vec{v} is from GA to GO. \vec{v} is then transferred to End A. See Remark 3. ($\text{Integer} > 0$)
CID	Element coordinate system identification. An 0 means the basic coordinate system. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 4-4 and Remark 3. ($\text{Integer} \geq 0$ or blank)
S	Location of spring damper. See Figure 4-4. ($0.0 \leq \text{Real} \leq 1.0$; Default = 0.5)
OCID	Coordinate system identification of spring-damper offset. See Remark 9. ($\text{Integer} \geq -1$; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 4-4)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0 . See Figure 4-5 and Remark 9. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Figure 4-4 shows the bush element geometry.
3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis.
4. For noncoincident grids (GA \neq GB), when GO or (X1, X2, X3) is given and no CID is specified, the line AB is the element x-axis and the orientation vector \vec{v} lies in the x-y plane (similar to the CBEAM element).



5. For noncoincident grids ($GA \neq GB$), if neither GO or $(X1, X2, X3)$ is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is grounded.
7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.
8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0 , then S is ignored and S1, S2, S3 are used.

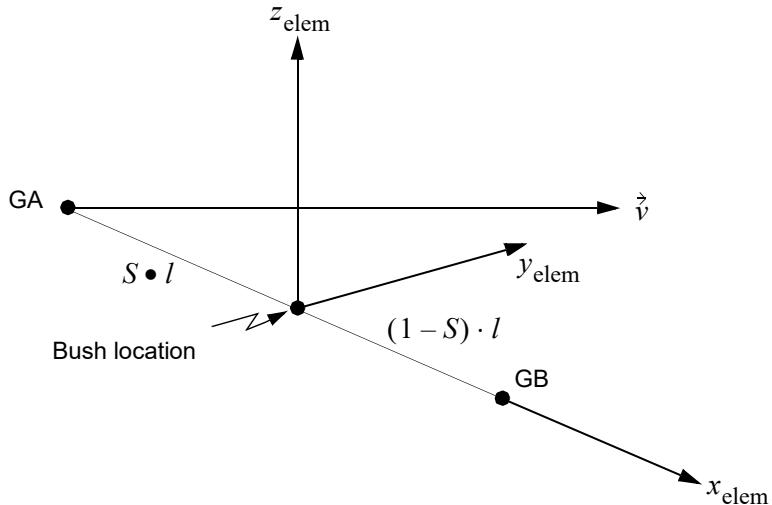


Figure 4-4 CBUSH Element



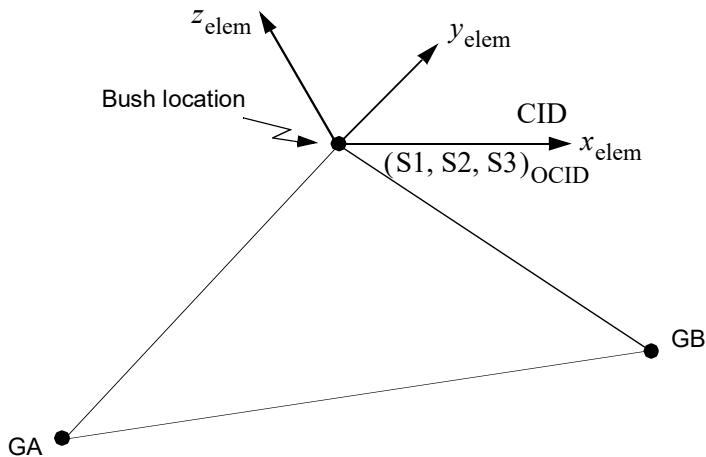


Figure 4-5 Definition of Offset S1, S2, S3

10. When $\text{CID} \geq 0$, the element x-axis is set as in Remark 3. This means that the element force is always computed as $K_e . (UB - UA)$; if $UA > UB$, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.
11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.

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

PBUSH

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	"K"	K1	K2	K3	K4	K5	K6	
		"B"	B1	B2	B3	B4	B5	B6	
		"GE"	GE1	GE2	GE3	GE4	GE5	GE6	
		"RCV"	SA	ST	EA	ET			

Example 1:

Stiffness and structural damping are specified.

PBUSH	35	K	4.35	2.4				3.1	
-------	----	---	------	-----	--	--	--	-----	--



		GE	.06						
		RCV	7.3	3.3					

Example 2:

Damping force per unit velocity are specified.

PBUSH	35	B	2.3						
-------	----	---	-----	--	--	--	--	--	--

Field	Contents
PID	Property identification number. (Integer > 0)
"K"	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
"B"	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks 2., 3., and 9. (Real; Default = 0.0)
"GE"	Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
"RCV"	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. (Character)
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)

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.

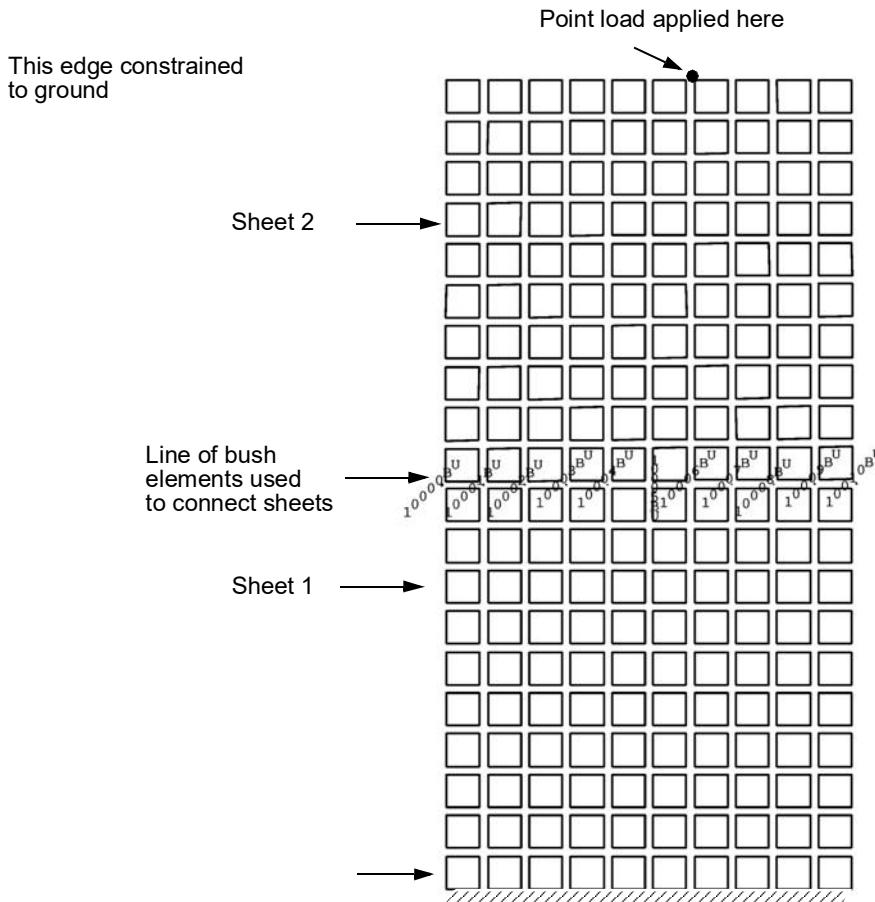
The CBUSH elements are easier to model since they connect grid points instead of grid point components. To illustrate the difference, the model shown in [Figure 4-6](#) (see [MSC_DOC_DIR/doc/linstat/bushweld.dat](#)) has two sheets of metal modeled with CQUAD4 elements. The sheets are placed next to each other. There are grid points at the common boundary of each sheet of metal, which are to be joined together. 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.

The boundary is modeled and loaded on the CELAS elements in the first subcase and on the CBUSH elements in the second subcase.

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 CELAS model and the second loading condition loads only the CBUSH model, allowing comparison of the response for both models in one combined analysis.



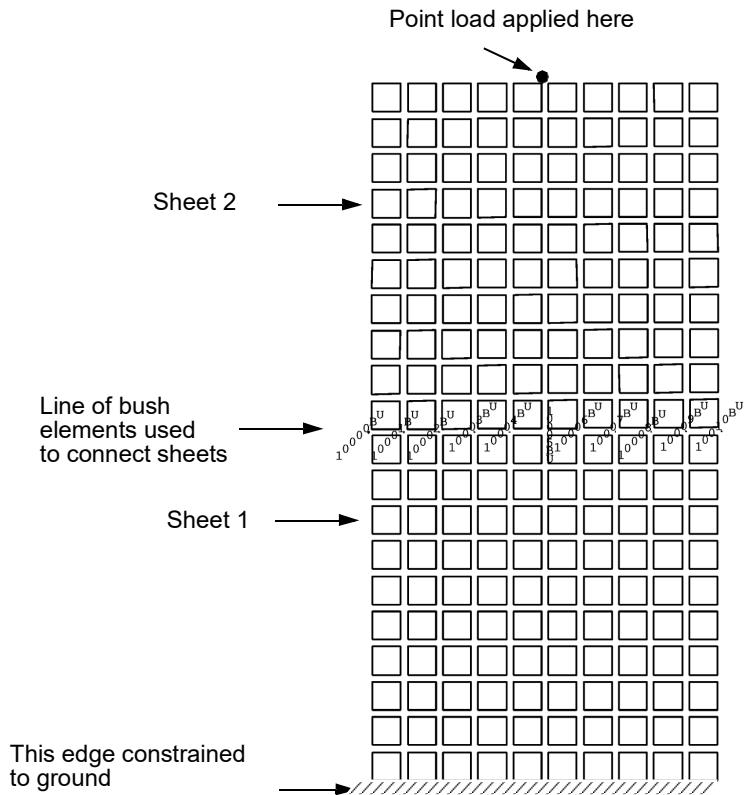


Figure 4-6 CELAS2 vs CBUSH Comparison Model

In order to use CELAS_i 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 CELAS_i 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 subcase. 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.

The input entries for a weld are shown for each modeling method in [Listing 4-2](#). 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 4-2 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,,,,
PBUSH,1,K1.+6,1.+61.+61.+61.+6
PARAM,EST,1 $ PRINT THE MEASURE OF ALL ELEMENTS
```

The use of PARAM,EST,YES outputs 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.

The OLOAD resultants and the SPC-force resultants for the two models (Figure 4-7) 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							
	T1	T2	T3	OLOAD	RESULTANT	R1	R2
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	
.							
	T1	T2	T3	SPCFORCE	RESULTANT	R1	R2
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 4-7 Output of Load and SPC-Force Resultants

The resultants in the CBUSH 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.

The CELASi, CBUSH, CINTC and permanent glue methods could all be used for this problem, but the CBUSH and permanent glue are the recommended methods.



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 Bulk Data entry [CROD](#) in the *MSC Nastran Quick Reference Guide* 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 Bulk Data entry [PROD](#) in the *MSC Nastran Quick Reference Guide* 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.



Field	Contents
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 elements 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](#) 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 4-8](#).

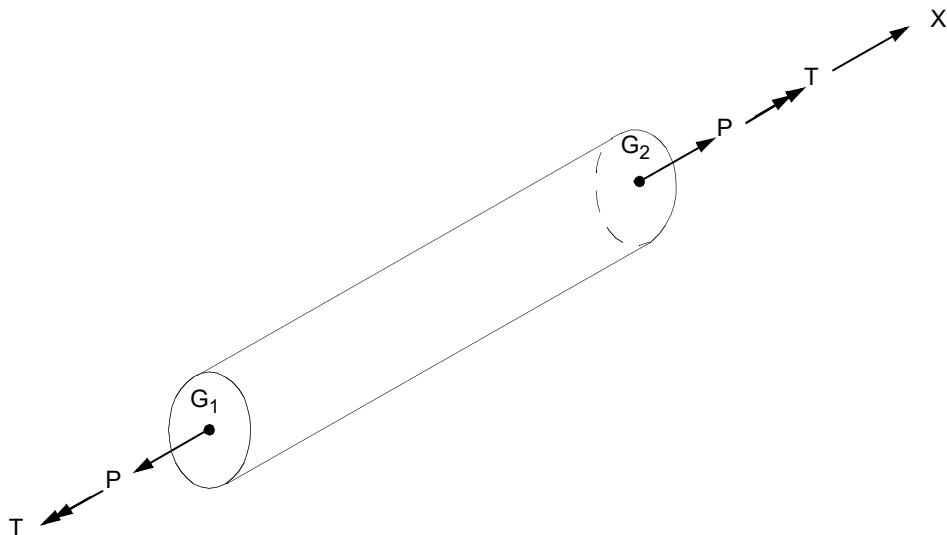


Figure 4-8 CROD Element Internal Forces and Moments

To illustrate the use of the CROD element, consider a three-member truss structure attached to a rigid wall as shown in [Figure 4-9](#). The input file is shown in [Listing 4-2](#). (See MSC_DOC_DIR/doc/linstat/rod2.dat)

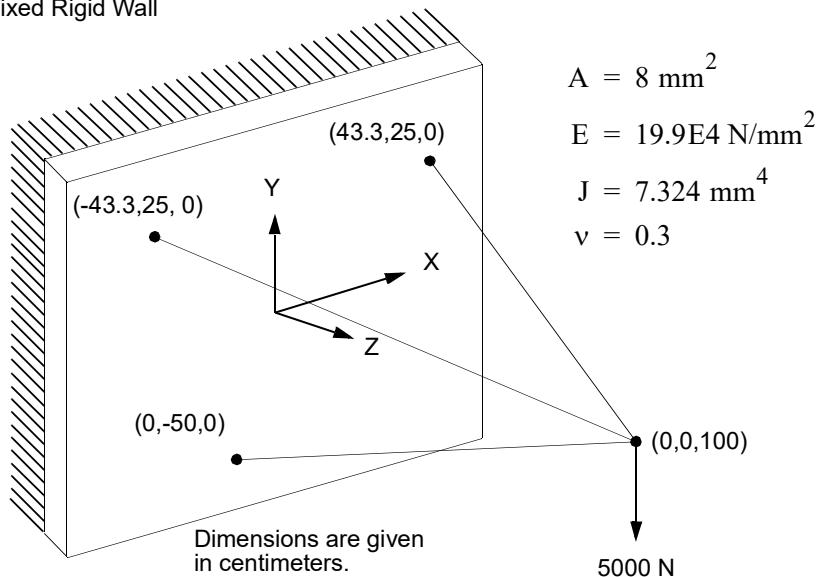
Figure 4-9 Three-Member Structure

Listing 4-3 Three-Member Truss Example

```
$  
$ FILENAME - ROD2.DAT  
$  
ID LINEAR, ROD2
```



Fixed Rigid Wall



```

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.        123456
$
$ 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 4-10](#).

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	-1.170466E+01	-2.575031E-05	0.0	0.0	0.0			
ELEMENT 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	4.658455E+02	0.0			2	4.658455E+02	0.0			
3	-9.316950E+02	0.0								

Figure 4-10 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 4-10](#). 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](#).

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:

- 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 torsional mass moment of inertia is included in the BAR mass matrix by using NASTRAN [BARMASS \(398\)=1](#). Note, by default, this term will not be included. For both values of PARAM COUPMASS, this term is added. If desired, the BARMASS default value can be changed via the NASTRAN rc file.

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 Bulk Data entry [CBAR](#) in the *MSC Nastran Quick Reference Guide*, 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	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Alternate Format

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



Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBAR, PBARL or PBRSECT entry. (Integer > 0 or blank*; Default = EID unless BAROR entry has nonzero entry in field 3.)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA≠GB)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 12. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. The direction of \vec{v} is from GA to G0. \vec{v} is then translated to End A. (Integer > 0; G0≠GA or GB)
OFFT	Offset vector interpretation flag. (character or blank) See Remark 12.
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see the figure). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.) Pin flags are not allowed in MSC Nastran SOL 700.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors \vec{w}_a and \vec{w}_b , respectively (see the figure) in displacement coordinate systems (or in element system depending upon the content of the OFFT field), at points GA and GB, respectively. See Remark 11. and 12. (Real; Default = 0.0) Offsets are not allowed in MSC Nastran SOL 700.

*See the [BAROR](#) entry for default options for field 3 and fields 6 through 9.

11. Offset vectors are treated like rigid elements.

- Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry [MDLPRM,OFFDEF,LROFF](#). For options of offsets, please refer to the Bulk Data entry [MDLPRM,OFFDEF,optn](#).
- The length of offset vectors is not affected by thermal loads. But the thermal load changes due to location changes by offsets are correctly computed if the enhanced method is used.
- BAR elements with offsets will give correct buckling results if the enhanced method is used.
- Masses are correctly transformed for offset effects if the enhanced method is used.
- In nonlinear solution sequences, such as SOL 106 or SOL 400, BAR is treated as linear element. If geometric nonlinear effects are required, please use Bulk Data entry [MDLPRM,BRTOBM,1](#) to convert BAR to BEAM.



12. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \hat{v} and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)

The properties of the CBAR elements are entered on the PBAR entry that is identified by the PID entered in field 3. The format of the Bulk Data entry **PBAR** in the *MSC Nastran Quick Reference Guide* 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.



Field	Contents
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 of 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 4-11](#). 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.

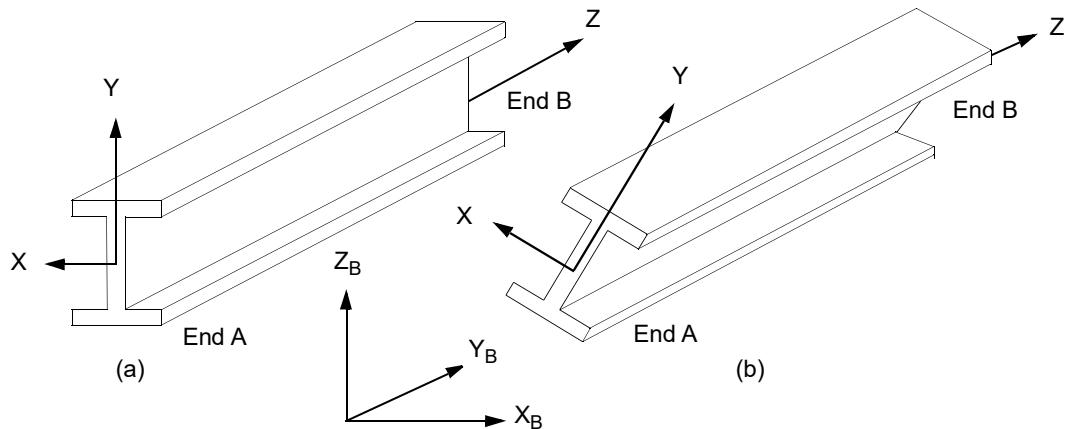
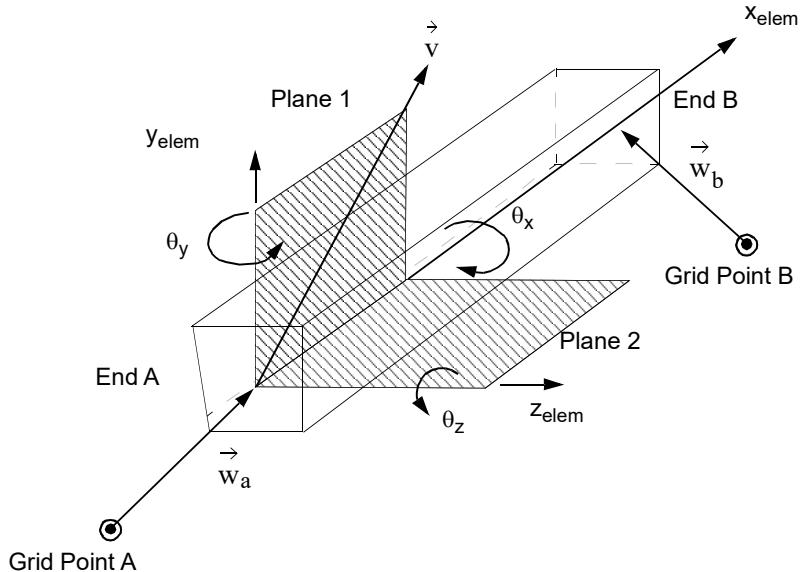


Figure 4-11 Demonstration of Beam Orientation

Defining the orientation of an 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 4-12](#).
 Vector \vec{v} defines plane 1, which contains the elemental x- and y-axes.



[Figure 4-12](#) CBAR Element Coordinate System

Referring to [Figure 4-12](#), 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 4-13](#) 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.



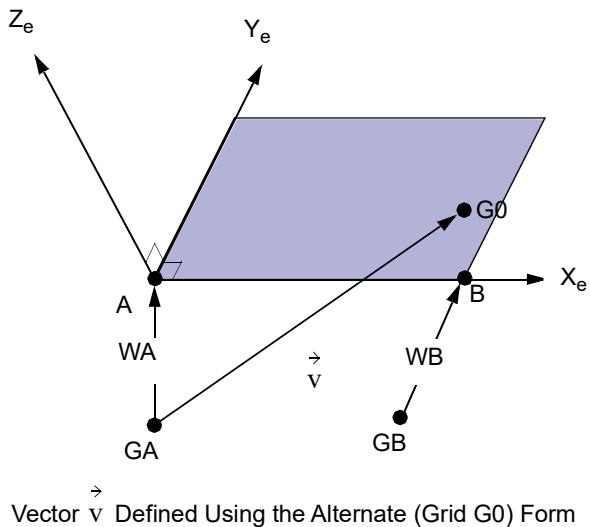
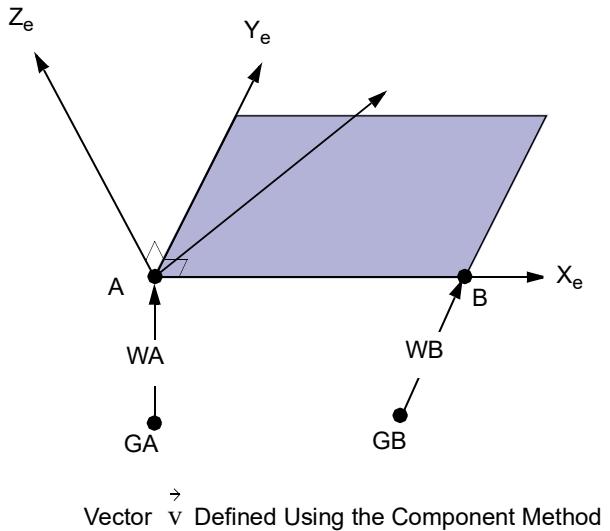


Figure 4-13 CBAR Element Orientation Vector

The offsets 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 CBAR 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 4-14 shows the forces acting on the CBAR 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.



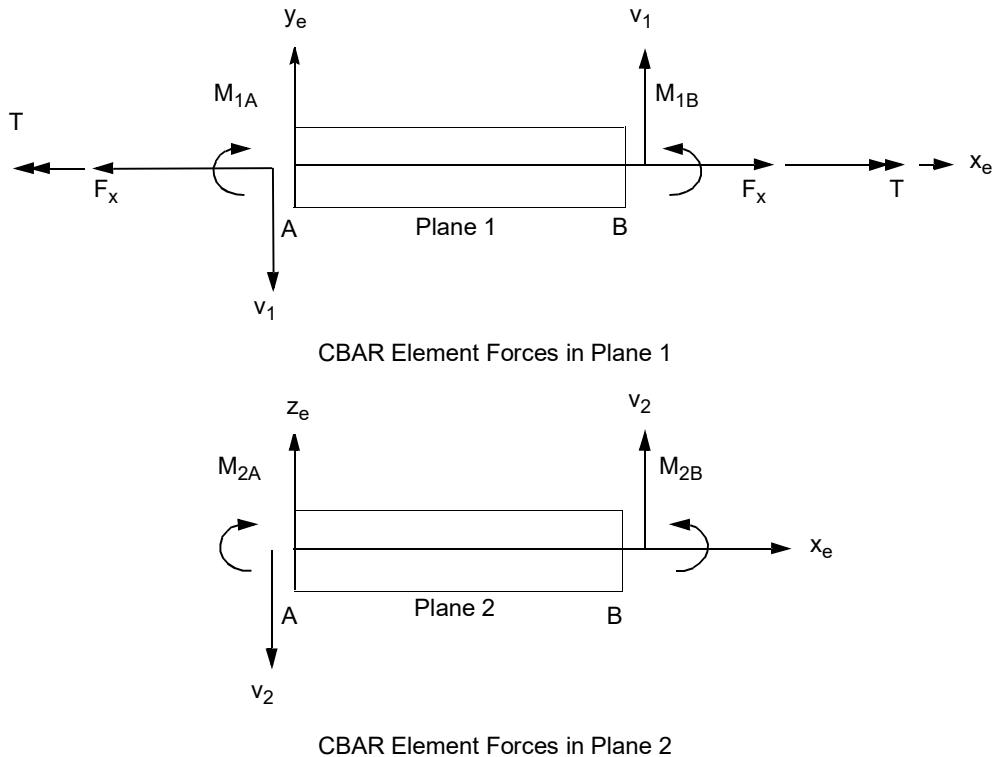


Figure 4-14 CBAR Element Forces

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 4-15](#); 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 4-15](#).



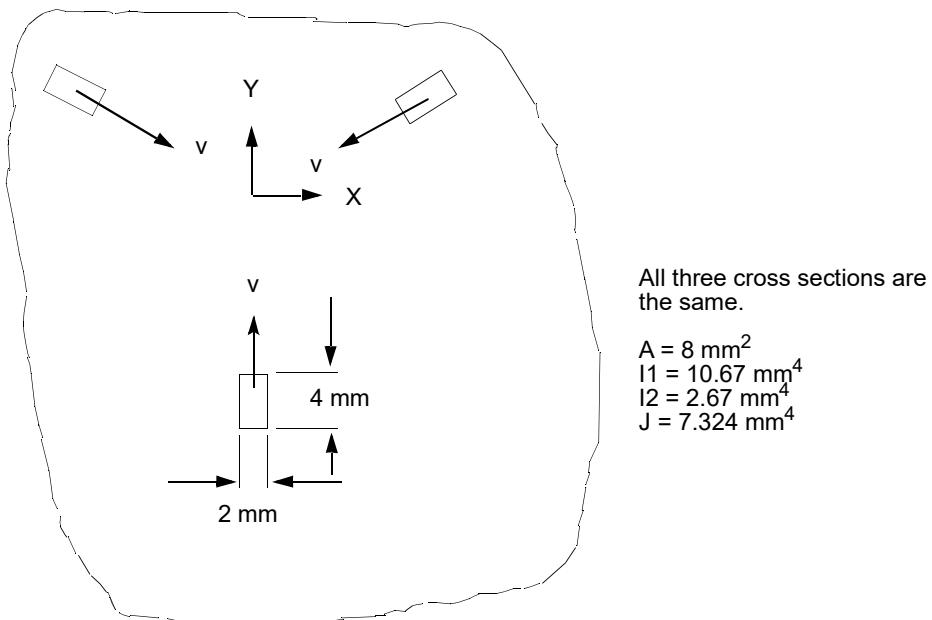


Figure 4-15 CBAR Cross Sections

In this example, the area moments of inertia for each of the members are $I_1 = 10.67$ and $I_2 = 2.67$ for Plane 1 and Plane 2, respectively. The Bulk Data input for this problem is shown in [Listing 4-4](#). (See

\rightarrow
MSC_DOC_DIR/doc/linstat/bar1.dat.) The vector 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

\rightarrow
present) tells MSC Nastran that the component method is being used. Note that the length of vector v is not critical--only the orientation is required to define Plane 1.

Listing 4-4 Vector v \rightarrow 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 4-5](#). (See [MSC_DOC_DIR/doc/linstat/bar1a.dat](#).)

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 4-4](#) and [Listing 4-5](#) produce the same results.

Listing 4-5 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 4-16](#). 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.



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	-1.1704445E+01	-2.574947E-05	1.447189E-02	0.0	0.0	
ELEMENT ID.		S A 1	S A 2	S S E S S I N	B A R E L E M E N T S	(C B A R)		
		S B 1	S B 2	S A 3	S A 4	AXIAL STRESS	S A - MAX	S A - 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	-9.32859E+02

Figure 4-16 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 4-17. 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.

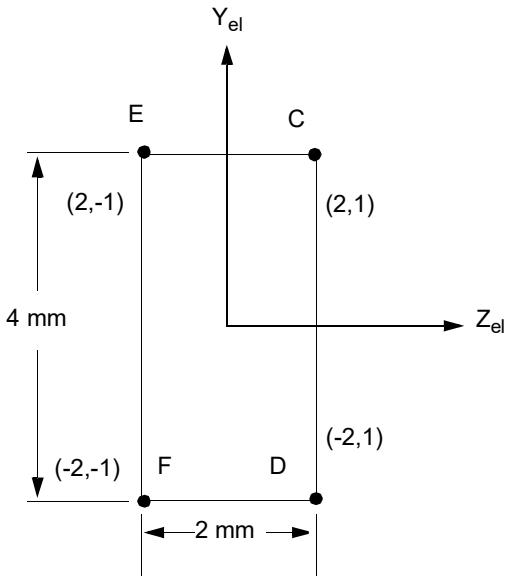


Figure 4-17 Stress Recovery Locations

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 , I_1 , I_2 , J) of each member are input, but the fact that 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.

PBART -- 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.

The beam cross-section library provides an interface to input common 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 Bulk Data entry [PBART](#) in the *MSC Nastran Quick Reference Guide* allows you to input cross section types along with their characteristic dimensions. You can choose from 21 different cross section shapes. These shapes are as follows: ROD, TUBE, TUBE2, I, CHAN (channel), T, BOX, BAR (rectangle), CROSS, H, T1, I1, CHAN1, Z, CHAN2, T2, BOX1, HEXA (hexagon), HAT (hat section), HAT1 and DBOX.

For some of these shapes (I, CHAN, T, and BOX), you can also select different orientations. All the 21 different shapes and orientations are shown in [Figure 4-18](#).

To define section attributes such as height and width on the DIMi fields, use the PBART entry. To define section properties such as area and moment of inertia, use the PBAR Bulk Data entry. The PBART 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 PBART and the PBAR entries is that stress recovery points are not specified by the user to obtain stress output for the PBART entry. The stress recovery points are automatically calculated at the locations shown in [Figure 4-18](#) 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 Remarks 6, 7, and 8 of the [PBART](#) in the *MSC Nastran Quick Reference 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.



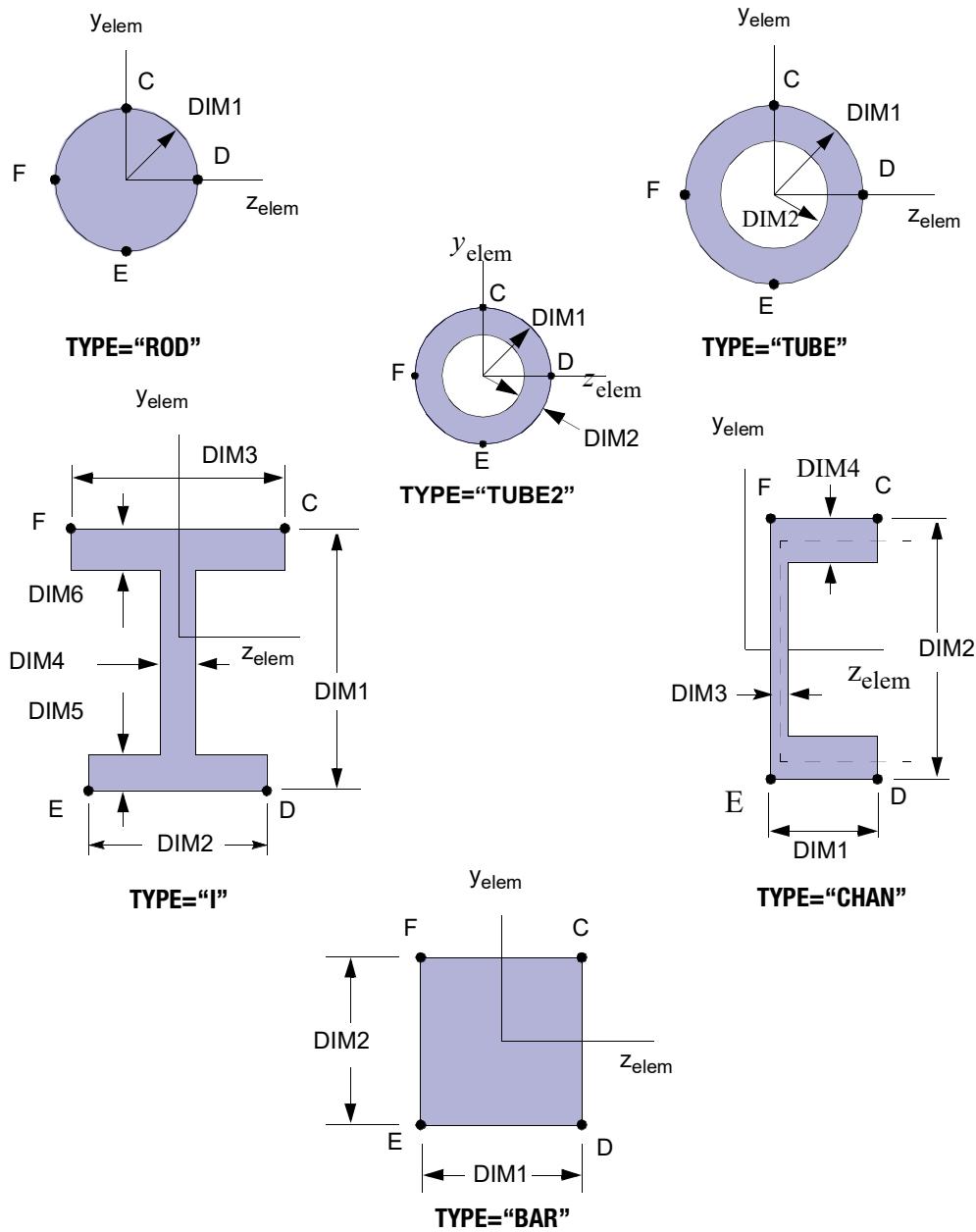


Figure 4-18 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element



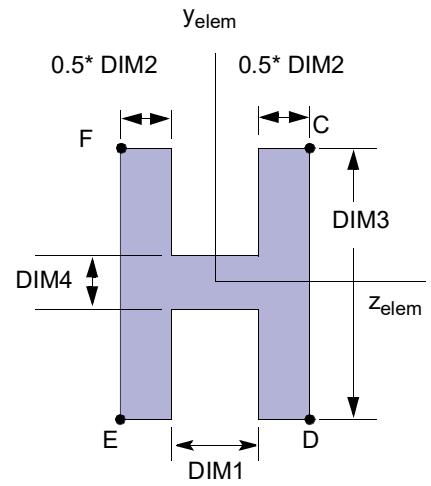
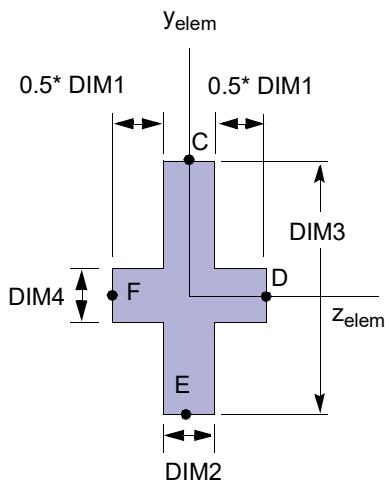
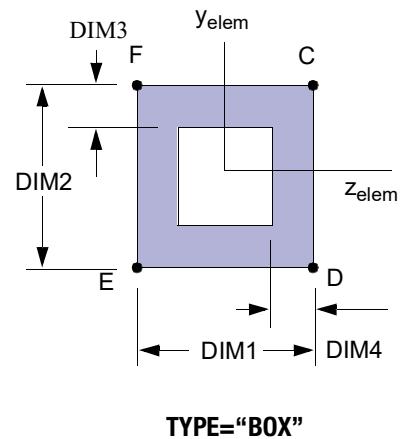
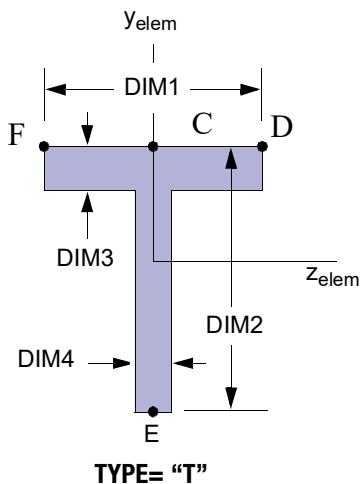


Figure 4-18 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)



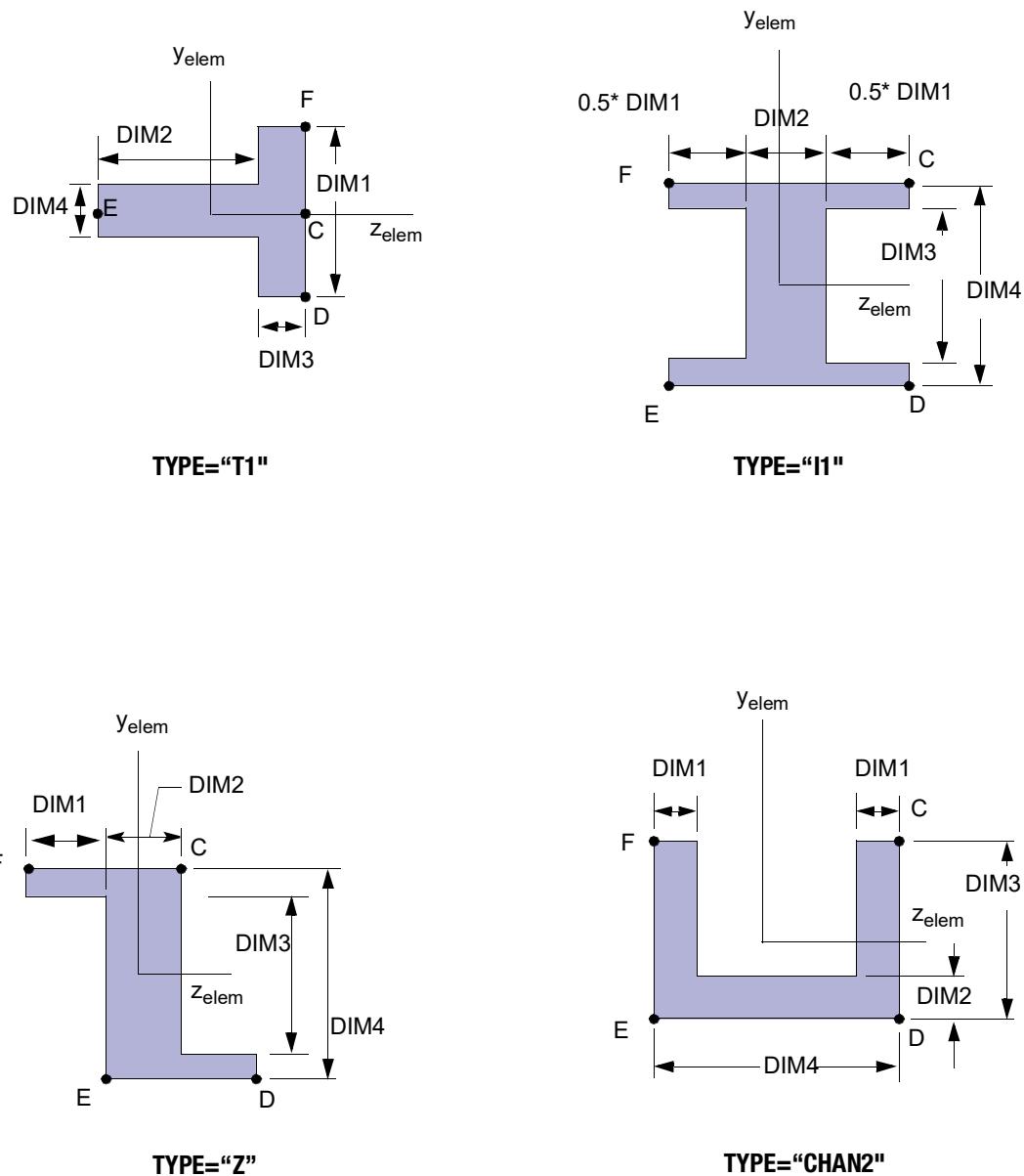


Figure 4-18 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)



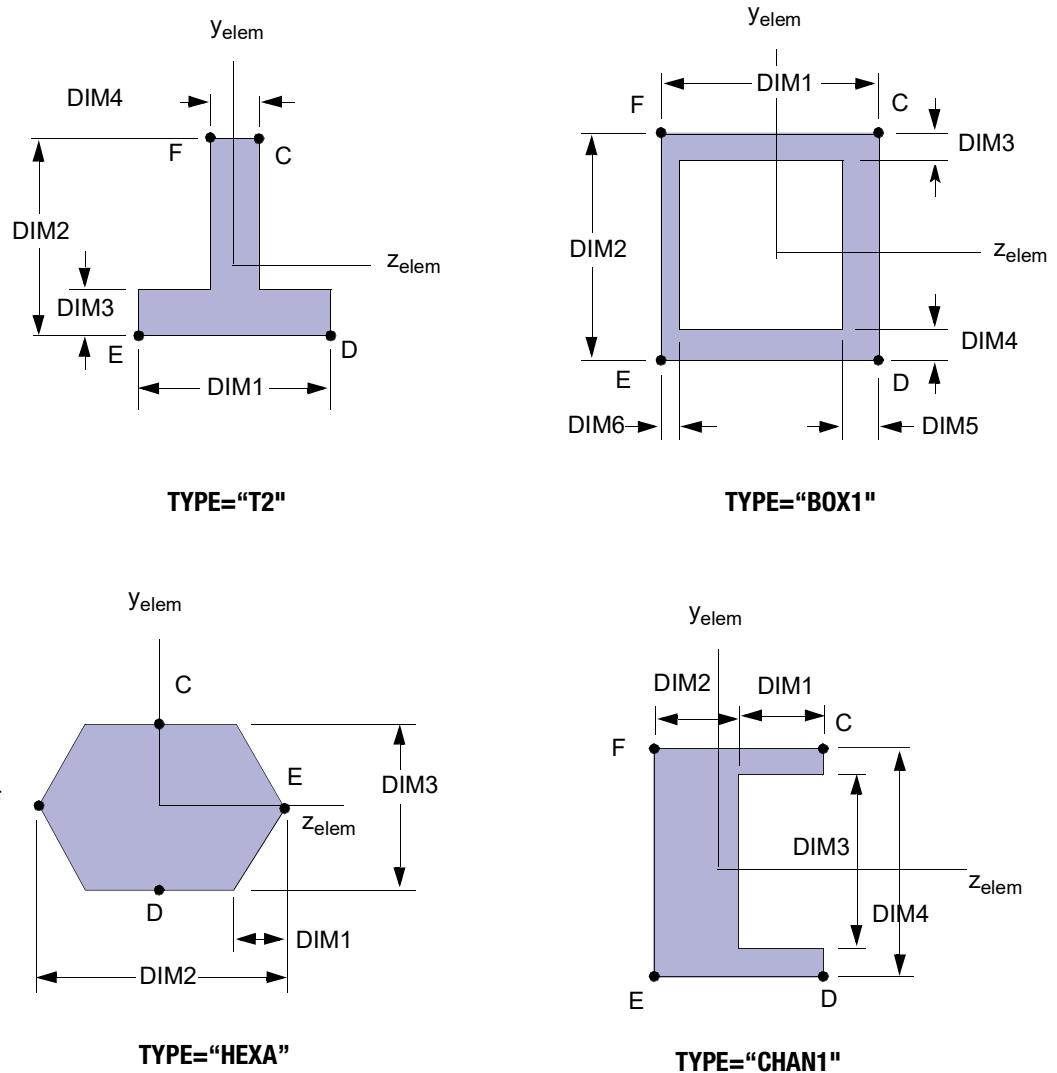


Figure 4-18 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)



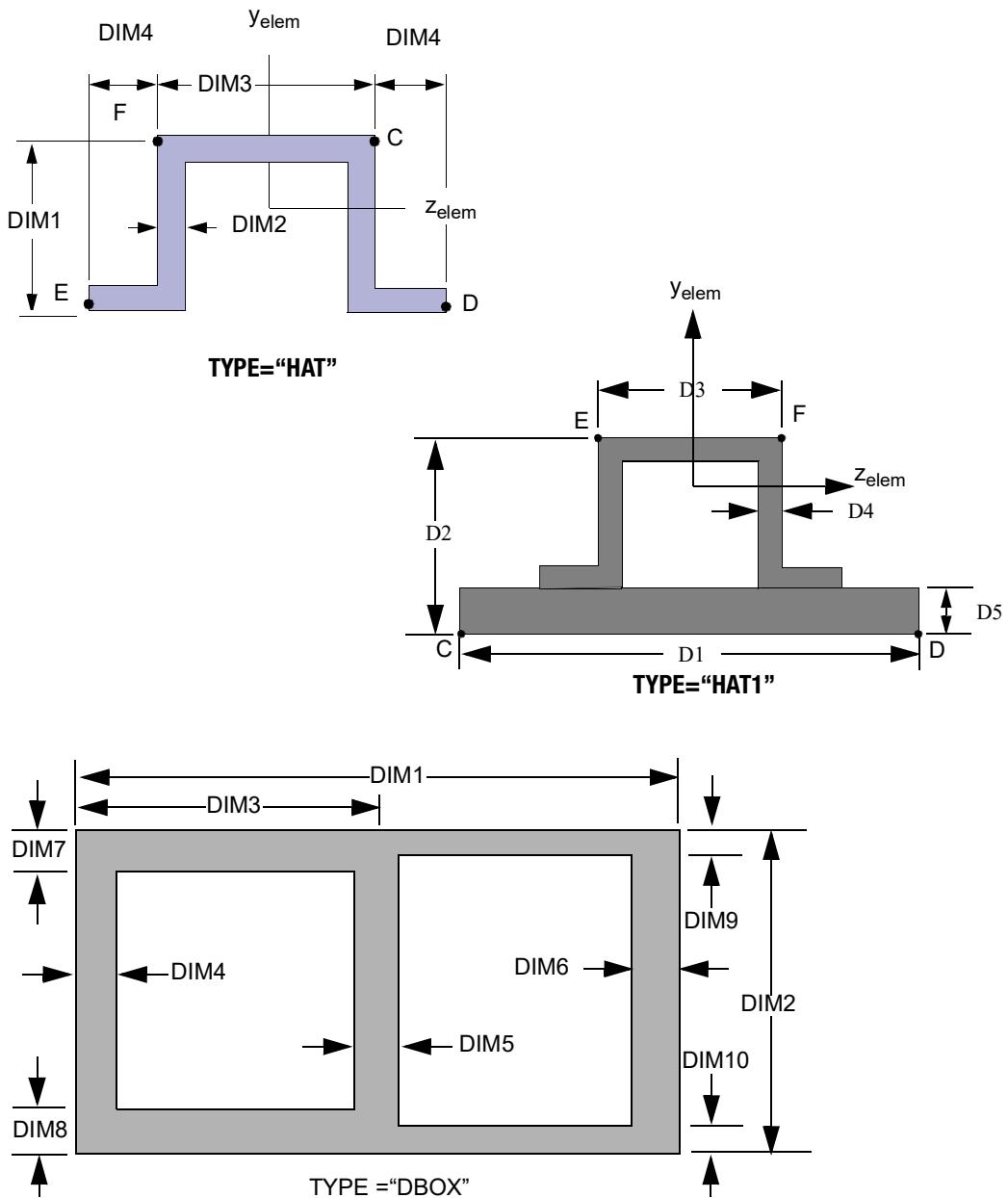


Figure 4-18 Definition of Cross-Section Geometry and Stress Recovery Points for CBAR Element (continued)

The format of the PBARL entry is as follows:

PBARL



1	2	3	4	5	6	7	8	9	10
PBRL	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", "TUBE2", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "DBOX" and HAT1 for GROUP+ "MSCBML0") See Figure 4-18 .
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 PBRL entry. A condensed version of the corresponding output is shown in [Figure 4-19](#). 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 4-6 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.      123456
$ 

```



```

$ 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
$           2.      4.      BAR
$ $ MATERIAL PROPERTIES
$ $ MAT1    1      19.9E4      .3
$ $ POINT LOAD
$ $ FORCE   1      4      5000.      0.      -1.      0.
$ ENDDATA

```

DISPLACEMENT VECTOR											
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	(CBAR)	SA-MAX	SA-MIN	M.S.-T
0 1	G	.0	.0	.0	.0	.0	.0				
0 2	G	.0	.0	.0	.0	.0	.0				
0 3	G	.0	.0	.0	.0	.0	.0				
0 4	G	.0	-1.170445E+01	-2.574947E-05	1.447124E-02	.0	.0				
ELEMENT ID.		S T R E S S E S	I N	B A R E L E M E N T S				(CBAR)	SA-MAX	SA-MIN	M.S.-T
0 1		8.434553E-01	1.054030E+01	-8.434553E-01	-1.054030E+01	4.658372E+02	4.763775E+02	4.552969E+02			
0 2		-2.004751E+00	-1.398322E+00	2.004751E+00	1.398322E+00		4.678419E+02	4.638324E+02			
0 3		-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			
		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 4-19 Displacement and Stress Results for Bar Structure Using PBARL

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 4-20. 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.



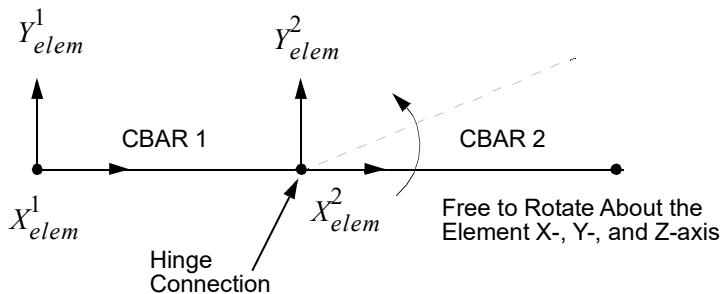


Figure 4-20 A Hinge Connection

The sample bridge model shown in Figure 4-21 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. The deflected shapes for the cases with and without releases are shown in Figure 4-22, and Figure 4-23, 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 4-21.

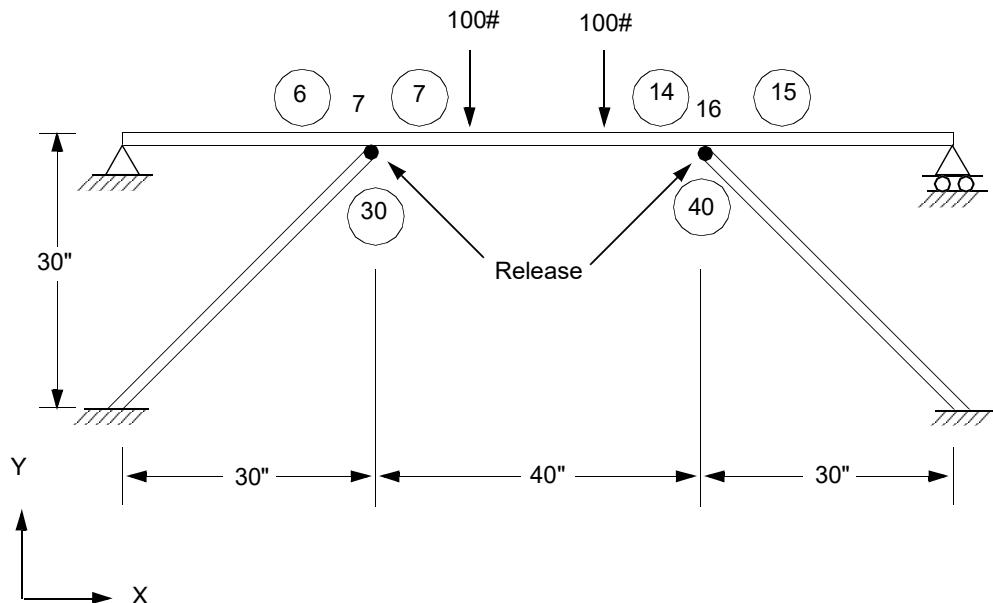


Figure 4-21 Bridge Model Demonstrating the Use of Release



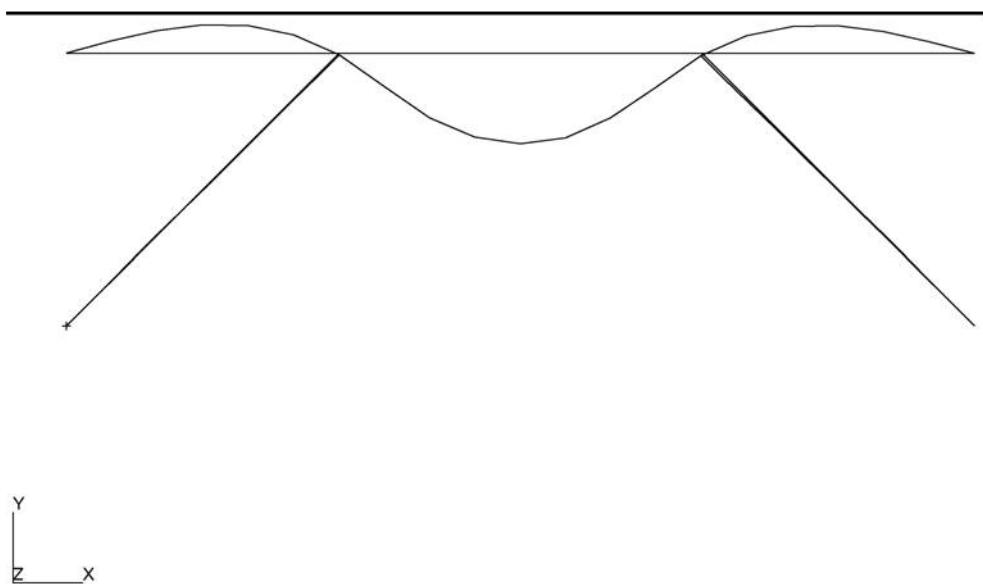


Figure 4-22 Deflected Shape of Bridge with Release at Brace
See sample problem MSC_DOC_DIR/doc/linstat/bridge1.dat.

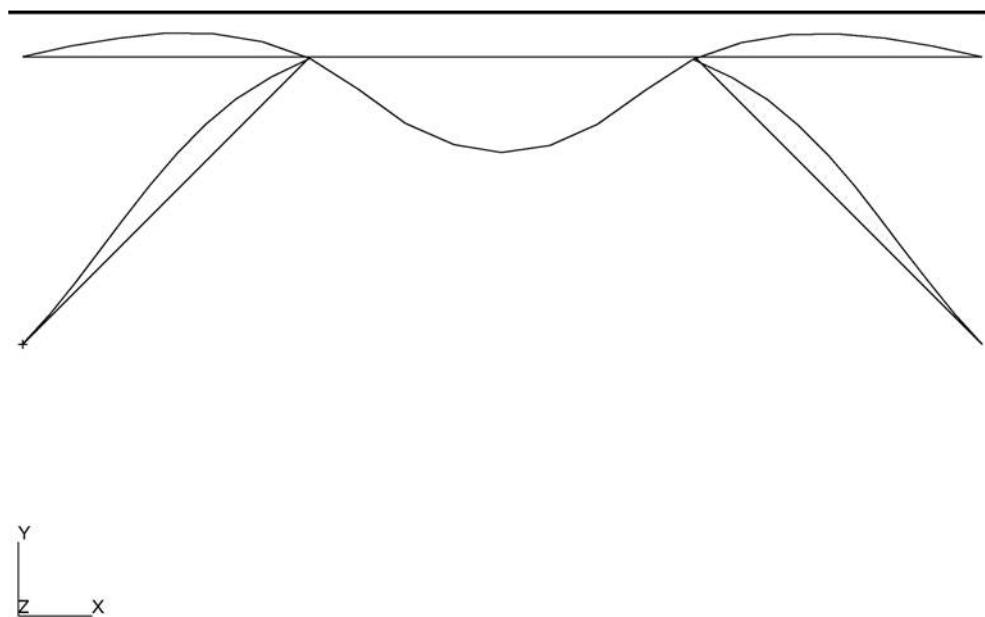


Figure 4-23 Deflected Shape of Bridge without Release at Brace
See sample problem MSC_DOC_DIR/doc/linstat/bridge2.dat.



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	1.298114E+00
0 2	9.251410E+02 1.850282E+03	-9.251410E+02 -1.850282E+03	-9.251410E+02 -1.850282E+03	9.251410E+02 1.850282E+03	1.298114E+00	9.264391E+02 1.851580E+03	-9.238429E+02 -1.848984E+03	-9.238429E+02 -1.848984E+03
0 6	4.625705E+03 5.550846E+03	-4.625705E+03 -5.550846E+03	-4.625705E+03 -5.550846E+03	4.625705E+03 5.550846E+03	1.298114E+00	4.627003E+03 5.552144E+03	-4.624407E+03 -5.549547E+03	-4.624407E+03 -5.549547E+03
0 7	5.550846E+03 -1.805041E+01	-5.550846E+03 1.805041E+01	-5.550846E+03 1.805041E+01	5.550846E+03 -1.805041E+01	-4.153344E+02	5.135511E+03 -3.972393E+02	-5.966180E+03 -4.333848E+02	-5.966180E+03 -4.333848E+02
0 14	-4.863751E+01 5.510063E+03	4.863751E+01 -5.510063E+03	4.863751E+01 -5.510063E+03	-4.863751E+01 5.510063E+03	-4.153344E+02	-3.666968E+02 5.094729E+03	-4.639719E+02 -5.925397E+03	-4.639719E+02 -5.925397E+03
0 15	5.510063E+03 4.591719E+03	-5.510063E+03 -4.591719E+03	-5.510063E+03 -4.591719E+03	5.510063E+03 4.591719E+03	.0	5.510063E+03 4.591719E+03	-5.510063E+03 -4.591719E+03	-5.510063E+03 -4.591719E+03
0 30	2.083161E+00 -7.759377E-08	-2.083161E+00 7.759377E-08	-2.083161E+00 7.759377E-08	2.083161E+00 -7.759377E-08	-5.893649E+02	-5.872817E+02 -5.893649E+02	-5.914481E+02 -5.893649E+02	-5.914481E+02 -5.893649E+02
0 40	4.021125E+00 -3.032399E-08	-4.021125E+00 3.032399E-08	-4.021125E+00 3.032399E-08	4.021125E+00 -3.032399E-08	-5.876757E+02	-5.836545E+02 -5.876757E+02	-5.916968E+02 -5.876757E+02	-5.916968E+02 -5.876757E+02

Figure 4-24 Abridged Stress Output of Bridge with Release at Brace

ELEMENT ID.	S SA1 SB1	T SA2 SB2	R SA3 SB3	E I N SA4 SB4	L E L E M E N T S AXIAL STRESS	A (C B A R) SA-MAX SB-MAX	B S A-MIN S B-MIN	M.S.-T M.S.-C
0 1	1.265061E-12 5.686420E+02	-1.265061E-12 -5.686420E+02	-1.265061E-12 -5.686420E+02	1.265061E-12 5.686420E+02	2.220387E+00	2.220387E+00 5.708624E+02	2.220387E+00 5.708624E+02	2.220387E+00 -5.664216E+02
0 2	5.686420E+02 1.137284E+03	-5.686420E+02 -1.137284E+03	-5.686420E+02 -1.137284E+03	5.686420E+02 1.137284E+03	2.220387E+00	5.708624E+02 1.139504E+03	5.708624E+02 -1.135064E+03	5.708624E+02 -1.135064E+03
0 6	2.843210E+03 3.411852E+03	-2.843210E+03 -3.411852E+03	-2.843210E+03 -3.411852E+03	2.843210E+03 3.411852E+03	2.220387E+00	2.845430E+03 3.410473E+03	-2.840990E+03 -3.409632E+03	-2.840990E+03 -3.409632E+03
0 7	6.634688E+03 1.062523E+03	-6.634688E+03 -1.062523E+03	-6.634688E+03 -1.062523E+03	6.634688E+03 1.062523E+03	-4.433274E+02	6.191360E+03 6.191954E+02	-7.078016E+03 -1.505850E+03	-7.078016E+03 -1.505850E+03
0 14	1.012320E+03 6.567751E+03	-1.012320E+03 -6.567751E+03	-1.012320E+03 -6.567751E+03	1.012320E+03 6.567751E+03	-4.433274E+02	5.689928E+02 6.124424E+03	-1.455648E+03 -7.011079E+03	-1.455648E+03 -7.011079E+03
0 15	3.384410E+03 2.820342E+03	-3.384410E+03 -2.820342E+03	-3.384410E+03 -2.820342E+03	3.384410E+03 2.820342E+03	.0	3.384410E+03 2.820342E+03	-3.384410E+03 -2.820342E+03	-3.384410E+03 -2.820342E+03
0 30	2.742704E+03 3.222836E+03	-2.742704E+03 -3.222836E+03	-2.742704E+03 -3.222836E+03	2.742704E+03 3.222836E+03	-5.937782E+02	2.148926E+03 2.629058E+03	-3.336482E+03 -3.816614E+03	-3.336482E+03 -3.816614E+03
0 40	2.711180E+03 3.183342E+03	-2.711180E+03 -3.183342E+03	-2.711180E+03 -3.183342E+03	2.711180E+03 3.183342E+03	-5.912410E+02	2.119939E+03 2.592101E+03	-3.302421E+03 -3.774583E+03	-3.302421E+03 -3.774583E+03

Figure 4-25 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 Bulk Data entry **CBEAM** in the *MSC Nastran Quick Reference Guide*, 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	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Alternative Format

CBEAM	EID	PID	GA	GB	G0			OFFT/BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM, PBCOMP, PBEAML or PBMSECT entry. (Integer > 0; Default = EID)*
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA≠GB)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 12. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. Direction of \vec{v} is from GA to G0. \vec{v} is then transferred to End A. (Integer > 0; G0≠GA or GB)
OFFT	Offset vector interpretation flag. See Remark 9. (Character or blank)
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 Figure 4-17). 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.) Pin flags are not presently allowed in MSC Nastran SOL 700.



Field	Contents
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors from the grid points to the end points of the axis of the shear center. See Remarks 11., 12. and 12. (Real; Default = 0.0)
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 $d\theta/dx$. (Integers > 0 or blank)

*See the [BEAMOR](#) entry for default options for field 3 and fields 6 through 9.

11. Offset vectors are treated like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry [MDLPRM](#),[OFFDEF](#),[LROFF](#). For options of offsets, please refer to the Bulk Data entry [MDLPRM](#),[OFFDEF](#),[option](#).
12. OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \vec{v} and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)



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 Bulk Data entry **PBEAM** in the *MSC Nastran Quick Reference Guide* is as follows:

PBEAM

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.

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.



Field	Contents
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 4-26](#), 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 same manner as it is for the CBAR element.

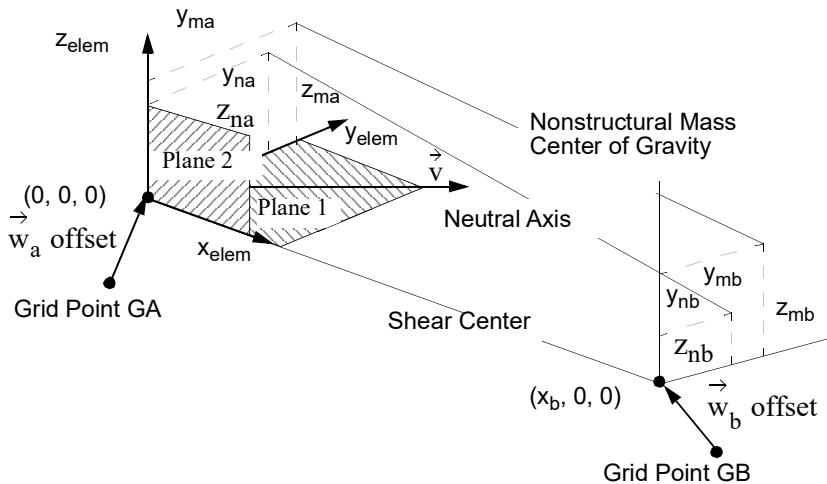


Figure 4-26 CBEAM Element Geometry System



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 4-9. 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. (See MSC_DOC_DIR/doc/linstat/beam1.dat)

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 4-27.

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	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 4-27 CBEAM Stress Output

As shown in Figure 4-27, 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 4-28](#).



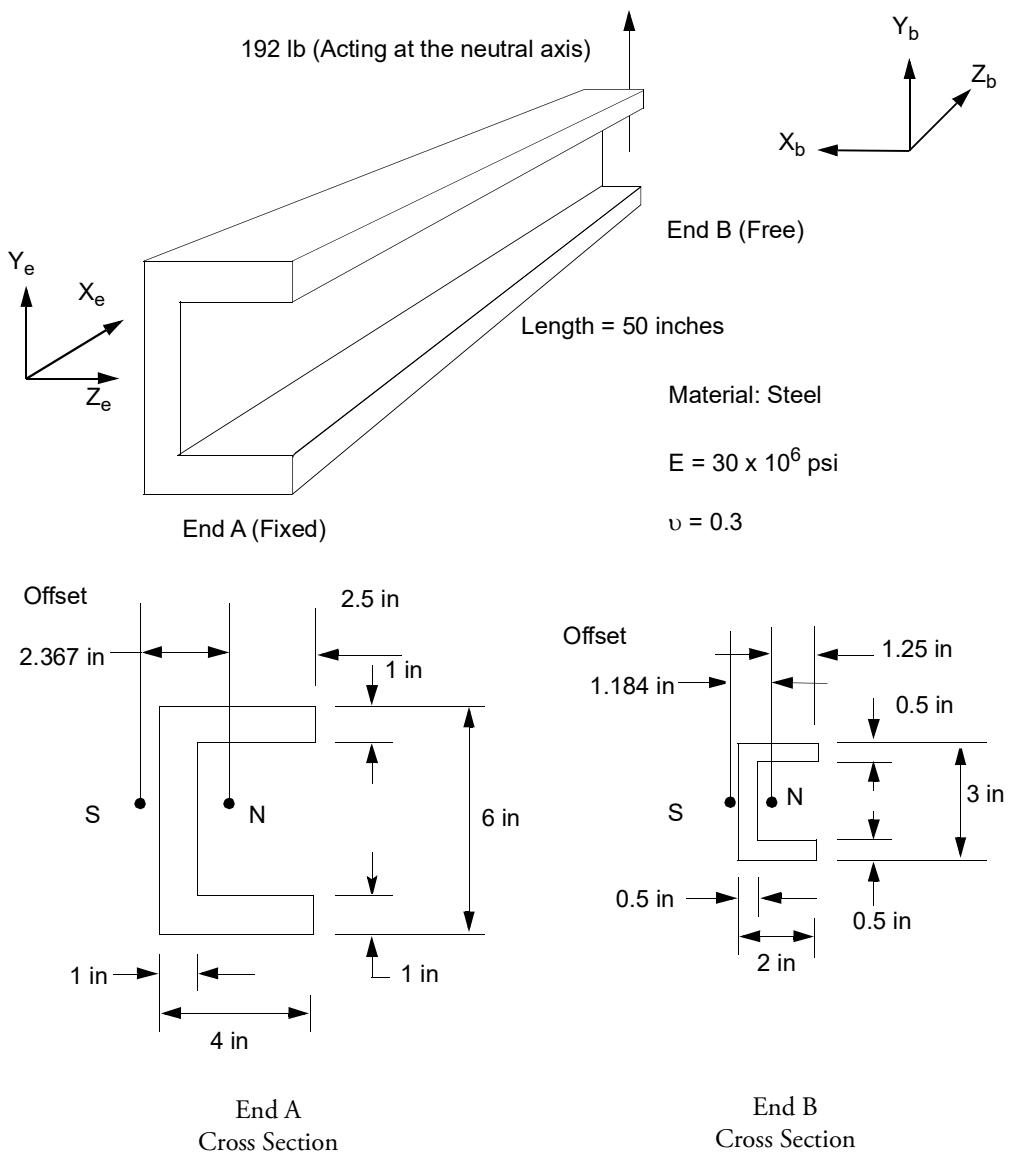


Figure 4-28 Tapered Beam Example

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 4-26](#)) 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 input file is shown in [Listing 4-7](#). (See sample problem MSC_DOC_DIR/doc/linstat/beam2.dat.) 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. 70.43 1.10
                                         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 4-29](#), 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 4-30](#). 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 4-31. The stress output shows only the longitudinal stress; hence, any stress due to torsional or warping is not included.

DISPLACEMENT VECTOR										
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 4-29 Displacement Output for the Tapered Beam

ELEMENT-ID	GRID	STAT DIST/ LENGTH	BEAM ELEMENTS		(C BEAM)		AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
			PLANE 1	PLANE 2	- BENDING MOMENTS -	- WEB PLANE 1			
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 4-30 Force Output for the Tapered Beam

ELEMENT-ID	GRID	STAT DIST/ LENGTH	BEAM ELEMENTS		(C BEAM)		S-MAX	S-MIN	M.S.-T	M.S.-C
			SXC	SXD	SXE	SXF				
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 4-31 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 4-32](#). The PBEAML entry is used for this purpose and the format for the Bulk Data entry [PBEAML](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PBEAML

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")	
	YES	Stresses recovered at all points on next continuation and shown in Figure 4-32 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:

- For structural problems, PBEAML entries must reference a MAT1 material entry.
- PID must be unique with respect to all other PBEAM and PBEAML property identification numbers.
- 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](#) in the *MSC Nastran Utilities 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 4-32](#). 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.



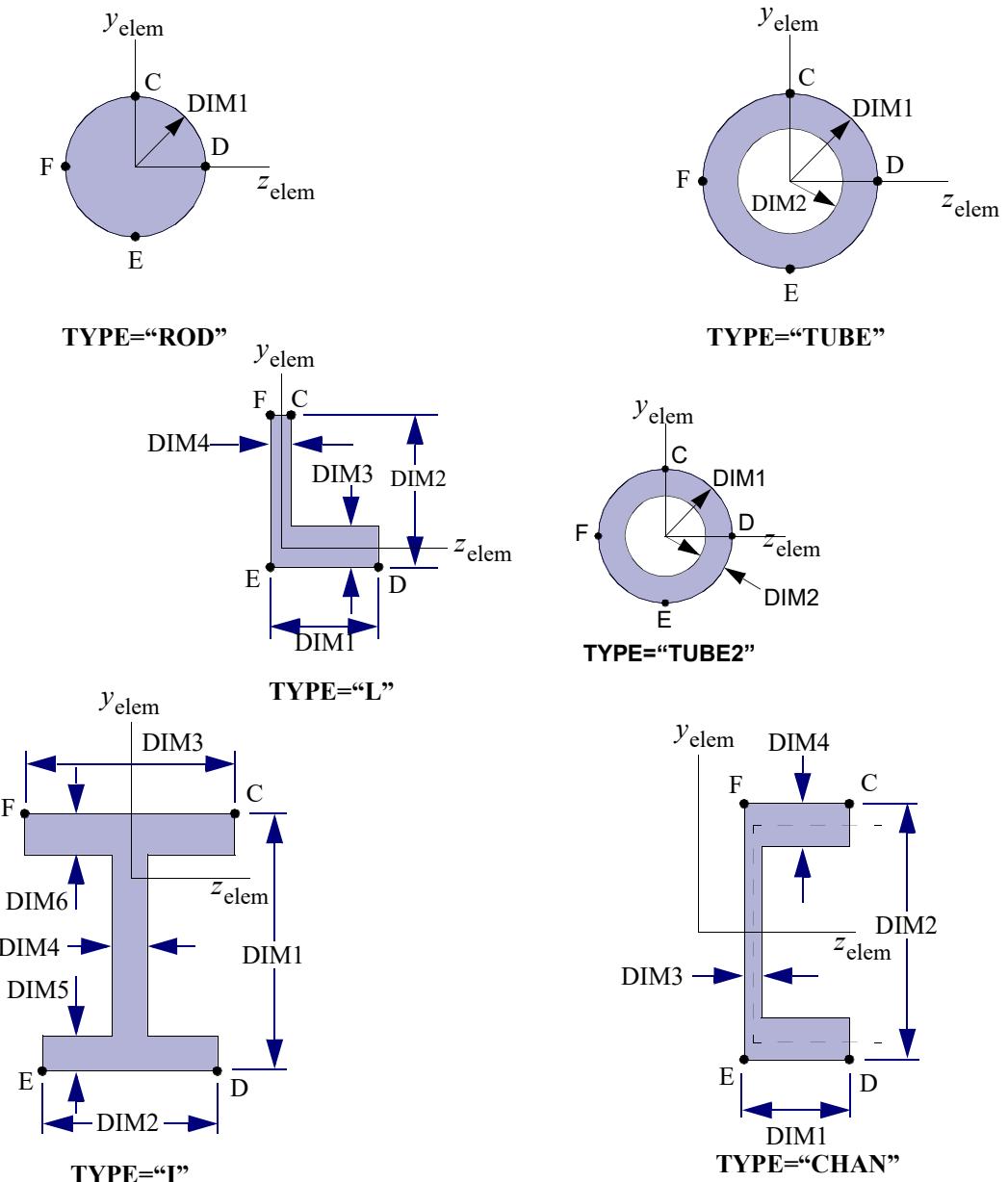


Figure 4-32 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBMLO”



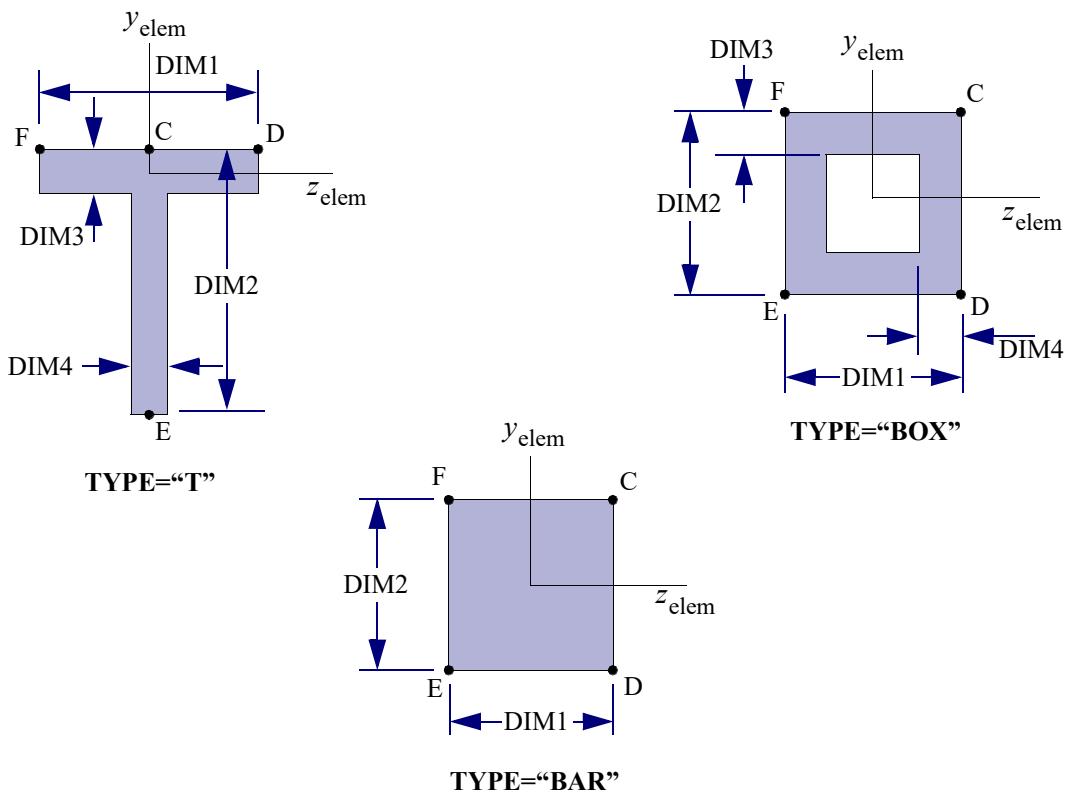


Figure 4-32 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"
(continued)



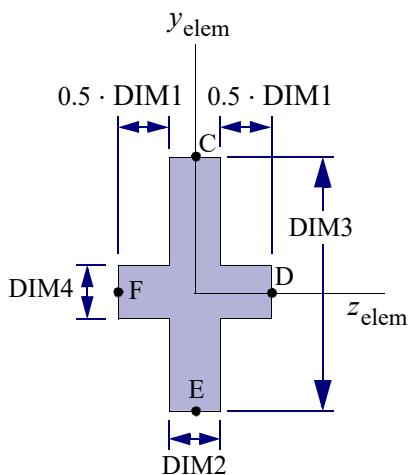
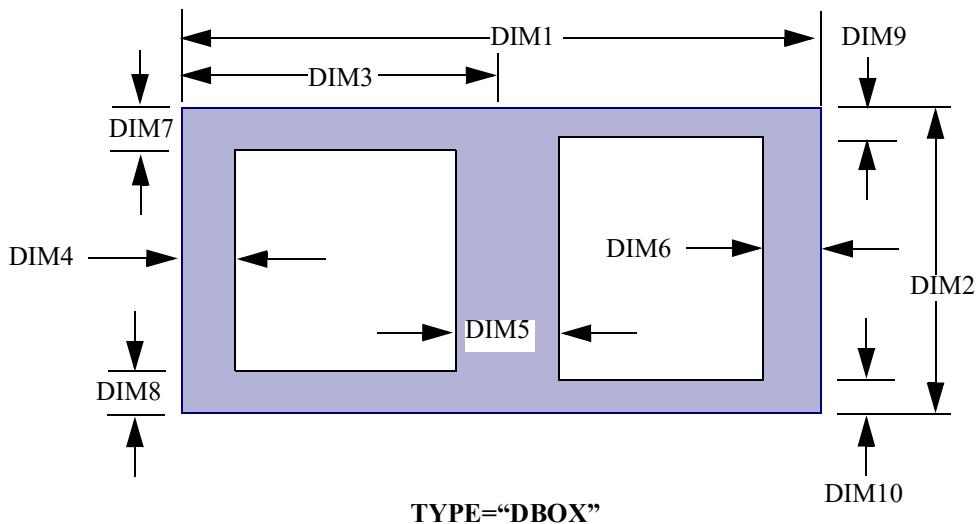
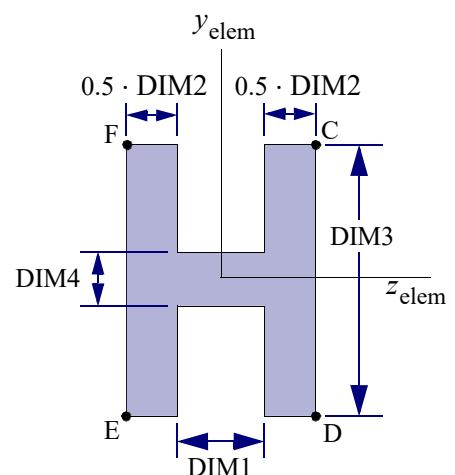
**TYPE=“CROSS”****TYPE=“H”**

Figure 4-32 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0”
(continued)



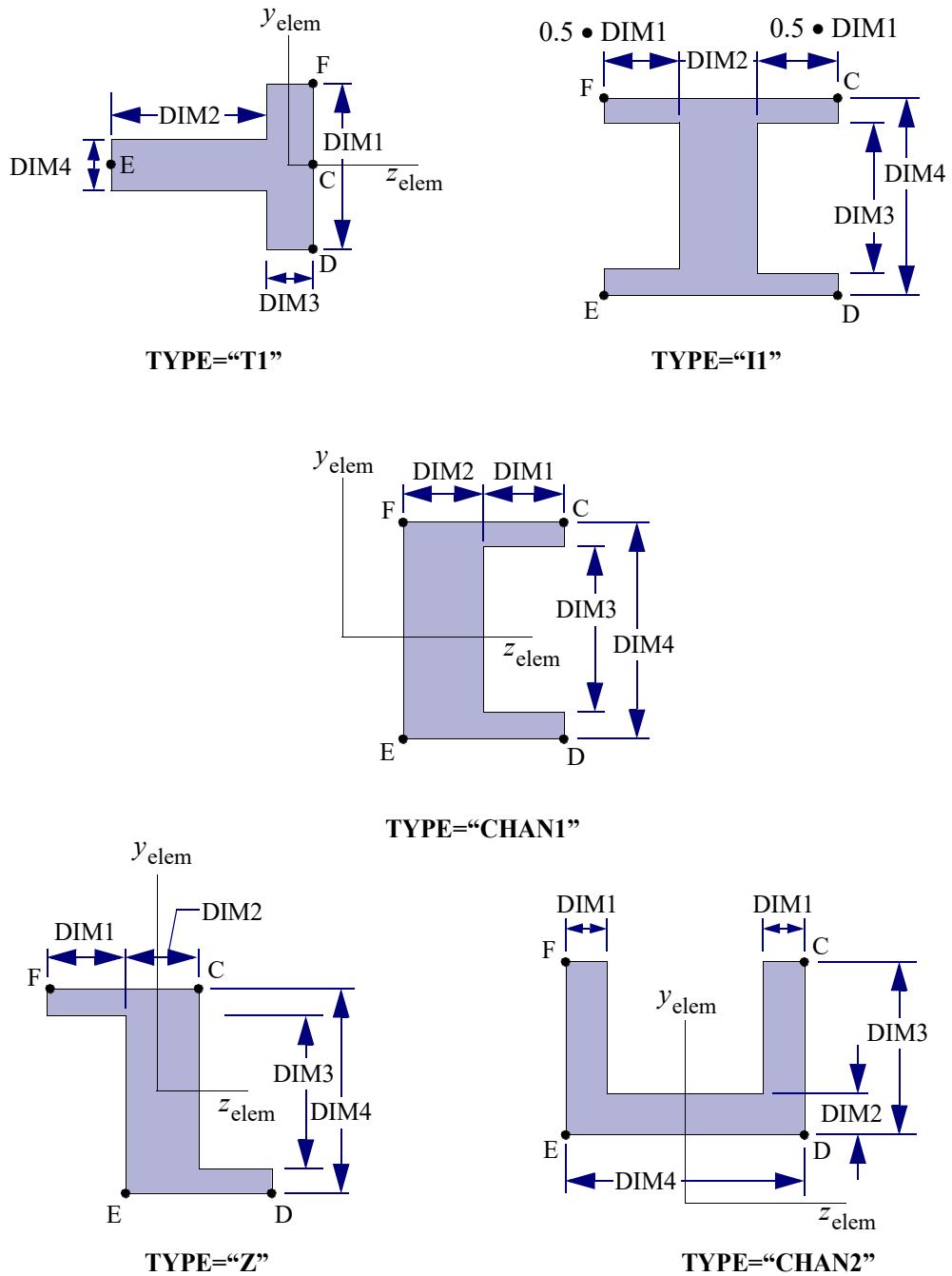


Figure 4-32 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"
(continued)



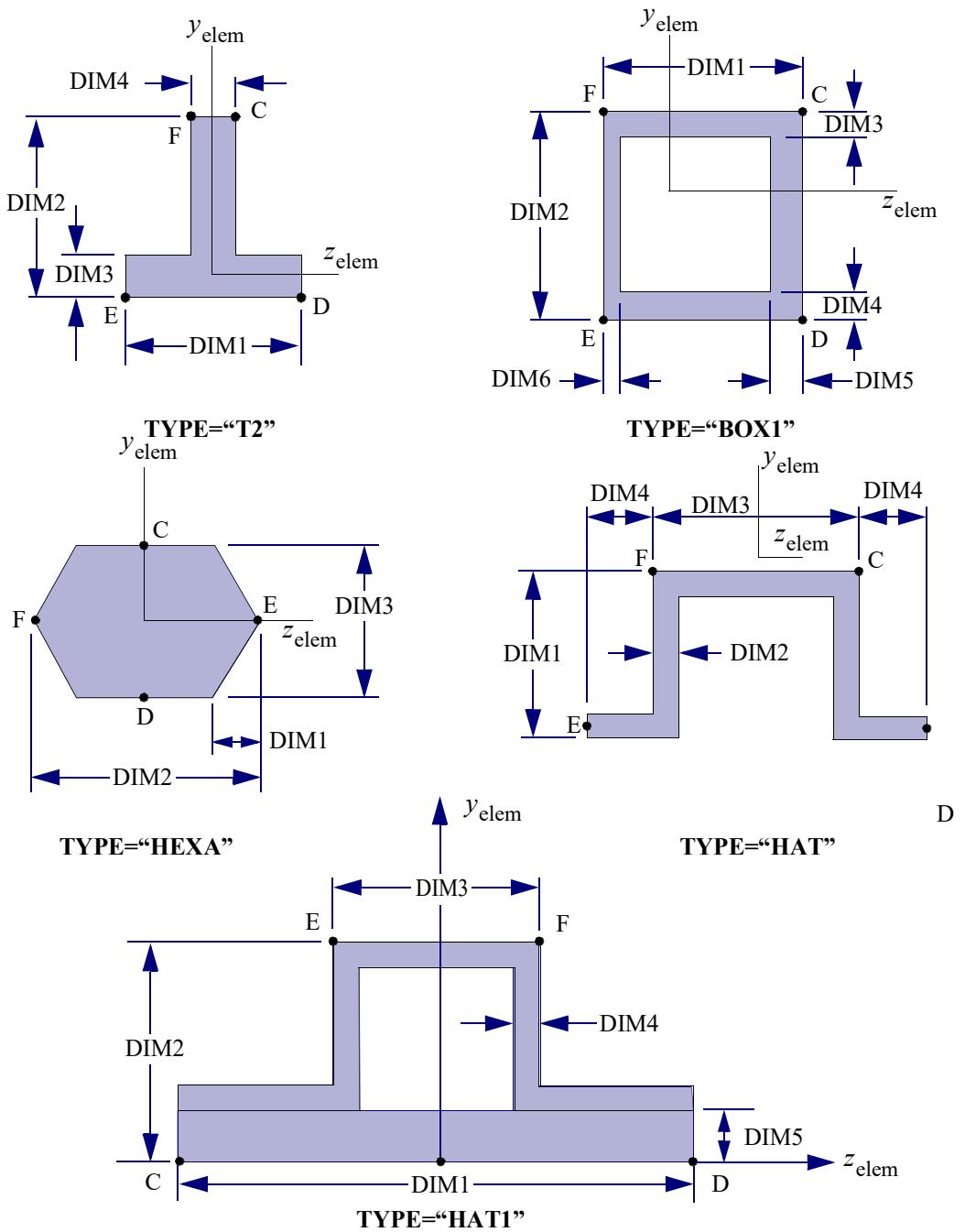


Figure 4-32 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

The problem shown in [Figure 4-28](#) is rerun using the PBEAML entry. (See [MSC_DOC_DIR/doc/linstat/beam2n.dat](#)) 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. Cross sectional properties cannot be output if 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)/XB, DIM1 at X(1)/XB, DIM2 at X(1)/X(B), DIM3 at X(1)/XB, 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.  

                    2.367   0.       0.       1.184   0.       0.  

          101       102  

$  

$23456789  

$  

PBEAML,11,21,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

		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 .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 4-33 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

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

where C_w is the warping coefficient.

The twist of the beam is defined as

$$\phi = \frac{d\theta_x}{dx} \quad (4-3)$$

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

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

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](#) and [Three-Dimensional Elements](#), 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 $\times 10^{-2}$ in	θ_z at the Free End $\times 10^{-3}$	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.

Arbitrary Beam Cross-Section (ABCS)

Beam elements have long been a staple in MSC Nastran. Over the years, the capability of beam element has grown steadily from constant cross section of PBAR to variable cross section of PBEAM. However, users are required to compute the sectional properties in order to utilize BAR and/or BEAM elements in the analysis. To alleviate the amount of effort from engineers, PBARL and PBEAML were added for popular cross sectional profiles. But modeling alternatives for 1-D structural components with arbitrary cross sectional shapes was needed particularly by the automotive industry. The Arbitrary Beam Cross-Section (ABCS) capability uses the PBRSECT and PBMSECT Bulk Data entries along with the POINT, SET1 and SET3 Bulk Data entries.

The shape of the beam cross section is defined using sets of POINTs as defined on the SET1 or SET3 Bulk Data entry. These sets are referenced by Bulk Data entries - [PBRSECT](#) for the BAR, and [PBMSECT](#) in the *MSC Nastran Quick Reference Guide* for the BEAM; which also defines the cross section form parameters, and reference material properties. The types of section that can be defined include a General Section, Open Profile, and Closed Profile, with various parameters required on the PBRSECT or PBMSECT entries to define outer perimeter, inner perimeter, and branch segments where applicable.

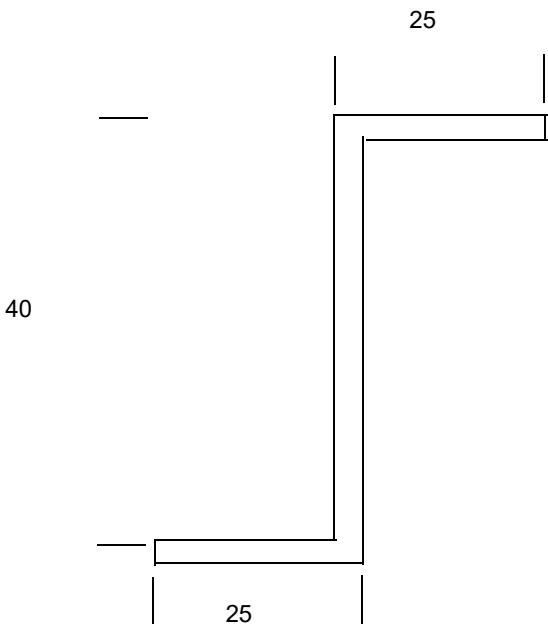
1. BRP for CP and OP must start or end branching from OUTP. BRP must not start or end from another BRP.
2. BRP must not branch out from the end of OUTP. This rule covers both CP and OP.
3. For CP and OP, a $T = rs$, where rs denotes a positive real single precision number, must be present even if the thickness for every segment is separately defined. This thickness will be used for all segments which do not have specific thickness defined for them.



4. When PT=(id1,id2) is utilized to define the thickness of a segment, the id1 and id2 must be next to each other on the SET1 or SET3. A warning message will be issued if this guideline is not observed.

Equivalent BAR and BEAM elements are created from the data supplied by the PBRSECT and PBMSECT entries. These equivalent element definitions are printed on the output file. Currently for the BEAM element, only a constant cross section beam is supported.

Parts of two example problems will be shown. One for the General Section (see [MSC_DOC_DIR/doc/linstat/zbeamgs.dat](#)) and one for the Open Profile (see [MSC_DOC_DIR/doc/linstat/zbeamop.dat](#)). [Figure 4-34](#) shows the Z-Section being defined.



[Figure 4-34](#) Z-Section - Uniform Thickness of 5.

The PBMSECT entry for [Figure 4-34](#) to define the z-section by General Section is:

```
PBMSECT, 100, 200, GS
        OUTP=10
```

The Bulk Data entry [POINT](#) in the *MSC Nastran Quick Reference Guide* defines the z-section in [Figure 4-34](#) in general form are shown below. Also included is the Bulk Data entry [SET3](#) in the *MSC Nastran Quick Reference Guide* to define the set of points to use for the section. SET3's could also select grids or elements to use.

```
$POINTS and SET3 for General Section
point, 1001,,    0.,    0.
point, 1002,,   25.,    0.
point, 1003,,   25.,   35.
point, 1004,,   45.,   35.
point, 1005,,   45.,   40.
point, 1006,,   20.,   40.
```



```
point, 1007,, 20., 5.
point, 1008,, 0., 5.
set3, 10, point, 1001, thru, 1008
```

The PBMSECT entry for [Figure 4-34](#) to define the z-section by Open Profile is below. Note the segment thickness of 5.

```
PBMSECT, 100, 200, OP
        OUTP=10, T=5.
```

The POINT Bulk Data entries to define the z-section in [Figure 4-34](#) in Open Profile are shown. Again the SET3 Bulk Data entry to define the set of points to use for the section is shown.

```
$POINTS and SET3 For Open Profile
point, 1001,, 0.0, 2.5
point, 1002,, 22.5, 2.5
point, 1003,, 22.5, 37.5
point, 1004,, 45. 37.5
set3, 10, point, 1001, thru, 1004
```

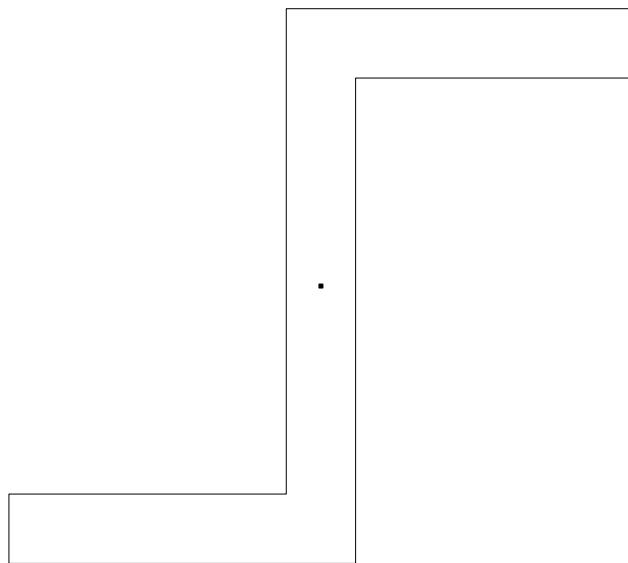
For output, ABCS provides complete stress recovery for the whole cross-section with torsion and shear effect included. The stresses recovered can be post-processed using GPSTRESS via SURFACE for each distinct cross-section. The ABCS algorithm is the default formulation for pre-defined shapes of PBARL/PBEAML. The ABCS algorithm call finite element formulation (FEF) is supported only on short-word machines. Previously, the equations used for PBARL/PBEAML were constructed under the assumption that the cross-sections are thin-walled. Now the properties, especially J, K1, K2, Cwa and Cwb, are computed correctly regardless if the section is thin-walled or not. To return to the previous formulation (Beam Library) set [MDLPRM](#),TWBRBML,1. Selection is noted in the output file.

ABCS output includes a number of features, such as the Output of a PostScript, ‘.ps’, file for displaying the outline of cross-section. Also included in the ‘.ps’ file is the markers of center of gravity and shear center and sectional properties. The generation of the .ps file is controlled by PARAM,ARBMPYES (default). The ‘ps’ file has the name of ‘zbeamgs.ps’ for an input file of ‘zbeamgs.dat’. The PS file includes following important information.

- Outline of the cross-section
- Marker for the location of center of gravity. For ABCS via PBMSECT, marker for the location of shear center is also shown.
- Properties of the cross-section
- As separate .ps is created for each PBRSECT or PBMSECT

‘PARAM,ARBMPNO’ turns off the generation of PS file. [Figure 4-35](#) shows the PBMSECT 100 from zbeamgs.dat.





```
PBMSECT      100
End A
      A   =  4.0000E+02          Cw  =  6.7511E+06
      I1  =  8.8333E-04          N1A= -6.3820E-05
      I2  =  3.8333E-04          N2A=  7.4756E-05
      I12=  4.3750E+04
      J   =  3.3048E+03
      K1  =  4.3692E-01
      K2  =  5.2662E-01
```

Figure 4-35 Outline and Properties for PBMSECT,100

Another output is the generation of the Finite Element Model (FEM), for each cross-section. This is controlled by PARAM,ARBMFEM,YES (default). PARAM,ARBMFEM,NO switches off the generation of '.bdf' file for FEM of cross-section.

The files generated have the following naming convention

AAA_xxxyy_zz.bdf

where AAA – By default, it assume the input file name. To alter AAA to a name other than input file name, use

ASSIGN opcase='any character string' \$

xx	character string of 'BR' for PBRSECT and 'BM' for PBMSECT
yyy	ID of PBRSECT or PBMSECT
zz	station ID. '01' for end A of PBMSECT. No zz section for PBRSECT.



There should be as many files holding FEM of cross-sections as PBRSECT/PBMSECT Bulk Data entries. Note that PBMSECT supports constant section beam only. [Figure 4-36](#) shows part of a sample .bdf file. The GRID and CTRIA6 numbering are based on the processing sequence of the arbitrary beam cross-section. The first GRID (or CTRIA6) of the first cross-section processed will have ID of 10001. The first GRID of 2nd cross-section processed has ID of 20001.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$ GRID and CTRIA6 entries for $ 
$ PBMSECT 100 BEGIN $ 
$ NUMBER OF GRID = 323 $ 
$ NUMBER OF CTRIA6= 130 $ 
$ 
CTRIA6 10001 1 10016 10009 10013 10099 10121 10110
CTRIA6 10002 1 10066 10003 10009 10247 10098 10100
CTRIA6 10003 1 10015 10010 10014 10102 10118 10113
CTRIA6 10004 1 10092 10007 10010 10299 10101 10103
CTRIA6 10005 1 10018 10011 10023 10104 10126 10224
$ 
.....
GRID* 10001 0.0000000D+00
* 0.0000000D+00
GRID* 10002 0.0000000D+00
* 2.5000000D+01
GRID* 10003 3.5000000D+01
* 2.5000000D+01
GRID* 10004 3.5000000D+01
* 4.5000000D+01
GRID* 10005 4.0000000D+01
GRID* 10006 4.0000000D+01
* 2.0000000D+01
GRID* 10007 5.0000000D+00
* 2.0000000D+01
GRID* 10008 5.0000000D+00
* 0.0000000D+00
$ 
.....
GRID and CTRIA6 entries for $ 
PBMSECT 100 END $ 
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

[Figure 4-36](#) FEM .bdf file for PBMSECT 100 (part)

Output stress data recovery for whole cross-section with included contribution from torsion and shear is available. In addition, ‘screened’ version of full stress recovery, which has the maximum/minimum direct stresses and max von Mises stress is also available.

Force (or ELFORCE) Case Control output request is necessary to perform stress recovery for whole cross-section. In addition, PARAM,ARBMS,YES (Default=NO) is needed.

As a data reduction process, max/min of σ , τ_{xy} , τ_{zx} and max von Mises stress of each station of every selected element are collected from stress recovery for the whole cross-section and printed as ‘screened’ stresses. These can be utilized as design response for SOL 200.

The file containing stresses for whole cross-section is in OP2 format and has same naming convention as the FEM for cross-section, but with the “.op2” extension.

For ‘screened’ stresses, an output example is shown in [Figure 4-37](#).



S T R E S S E S I N B E A M E L E M E N T S (S C R E E N E D)									
ELEMENT-ID	STATION	AXIAL		SHEAR-XY		SHEAR-XZ		VON MISES	
		MAX	MIN	MAX	MIN	MAX	MIN	MAX	MAX
1	0.000	2.264151E-01	-2.264151E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	2.268072E-01	
1	1.000	2.037736E-01	-2.037736E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	2.042092E-01	
2	0.000	2.037736E-01	-2.037736E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	2.042092E-01	
2	1.000	1.811321E-01	-1.811321E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.883966E-01	
3	0.000	1.811321E-01	-1.811321E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.883966E-01	
3	1.000	1.584906E-01	-1.584906E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.765443E-01	
4	0.000	1.584906E-01	-1.584906E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.765443E-01	
4	1.000	1.358491E-01	-1.358491E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.655875E-01	
5	0.000	1.358491E-01	-1.358491E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.655875E-01	
5	1.000	1.132075E-01	-1.132075E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.557153E-01	
6	0.000	1.132075E-01	-1.132075E-01	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.557153E-01	
6	1.000	9.056604E-02	-9.056604E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.471463E-01	
7	0.000	9.056604E-02	-9.056604E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.471463E-01	
7	1.000	6.792453E-02	-6.792453E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.401197E-01	
8	0.000	6.792453E-02	-6.792453E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.401197E-01	
8	1.000	4.528302E-02	-4.528302E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.348767E-01	
9	0.000	4.528302E-02	-4.528302E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.348767E-01	
9	1.000	2.264151E-02	-2.264151E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.316308E-01	
10	0.000	2.264151E-02	-2.264151E-02	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.316308E-01	
10	1.000	2.219271E-15	-2.219271E-15	3.623471E-03	-7.354781E-02	7.764793E-03	-5.492842E-02	1.305308E-01	

Figure 4-37 Screened stresses from the Beam elements

A contour plot with data from von Mises stress are shown in [Figure 4-38](#). In this figure, the number after ‘Grid Point Stresses’ has XXYY format where

XX	EID of a BAR/BEAM element.
YY	station ID. ‘00’ for end A. ‘01’ for end B if no intermediate station.

For this case, _100 means beam 1 end A. Use only matching ‘.bdf’ and ‘.op2’ files from [the same job](#) to visualize the stress pattern.



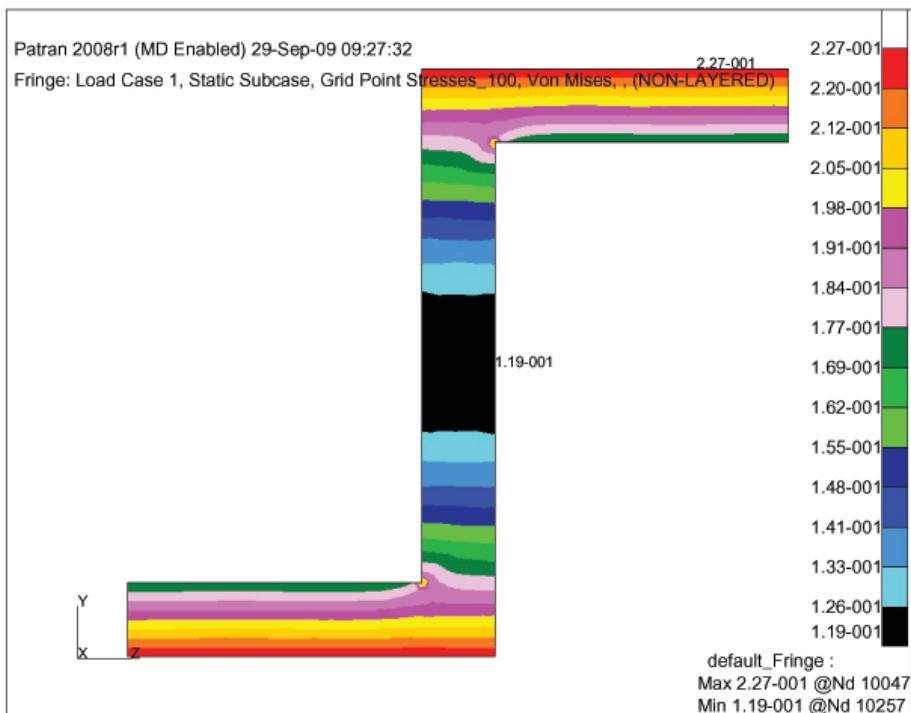


Figure 4-38 von Mises Stress from PBMSECT 100.

Composite Beam - Variational Asymptotic Method (VAM)

The variational asymptotic method (VAM) is used to compute the beam properties of an arbitrary cross section. VAM is available for both isotropic and composite materials and is very useful for composite materials.

Beam-like structures made of composite materials, such as rotor blades, can be modeled with 3-D elements such as CHEXA, CPENTA, and CTETRA, which can account for interaction among plies, but 3-D elements increase the model size. Using VAM and the CBEAM3 three-noded beam element, you can model composite beams. The layup of composite beam plies is described on the PCOMP/PCOMPG Bulk Data entries. In addition, the cross section of the composite beam can be expressed conveniently with the CP/OP options of the PBMSECT Bulk Data entry.

The following keywords are for composite beams on the Bulk Data entry [PBMSECT](#) in the *MSC Nastran Quick Reference Guide*:

`Core (id)=[PCID, PT=(pid1,pid2)]`

Specifies the composite layup for the core part of the composite. PCID is the ID of a PCOMPi/PCOMPG Bulk Data entry. PT=(pid1,pid2) defines the start and end points of line segment(s) which utilize PCID.



Layer (id)=[PCID,SETID]

Specifies the composite layup for additional layer(s) placed on the top or bottom of core. PCID is the ID of a PCOMP/PCOMPG Bulk Data entry. SETID selects a SET1/SET3 with POINT IDs.

OUTM=value (Integer > 0)

Points to the ID of **BEGIN BULK (Case)** ARBMODEL. OUTM is designed specifically for ABCS with finite element discretization already available. Note that OUTM must not appear together with other keywords, such as OUTP or INP, on a PBMSECT entry.

It should be noted that:

1. If Core and/or Layer appears in PBMSECT, the PID of PBMSECT can not be referenced on a CBEAM entry. Instead, it should be referenced on a CBEAM3 entry.
2. Keywords T (for thickness of isotropic beam) and Core (and Layer, for composite beam) cannot coexist on a PBMSECT entry.
3. If OUTM=arbid is used on PBMSECT, element connection, grid location, PSHELL, and material bulk data entries must be provided after the entry “BEGIN ARBMODEL=arbid”.
4. The “arbid” that is specified on “BEGIN ARBMODEL” is considered global, and can be referenced by PBMSECT with OUTM=arbid in a different “BEGIN SUPER” Bulk Data Section for part superelements.
5. PBMSECT with Core or Layer must be used along with “PARAM,ARBMSTYP,TIMOSHEN” in the Bulk Data Section.

Selection of VAM for Properties of the Arbitrary Beam

PARAM, ARBMSTYP, TIMOSHEN

The default value for parameter ARBMSTYP is “VKI” which does not support the composite beam.

PARAM, ARBMASP, 2

The default value for parameter ARBMASP is “2” (integer). Since the thickness of a ply is usually the smaller dimension, ARBMASP affects the size of a CQUAD4 lengthwise along a segment. PARAM,ARBMASP is functional only when PARAM,ARBMSTYP=TIMOSHEN.

Similar to ABCS with isotropic material, full stress recovery must be requested. The required input for ABCS logic to perform stress recovery for the entire cross section is the element force. Hence, to activate stress recovery for the entire cross section, the following must be present in the input file:

1. ELFORCE (or simply FORCE) command in the Case Control Section; and
2. PARAM,ARBMS,YES as a Bulk Data entry (Default = NO).

Due to the potential for a large amount of output, these stresses are only available in OUTPUT2 format. In addition, the direct stresses corresponding to the maximum failure index computed during full ABCS stress recovery are collected for each grid point of a CBEAM3 element and printed.

Generation of finite element model for cross sections is similar to ABCS with isotropic material. The composite beam model is requested with the parameter PARAM,ARBMFEM,YES. The file generated has following naming convention:

AAA_xx yyyy_zz.bdf



where AAA, by default, assumes the input file name. To alter AAA to a name other than the input file name, use

ASSIGN opcase='any character string'

xx = Character string of “BR” for PBRSECT and “BM” for PBMSECT.
 yyy = ID of PBRSECT or PBMSECT.
 zz = Station ID. “01” for end A of PBMSECT. No zz section for PBRSECT.

As the naming implies, there should be a cross sectional FEM referenced by a PBRSECT/PBMSECT Bulk Data entry. Note that PBMSECT supports constant section beam only. For a composite beam, a PSHELL entry for each ply of PCOMP/PCOMPG is also part of the FEM. However, the material entries, such as MAT1, MAT2, and/or MAT8, are not available in the FEM Bulk Data Sections.

The model can be modified (with material entries inserted) and used via ARBMODEL=arbid under the OUTM option of PBMSECT. A word of caution on modification: CQUAD4 elements are ordered in such a way that a line from GID1 to GID2 defines the x-axis of the element coordinate system. Current implementation does not support reordering of element connections based on the user-provided coordinate system.

The stresses for the entire cross section are written to a file in .op2 format and has the same naming convention as the FEM for the cross section. For stresses, the file has the extension of .op2.

For “screened” stresses, an output example is shown as follows:

Guidelines

1. CORE (or C) should point to a PCOMP/PCOMPG entry that has the common plies for most, if not all, segments. For example,

```

$.....2.....3.....4.....5.....6.....7.....8.....9.....10.....
PBMSECT 32.....OP
          OUTP=101, BRP(1)=102, CORE=204,
          layer=(210,101), L(2)=(210,103)
+
PCOMP 204.....-0.254
      501     0.127    -15.0      501     0.127    -15.0
      501     0.127    -15.0      501     0.127    -15.0
      210     0.0
      501     0.127    -15.0
point 2           11.724   6.349
point 3           11.724   -6.349
point 4           -11.724  -6.349
point 5           -11.724  6.349
SET1 101       2      thru   5
SET1 102       5
SET1 103      -5      -4      -3      -2
s

```

2. PBMSECT,32 is a box beam made of composite material. All segments have a common CORE=204 with four plies. Segments from POINTs 2 through 5 have one ply on top, $\text{layer} = (210, 101)$, and one ply at the bottom, $\text{I}_z(2) = (210, 103)$.



3. The negative numbers in SET1,103 indicate that the top segment traverses in opposite sequence to the bottom segment.
4. The centerline of a profile defined by OUTP and BRP must fall in between plies.
5. Plies of a PCOMP/PCOMPG are always placed from bottom to top.
6. During transition from OUTP to BRP (or vice versa), all plies above or below the centerline, defined via OUTP/BRP/SET1 entries, must be carried over.
7. Interior ply insertion or deletion between neighboring segments is not supported. A ply or plies added on top of the outermost ply can be made via the Layer keyword of PBMSECT.
8. A PCOMP/PCOMPG entry referenced on a PBMSECT entry via Core/Layer is converted to multiple PSHELL entries. The value in the THETA field on the PCOMP/PCOMPG entry is placed in the T (thickness) field of the PSHELL entry. Due to the nonnegative requirement of the T field of a PSHELL entry, a negative value of THETA is converted to a supplementary positive value. An example of this conversion is shown as follows:

```

pcOMP    101      -0.1
        501      0.05     0.0
        501      0.05    -45.0
        501      0.05     0.0

PSHELL    10101    501   0.000
PSHELL    10102    501   90.000
$PSHELL   10103    501  -45.000
PSHELL    10103    501  135.000
PSHELL    10104    501   45.000
PSHELL    10105    501   0.000

```

9. Use only matching .bdf and .op2 files from the same job to visualize the stress pattern.

Limitations

1. Although CBEAM3 has curved beam capability, CBEAM3 must be straight when used as a composite beam.
2. TEMPRB is not supported with VAM.

Example

Part of a sample file, (see MSC_DOC_DIR/doc/linstat/vabcore1), with two PBMSECT entries is used here to demonstrate the features implemented. A bracket is modeled with PBMSECT,31 with isotropic material, and PBMSECT,32 with composite material. Some key Bulk Data entries are shown as follows:

```

$
$
param,arbmsTyp,timoshenko
param,arbmsS ,yes
.

.

$ Elements and Element Properties for region : prop2
CBEAM3    2         32          302         102        1301       1204
.

.

$<<<rocker at b pillar>>>
$ U profile
point    1                  0.8      14.5

```



```

point   2           0.9    13.5
point   3           1.0    13.0
point   4           1.1    12.5
point   5           1.5    12.0
point   6           1.75   11.5
point   7           2.6    10.9
point   8           2.7    10.0
point   9           2.6    9.0
point   10          2.4    8.0
point   11          1.9    6.0
point   12          1.1    4.0
point   13          0.1    1.4
point   14          3.0    1.8
point   15          4.5    2.0
point   16          6.1    2.2
point   17          6.7    5.6
point   18          7.4    9.56
point   19          7.35   10.4
point   20          7.7    13.0
point   21          7.6    15.4
SET1    101         1      thru   21
$
$ center branch
$15 point   31           4.5    2.0
point   32           5.5    2.8
point   33           6.8    9.6
point   34           6.6    15.1
SET1    103         15      32      33      34
$
$ Isotropic case
PBMSECT 31         1      OP     0.015
          OUTP=101,T=0.1,brp=103
$
$ Composite case
PBMSECT 32         1      OP     0.015
          OUTP=101,C=101,brp=103,c(1)=[201,pt=(15,34)]
pcomp   101        -0.1
          501        0.05   0.0      501        0.05   90.0
          501        0.05   -45.0    501        0.05   45.0
          501        0.05   0.0
pcomp   201
          501        0.05   -45.0    501        0.05   45.0
          501        0.05   0.0
MAT1    501        3.+6   .3
.

```

This model uses the default values for PARAM,ARB MPS and PARAM,ARB MFEM. The finite element model for the composite beam of PBMSECT,32 is shown as follows:



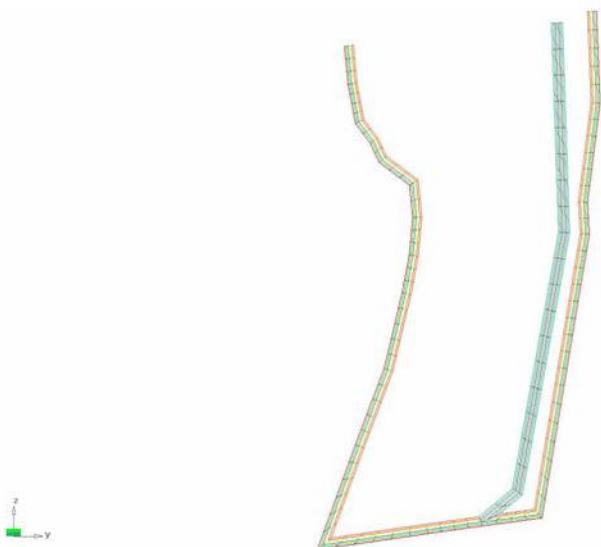


Figure 4-39

The properties of composite PBMSECT,32 computed as a 6x6 D matrix. A PBEAM3 is created internally and printed, if requested. The equivalent PBEAM3 for PBMSECT,32 is shown as follows:

```
*** USER INFORMATION MESSAGE 4379 (IFP9B)
  THE USER SUPPLIED PBMSECT BULK DATA ENTRIES ARE REPLACED BY THE FOLLOWING PBEAM3 ENTRIES.
  CONVERSION METHOD FOR PBAML/PBEAML - .
PBEAM3      32      0  4.7202E+00  8.3059E+01  2.9578E+01 -1.5664E+01  3.2316E+01  0.0000E+00
           1.8014E+01  4.2136E+00  1.7100E+01 -2.7858E+00  3.8881E+00 -3.5404E+00  4.7202E+00  2.6994E+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
           1.2253E+08 -2.1160E+05  8.1193E+04 -2.4761E+06 -3.7193E+06  7.9040E+05 -2.1160E+05  2.1792E+06
          -1.7859E+06  1.9780E+07  5.4643E+05 -3.5845E+05  8.1193E+04 -1.7859E+06  2.7228E+07  1.7190E+07
          2.9835E+04  2.1407E+06 -2.4761E+06  1.9780E+07  1.7190E+07  2.2332E+08  5.8182E+06 -1.2186E+06
         -3.7193E+06  5.4643E+05  2.9835E+04  5.8182E+06  2.1349E+09 -4.0706E+08  7.9040E+06 -3.5845E+05
         2.1407E+06 -1.2186E+06 -4.0706E+08  7.5602E+08
```

The 6x6 D matrix starts in the Wi(j) section of PBEAM3 (the seventh line of the previously described PBEAM3). Note that the previously described PBEAM3 is for information only. It does not have a punched version for later use.

The stress output for the composite CBEAM3, if PARAM,ARBMS,YES is specified, is shown as follows:



SUBCASE 1

S T R E S S E S I N L A Y E R E D C O M P O S I T E E L E M E N T S (B E A M 3)

ELEMENT ID	GRID ID	PLY ID	D I R E C T	S T R E S S E S	FAILURE	MAXIMUM	STRENGTH
2	302	2	NORMAL-1	NORMAL-2	NORMAL-3	SHEAR-12	SHEAR-23
			1.277E+02	1.189E+02	-4.705E+00	1.490E+02	-1.166E+00
	102	2	1.242E+02	1.185E+02	-4.690E+00	1.473E+02	-1.119E+00
	1301	2	1.259E+02	1.187E+02	-4.698E+00	1.481E+02	-1.143E+00
						-7.267E+00	TSAI-WU
						5.511E-03	6.032E+01

SUBCASE 2

S T R E S S E S I N L A Y E R E D C O M P O S I T E E L E M E N T S (B E A M 3)

ELEMENT ID	GRID ID	PLY ID	D I R E C T	S T R E S S E S	FAILURE	MAXIMUM	STRENGTH
2	302	2	NORMAL-1	NORMAL-2	NORMAL-3	SHEAR-12	SHEAR-23
			2.060E+01	1.647E+01	-5.759E-01	2.148E+01	-2.284E-01
	102	2	1.685E+01	1.610E+01	-7.230E-01	1.993E+01	-1.377E-01
	1301	2	1.812E+01	1.626E+01	-7.292E-01	2.048E+01	-1.593E-01
						-5.863E-01	TSAI-WU
						7.323E-04	4.376E+02

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 4-40](#)).
- The geometric center of the element may be offset in two directions (see [Figure 4-40](#)).
- 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 Bulk Data entry [CBEND](#) in the *MSC Nastran Quick Reference Guide* is as follows:

CBEND

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. \rightarrow
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. \rightarrow
GEOM	Flag to select specification of the bend element.



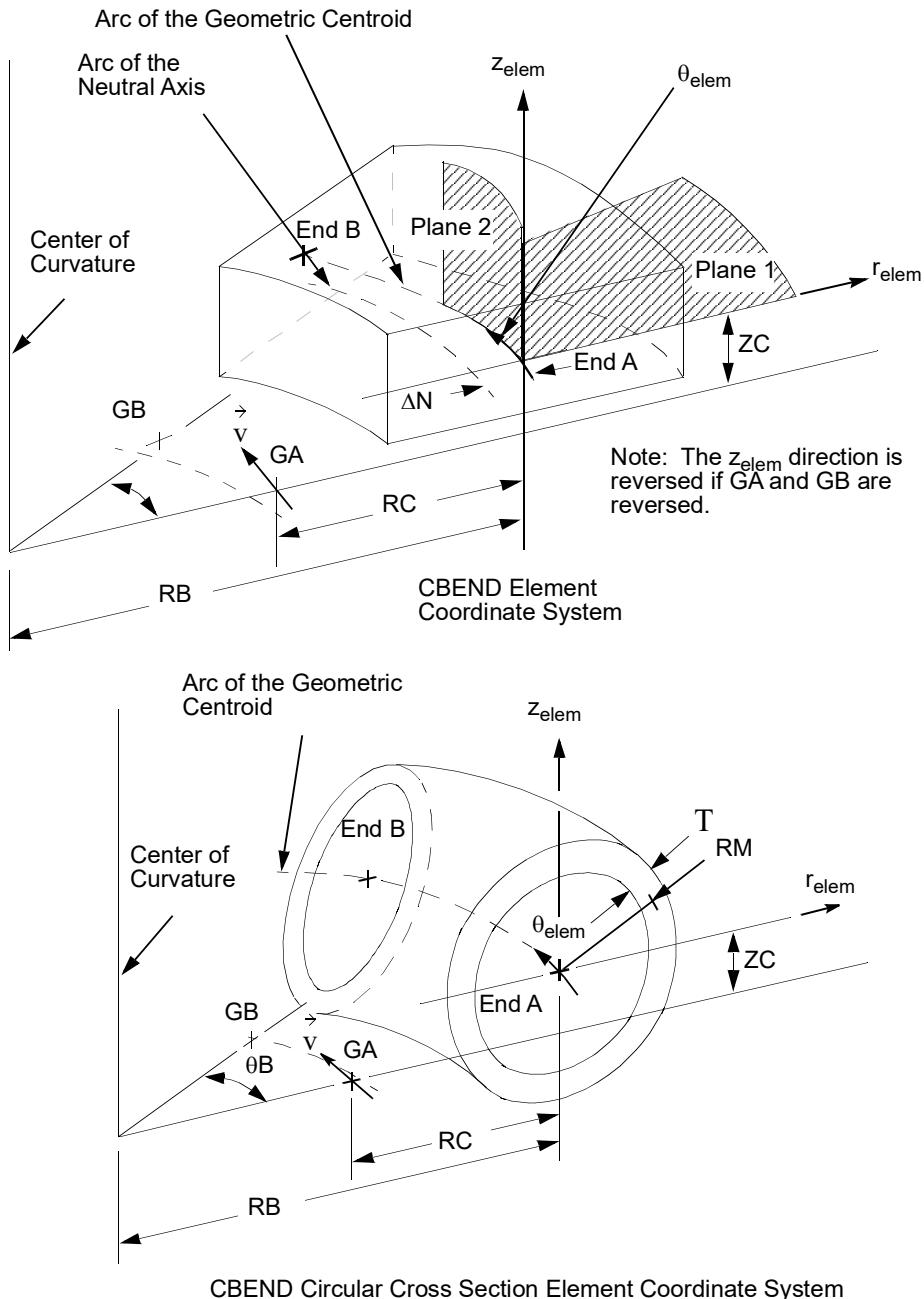


Figure 4-40 The CBEND Element



The Bulk Data entry **PBEND** in the *MSC Nastran Quick Reference Guide* 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:

PBEND

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

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	THETAB	
			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.



Field	Contents
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.

CBEAM3 Three-Node Beam Element

A general three-node beam element has been implemented as a curved one-dimensional Timoshenko beam element so that both the initial curvatures of beam reference axis and the cross-section shears are included in the formulation of linear strain-displacement relations. The geometry of beam axis is specified by the offset vectors from the grid points to the shear centers of the beam cross-sections. The quadratic interpolation is used for both beam axis and the shape functions. When a three-node beam element degenerates to a two-node straight beam element, the linear interpolation is adopted. Variable cross-sectional properties are interpolated quadratically for a three-node and linearly for a degenerate two-node beam element.

Unlike CBEAM, the three-node beam element is developed based on the displacement method, in which the displacements at nodal points are taken as primary variables and the variational principle is applied to minimize the total element energy in formulating element stiffness, consistent mass and differential stiffness matrices.

At each beam element nodal point, there are three translative and three rotational degrees of freedom, respectively. When the beam cross-section torsional warping effect is considered, another degree of freedom, which represents the warping variable, is added to the six nodal degrees of freedom at each grid point. The warping degrees of freedom are represented by either scalar or grid points.

The three-node beam elements can be used in conjunction with higher-order shell elements, such as TRIA6 and QUAD8 that are used to model stiffeners. It can also be applied as an alternative to the existing straight two-node beam element in modeling favorably a structural geometry with initial curvatures.

In addition to normal stresses and strains at beam cross-sections, both shear stresses and strains are also recovered at cross-sectional stress output points. When warping effect is considered, normal stresses caused by cross-sectional bi-moments are computed and output accordingly.

The format of a three-node beam element is specified by the Bulk Data entry **CBEAM3** in the *MSC Nastran Quick Reference Guide*.

CBEAM3

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM3	EID	PID	GA	GB	GC	X1	X2	X3	
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

Example:



CBEAM3	101	2	201	332	1000	1.0	3.5	-2.0	
		3.0		3.0	2.2	-1.0			
	2.5	10.	15.	20.0	206	301	312		

Alternate Format and Example:

CBEAM3	EID	PID	GA	GB	GC	G0			
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

CBEAM3	101	2	201	332	1000	105			
		3.0			2.2	1.0			
	2.5	10.	15.	20.0	206	301	312		

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM3, PBEAML or PBMSECT entries. (Integer > 0; Required)
GA, GB, GC	Grid point identification numbers of connection points. GA and GB are grid point identification numbers at the two ends of the beam element while GC is the one at the grid point in between. (Integer > 0 or blank; GA, GB and GC must be distinct from each other. See Remark 6.)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. The direction of \vec{v} is from GA to G0. \vec{v} is then transferred to End A. (Integer > 0; G0≠GA or GB or GC)
WiA, WiB, WiC	Components of offsets vectors, measured in the displacement coordinate systems at grid points A, B, and C, from the grid points to the points on the axis of shear center. See Remark 8. (Real; Default = 0.0)
TWA, TWB, TWC	Pretwist angles in degrees at A, B, and C, respectively. (Real; Default = 0.0)
SA, SB, SC	Scalar or grid point identification numbers for A, B, and C, respectively. The degrees of freedom at these points are warping variables. (Integer > 0 or blank)

The offset vectors \vec{W}_a , \vec{W}_b , and \vec{W}_c , are measured from grid points GA, GB and GC to the corresponding shear centers, A, B and C, respectively, at the beam cross-sections, as shown in Figure 4-41. Shear centers, A, B and C, are coplanar points. They define a plane in space if they are non-collinear. A quadratic approximation of the locus of beam shear center is uniquely defined by the spatial locations of these three shear centers. In what follows, we will refer the locus of beam shear center as the beam reference axis, or



simply, beam axis. Taking the locus of shear center as the beam primary reference axis is consistent with the CBEAM element. Theoretically, the beam reference axis can be chosen arbitrarily. We also assume that the beam reference axis is a smooth spatial curve. The beam cross-section is perpendicular to the beam axis.

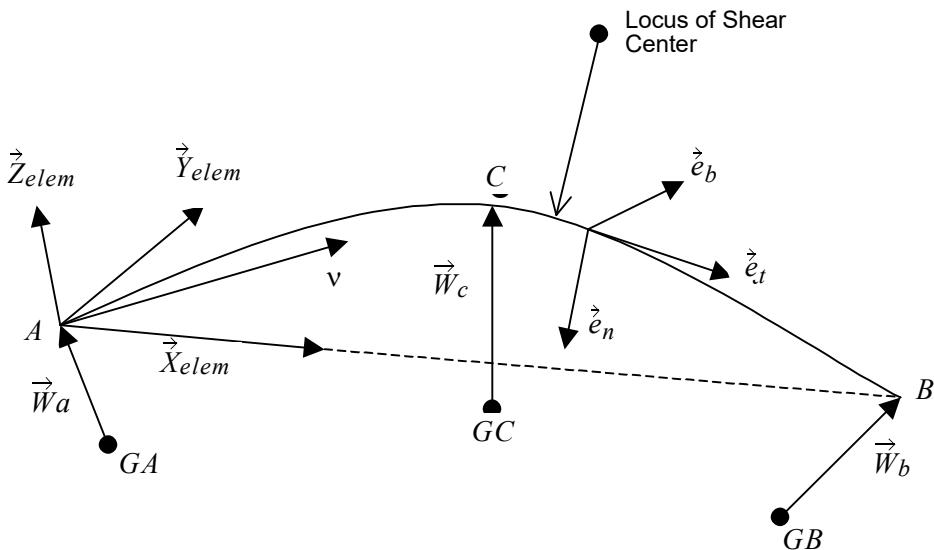


Figure 4-41 Locus of Shear Center of Three-node Beam Element

The element coordinate system (\vec{X}_{elem} , \vec{Y}_{elem} , \vec{Z}_{elem}) is established by the orientation vector v in the same way as what is defined for CBEAM element, as shown in [Figure 4-41](#).

For a spatial curve, such as the beam reference axis, the natural coordinate system is its Frenet-Serret frame (\vec{e}_t , \vec{e}_n , \vec{e}_b), as shown in [Figure 4-41](#). Here \vec{e}_t , \vec{e}_n and \vec{e}_b are the unit tangential, normal and bi-normal vectors, respectively. Tangent vector \vec{e}_t points to the direction from A to B along the beam axis. Vector \vec{e}_n is in the plane defined by these three non-collinear shear centers and points inwardly with respect to the bending of beam axis. Bi-normal vector is perpendicular to both \vec{e}_t and \vec{e}_n by following the right-hand rule of $\vec{e}_b = \vec{e}_t \cdot \vec{e}_n$. When the beam axis is a straight line, i.e., three sectional shear centers are collinear, the Frenet-Serret frame is undefined. Then it will be replaced by the element coordinate system.

To formulate finite element equations and define the beam element properties, a local convected coordinate system (\vec{e}_x , \vec{e}_y , \vec{e}_z) is created. The x -axis is always along the beam axis and its convected x coordinate is measured by the arc-length, s . Its base vector, \vec{e}_x , is taken as same as vector \vec{e}_t . The base vectors of the local coordinate system, \vec{e}_y and \vec{e}_z , are defined in such a way that they rotate an angle ϕ (pre-twist angle), from \vec{e}_n and \vec{e}_b , respectively, in the plane of beam cross-section, as shown in [Figure 4-42](#). The orthogonal



curvilinear coordinate system (\vec{e}_x , \vec{e}_y , \vec{e}_z) is introduced here as the local convected coordinate system, instead of the Frenet-Serret frame, to model the pre-twist of the beam cross-section. When $\phi = 0$, the Frenet-Serret frame (or the element coordinate system if the beam axis is straight) becomes the local convected coordinate system. The angle, ϕ , may vary along the beam axis. The pre-twist angles at three beam cross-sections are given in Bulk Data entry, CBEAM3.

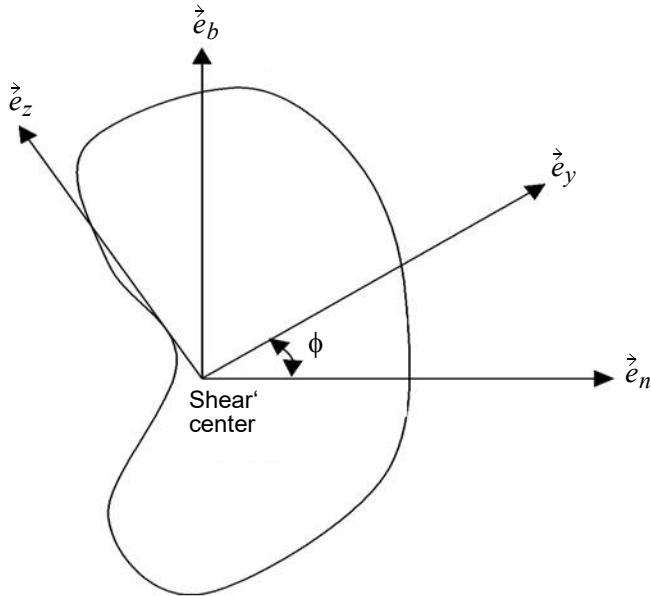


Figure 4-42 Local Coordinate System on a Beam Cross-Section

The properties of a three-node beam element listed in Bulk Data entry, PBEAM3 in the *MSC Nastran Quick Reference Guide*, are referred to the local coordinate system. The output of element stresses, strains and forces is also in the local coordinate system.

PBEAM3

1	2	3	4	5	6	7	8	9	10
PBEAM3	PID	MID	A(A)	IZ(A)	IY(A)	IYZ(A)	J(A)	NSM(A)	
	CY(A)	CZ(A)	DY(A)	DZ(A)	EY(A)	EZ(A)	FY(A)	FZ(A)	
	SO(B)		A(B)	IZ(B)	IY(B)	IYZ(B)	J(B)	NSM(B)	
	CY(B)	CZ(B)	DY(B)	DZ(B)	EY(B)	EZ(B)	FY(B)	FZ(B)	
	SO(C)		A(C)	IZ(C)	IY(C)	IYZ(C)	J(C)	NSM(C)	
	CY(C)	CZ(C)	DY(C)	DZ(C)	EY(C)	EZ(C)	FY(C)	FZ(C)	
	KY	KZ	NY(A)	NZ(A)	NY(B)	NZ(B)	NY(C)	NZ(C)	
	MY(A)	MZ(A)	MY(B)	MZ(B)	MY(C)	MZ(C)	NSIY(A)	NSIZ(A)	
	NSIYZ(A)	NSIY(B)	NSIZ(B)	NSIYZ(B)	NSIY(C)	NSIZ(C)	NSIYZ(C)	CW(A)	



	CW(B)	CW(C)	STRESS						
	WC(A)	WYC(A)	WZC(A)	WD(A)	WYD(A)	WZD(A)	WE(A)	WYE(A)	
	WZE(A)	WF(A))	WYF(A)	WZF(A)	WC(B)	WYC(B)	WZC(B)	WD(B)	
	WYD(B)	WZD(B)	WE(B)	WYE(B)	WZE(B)	WF(B)	WYF(B)	WZF(B)	
	WC(C)	WYC(C)	WZC(C)	WD(C)	WYD(C)	WZD(C)	WE(C)	WYE(C)	
	WZE(C)	WF(C)	WYF(C)	WZF(C)					

Element Properties, Materials and Loads

There are some similarities between PBEAM and PBEAM3 entries in terms of both format and content. Attention, however, should be paid to their differences. The local coordinate system is primarily referred in defining cross-sectional properties, such as area moments and product of inertia, and locations of neutral axis and nonstructural mass center of gravity. Four stress output points on a beam cross-section are also specified in the local coordinate system. Unlike PBEAM, in which you can specify the properties at as many as nine intermediate stations, PBEAM3 requires their definitions at only three grid locations. For a degenerate two-node beam element, those fields related to the mid-grid are ignored. Values of warping function and its gradients at stress output points are required if torsional warping stresses are to be recovered.

Shear effectiveness factors may not be zero. Since the element is formulated on the Timoshenko beam theory and quadratic shape functions are used for interpolation, zero shear effectiveness factors will not automatically lead to a three-node Euler-Bernoulli beam element.

Limitations

There are some limitations in the current implementation with the three-node beam element. The following is not supported:

- Heat Transfer
- X-Y PLOT output
- Random analysis
- PBEAML and PBCOMP related features
- CBEAM related features, such as pin-flags and shear relief

Element forces, moments, stresses and strains, are computed at three beam cross-sections, in the local coordinate system. These three cross-section locations are related to either three grid points or two Gauss integration points and the parametrized origin. Both normal and shear stresses and strains are output at four cross-sectional stress recovery points in the local coordinate system.

Sample output of element forces, stresses and strains are shown in [Examples List](#) (App. A).

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 Bulk Data entry [CQUAD4](#) in the *MSC Nastran Quick Reference Guide* is as follows:

CQUAD4

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

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHELL or PCOMP/PCOMPG 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 Bulk Data entry [CTRIA3](#) in the *MSC Nastran Quick Reference Guide* element is similar.

The material properties for the CQUAD4 and CTRIA3 are specified with the PSHELL or PCOMP/PCOMPG entries. The PCOMP/PCOMPG entry is specifically used for composites and is discussed in [Material Properties and Composites](#).



For all applications other than composites, the PSHELL entry should be used. The format of the Bulk Data entry [PSHELL](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PSHELL

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 Ti.
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.

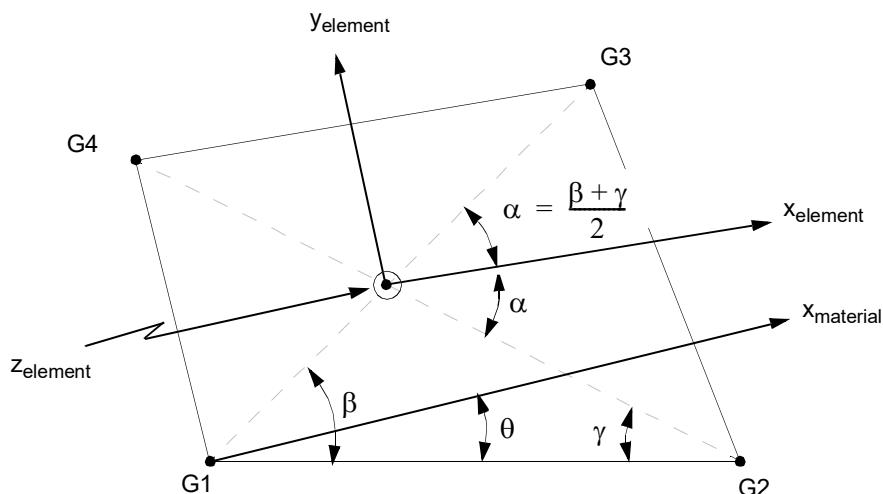
The most common use of the PSHELL entry is to model an isotropic thin plate. The preferred method to define an isotropic plate is to enter the same MAT1 ID for the membrane properties (MID1) and bending properties (MID2) only and leave the other fields blank. For a thick plate, you may also wish to enter 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](#).

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 4-43](#). 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 4-43](#)). 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 4-43](#) CQUAD4 Element Geometry and Coordinate Systems

It can be seen from [Figure 4-43](#) 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](#).

[Figure 4-44](#) 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 output forces and moments generated by the shell 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. Total forces and moments are output by the line and solid elements.

Shown in [Figure 4-44](#) 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 [Sample Output Listings](#). 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.

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. If you only look at the contour plot you can easily be misled if the maximum stress is not shown on the plot. 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](#) and [Model Verification](#).



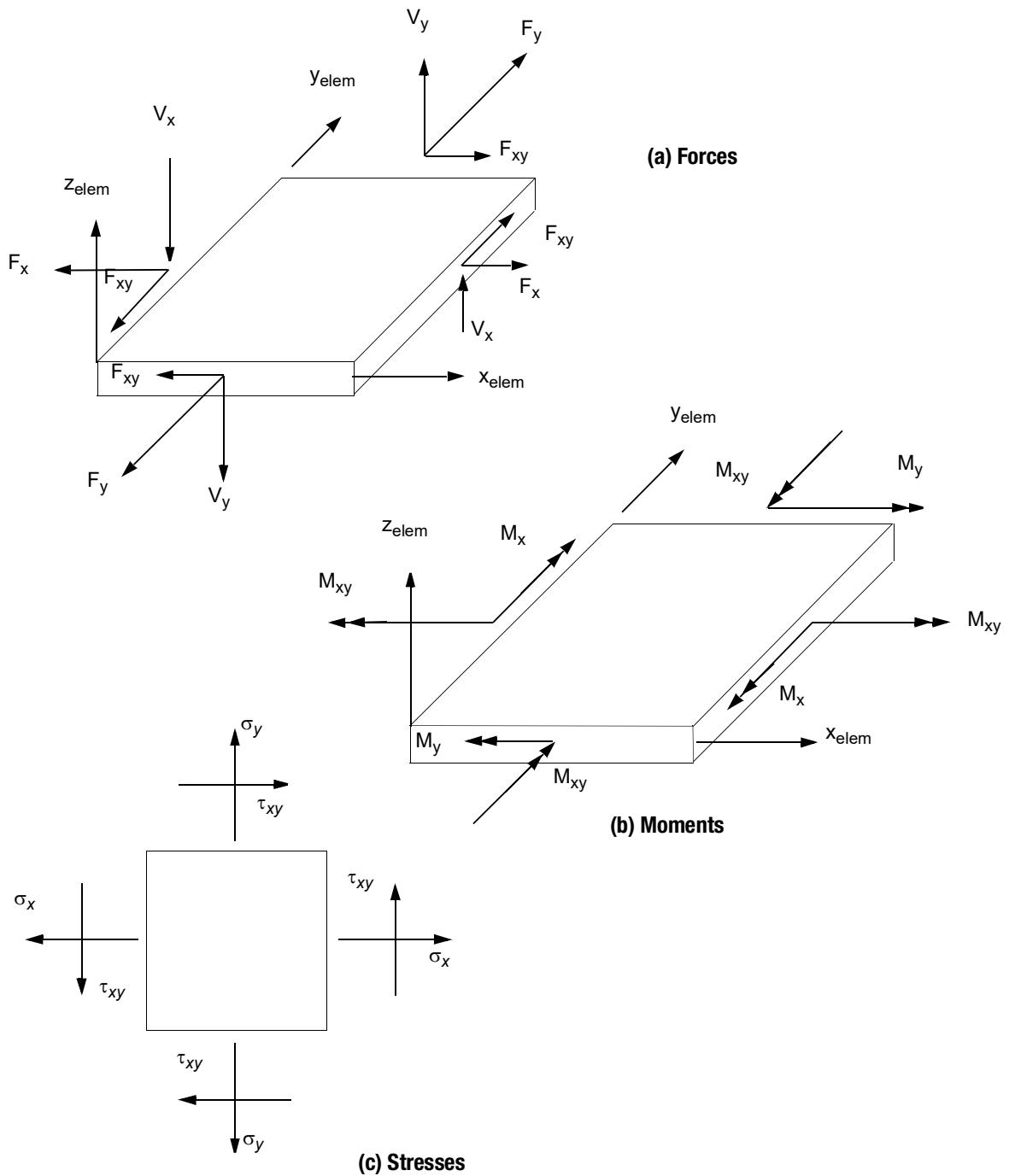


Figure 4-44 Forces, Moments, and Stresses in Plate Elements



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. This zero stiffness results in a singular stiffness matrix and the job fails.

By default, MSC Nastran removes most of these singularities with the **AUTOSPC (Case)=YES** Case Control command being the default and the **PARAM,K6ROT,100**. Bulk Data entry being the default. Singularities not removed by the defaults will show up in a Grid Point Singularity table such as Figure 4-39.

To avoid zero stiffness problems, 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). 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 set the **PARAM,K6ROT,0.0**. When this parameter is used, a table is printed in the output file indicating which degrees of freedom in the model are constrained. An example to force the Grid Point Singularity table output for the CQUAD4 element to model a flat cantilever plate is shown in [Figure 4-45](#). 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.

The default of 100. for K6ROT will eliminate the singularity associated with the normal rotation degree of freedom by applying an artificial stiffness term to the degrees of freedom. 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](#).

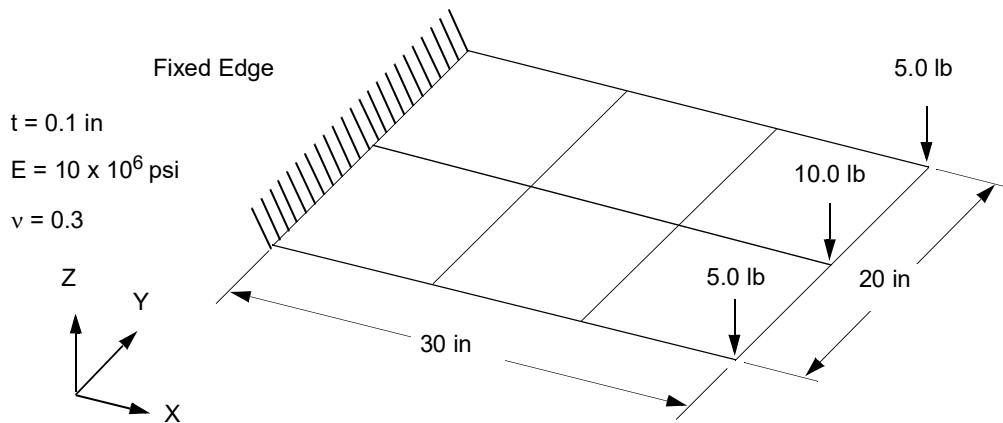


Figure 4-45 Cantilever Plate Model

The input file is shown in [Listing 4-9](#). (See MSC_DOC_DIR/doc/linstat/plate2.doc)



Listing 4-9 Cantilever Plate Input File

```

$ FILENAME - PLATE2.DAT
$
BEGIN BULK
PARAM,K6ROT,0.0
PARAM,POST,-1
$
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

```

AUTOSPC is used to constrain the rotational degrees of freedom normal to the plate. Reviewing the grid point singularity table output, as shown in [Figure 4-46](#), you can see 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](#) describes the sets used in MSC Nastran. The asterisk at the end of the lines indicates that the action was taken.

G R I D P O I N T S I N G U L A R I T Y T A B L E								
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE UNION	NEW USET EXCLUSIVE 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 4-46 Parameter AUTOSPC Output.



The displacement and stress output for the plate model are shown in [Figure 4-47](#).

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 (QUAD 4)											
ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON MISES			
1.361826E+04	-5.000000E-02	NORMAL-X -1.500000E+04	NORMAL-Y -3.580141E+03	SHEAR-XY -6.687832E+02	ANGLE -86.6598	MAJOR -3.541109E+03	MINOR -1.503903E+04				
8.670206E+03	5.000000E-02	1.500000E+04	3.580141E+03	6.687832E+02	3.3402	1.503903E+04	3.541109E+03	1.361826E+04			
3	-5.000000E-02	-9.000000E+03	-7.095209E+02	-1.350937E+02	-89.0667	-7.073201E+02	-9.002200E+03				
4	-5.000000E-02	5.000000E-02	9.000000E+03	7.095209E+02	.9333	9.002200E+03	7.073201E+02	8.670206E+03			
5	-5.000000E-02	-3.000000E+03	-7.205607E+02	3.975338E+02	83.9688	7.625621E+02	-3.042001E+03	3.486400E+03			
6	-5.000000E-02	3.000000E+03	7.205607E+02	-3.975338E+02	-6.0312	3.042001E+03	-7.625621E+02	3.486400E+03			
7	-5.000000E-02	-1.500000E+04	-3.580141E+03	6.687832E+02	86.6598	-3.541109E+03	-1.503903E+04	1.361826E+04			
8	-5.000000E-02	1.500000E+04	3.580141E+03	-6.687832E+02	-3.3402	1.503903E+04	3.541109E+03	1.361826E+04			
9	-5.000000E-02	-9.000000E+03	-7.095209E+02	1.350937E+02	89.0667	-7.073201E+02	-9.002200E+03	8.670206E+03			
10	-5.000000E-02	9.000000E+03	7.095209E+02	-1.350937E+02	-.9333	9.002200E+03	7.073201E+02	8.670206E+03			
11	-5.000000E-02	-3.000000E+03	7.205607E+02	-3.975338E+02	-83.9688	7.625621E+02	-3.042001E+03	3.486400E+03			
12	-5.000000E-02	3.000000E+03	-7.205607E+02	3.975338E+02	6.0312	3.042001E+03	-7.625621E+02	3.486400E+03			

Figure 4-47 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

$$U = U_o + z\theta_y \quad (4-5)$$

$$V = V_o - z\theta_x \quad (4-6)$$

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

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial U_o}{\partial X} \\ \frac{\partial V_o}{\partial Y} \\ \frac{\partial U_o}{\partial Y} + \frac{\partial V_o}{\partial X} \end{Bmatrix} + z \begin{Bmatrix} \frac{\partial \theta_y}{\partial x} \\ -\frac{\partial \theta_x}{\partial y} \\ \frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x} \end{Bmatrix} = \begin{Bmatrix} \varepsilon_x^o \\ \varepsilon_y^o \\ \gamma_{xy}^o \end{Bmatrix} - z \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix} \quad (4-7)$$



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. (See MSC_DOC_DIR/doc/linstat/strain2.doc.) The output is shown in [Figure 4-48](#). 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
```



MEAN STRAIN AND CURVATURE								SUBCASE 1	
ELEMENT ID.	STRAIN CURVATURE	STRAINS IN QUADRILATERAL ELEMENTS (QUAD4)				PRINCIPAL STRAINS (ZERO SHEAR)			VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
1	0.0 -1.000000E+00	1.028946E-05	-2.368387E-06	1.482308E-21	0.0000	1.028946E-05	-2.368387E-06	7.770363E-06	
		-6.173684E-04	1.421053E-04	-7.589415E-19	-90.0000	1.421053E-04	-6.173684E-04	4.662231E-04	
STRAIN AT FIBER LOCATIONS									
ELEMENT ID.	FIBER DISTANCE	STRAINS IN QUADRILATERAL ELEMENTS (QUAD4)				PRINCIPAL STRAINS (ZERO SHEAR)			VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
1	-5.000000E-01 5.000000E-01	-2.983947E-04 3.189737E-04	6.868425E-05 -7.342102E-05	-3.779885E-19 3.809531E-19	-90.0000 0.0000	6.868425E-05 3.189737E-04	-2.983947E-04 -7.342102E-05	2.253412E-04 2.408819E-04	

Figure 4-48 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 4-49](#).

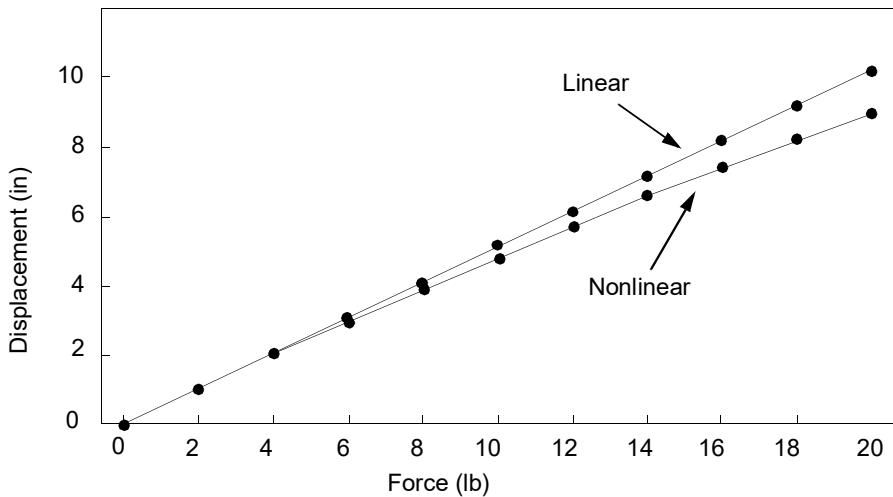


Figure 4-49 Nonlinear vs. Linear Results

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 the *MSC Nastran Reference Guide*.



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 Bulk Data entry **CQUAD8** in the *MSC Nastran Quick Reference Guide* is as follows:

CQUAD8

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/PCOMPG entry.
G1, G2, G3, G4	Identification numbers of connected corner grid points.
G5, G6, G7, G8	Identification numbers of connected edge grid points (optional).
Ti	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 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.

To view results in the material coordinate system, set PARAM,OMID,YES. The default value for OMID is NO.

The parameter 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 Patran, assume element results are in the element coordinate system, thereby producing incorrect values if they are not in that system.

Figure 4-50 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, where as 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 4-51**.

STRAINS IN QUADRILATERAL ELEMENTS (QQUAD4)											OPTION = BILIN	
ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRAINS IN MATERIAL COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			VON MISES			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR				
1	CEN/4	-5.00000E-02	4.380871E-04	3.387215E-04	8.973553E-04	41.8406	8.398243E-04	-6.301572E-05	5.820263E-04			
1		5.00000E-02	-4.380871E-04	-3.387215E-04	-8.973553E-04	48.1594	6.301572E-05	-8.398243E-04	5.820263E-04			
		-5.00000E-02	4.592682E-04	3.387215E-04	7.979897E-04	40.7048	8.025165E-04	-4.526860E-06	5.365264E-04			
		5.00000E-02	-4.592682E-04	-3.387215E-04	-7.979897E-04	-49.2952	4.526860E-06	-8.025165E-04	5.365264E-04			
a. Strain Output Fragment with OMID = YES (Key features in bold)												
STRAINS IN QUADRILATERAL ELEMENTS (QQUAD4)											OPTION = BILIN	
ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			VON MISES			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR				
1	CEN/4	-5.00000E-02	-6.027338E-05	8.370820E-04	9.936564E-05	86.8406	8.398243E-04	-6.301573E-05	5.820264E-04			
1		5.00000E-02	6.027338E-05	-8.370820E-04	-9.936564E-05	-3.1594	6.301573E-05	-8.398243E-04	5.820264E-04			
		-5.00000E-02	1.338338E-20	7.979897E-04	1.205468E-04	85.7048	8.025166E-04	-4.526860E-06	5.365264E-04			
		5.00000E-02	1.338338E-20	-7.979897E-04	-1.205468E-04	-4.2952	4.526860E-06	-8.025166E-04	5.365264E-04			
b. Strain Output Fragment with OMID=NO (Key features in bold)												

Figure 4-50 Effect of Parameter OMID on Element Strain Output



SUBCASE 1 FORCES IN QUADRILATERAL ELEMENTS (QUAD4)										OPTION = BILIN	
RESULTS IN MATERIAL COORD SYSTEM											
ELEMENT ID	GRID-ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -			
		FX	FY	FXY	MX	MY	MXY	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 4-51 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.

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 OMID parameter 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 CQUADR and CTRIAR Elements

The Bulk Data entries [CQUADR](#) and [CTRIAR](#) are identical to the CQUAD4 and CTRIAR, however 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. It is recommended 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. Using System Cell [QRMETH \(370\)](#) you can convert from one form to the other for comparisons. For a more detailed description of the CQUADR and CTRIAR entries, see [QUADR and TRIAR Elements](#) in the *MSC Nastran Reference Guide*.

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 Bulk Data entry [CSHEAR](#) in the *MSC Nastran Quick Reference Guide* is as follows:

CSHEAR

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 Bulk Data entry [PSHEAR](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PSHEAR

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 4-52](#).



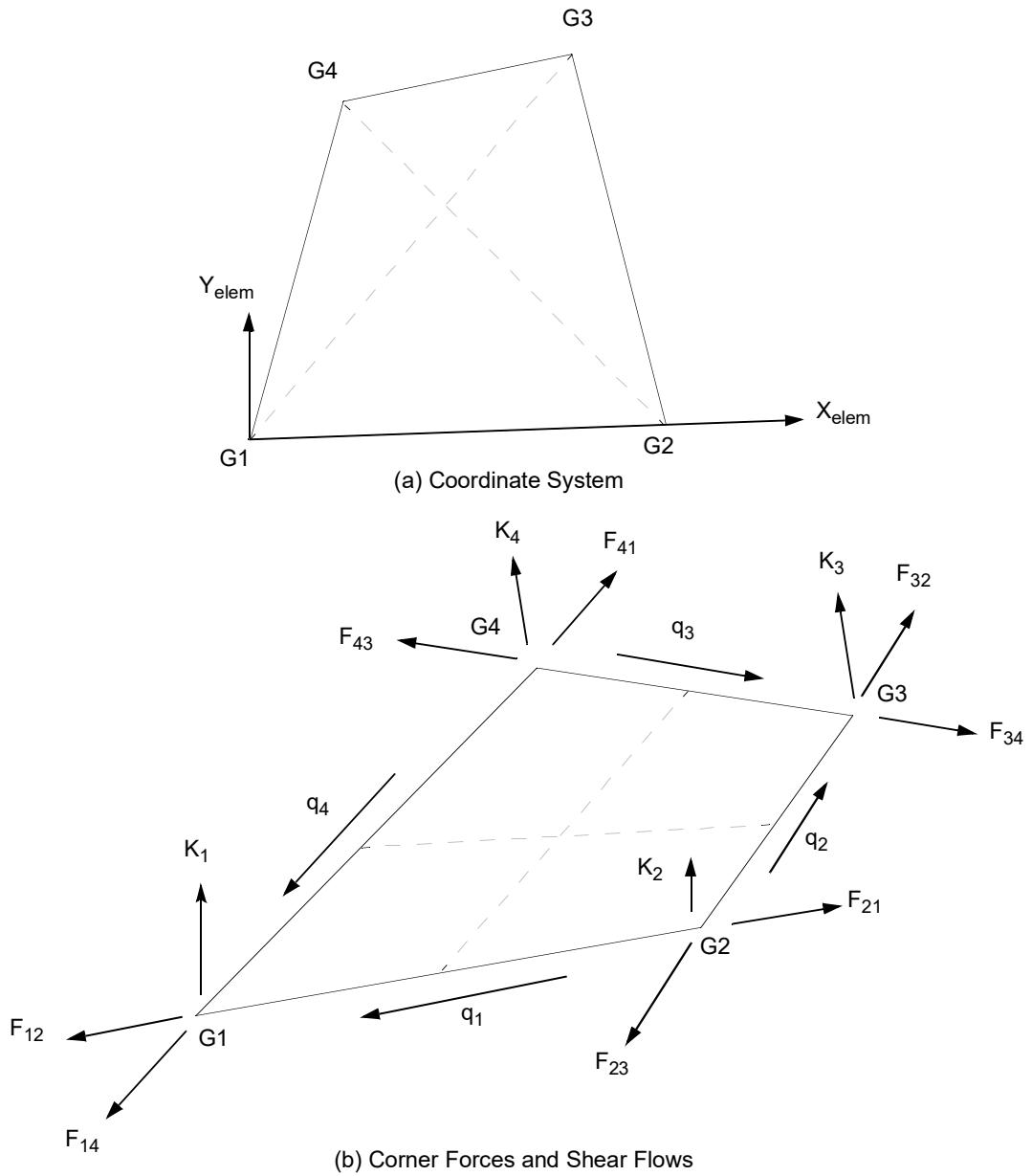


Figure 4-52 Coordinate System and Element Forces for the Shear Panel

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 Bulk Data entries [CRAC2D](#) and [ADUM8](#) in the *MSC Nastran Quick Reference Guide* are as follows:

CRAC2D

1	2	3	4	5	6	7	8	9	10
CRAC2 D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

ADUM8

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. 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 Bulk Data entry [PRAC2D](#) in the *MSC Nastran Quick Reference Guide*, and its format is as follows:

PRAC2D

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



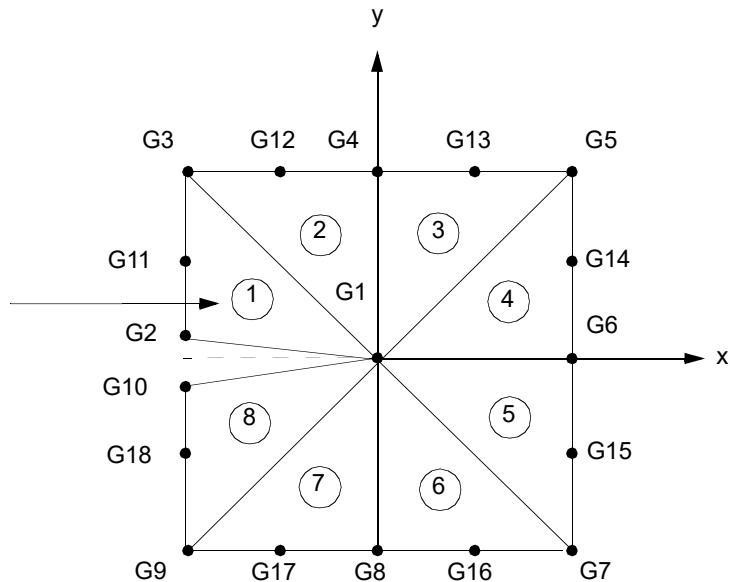
Field	Contents
PID	Property identification number.
MID	Material identification number.
T	Element thickness.
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress.
NSM	Non-structural mass per unit area.
GAMMA	Exponent used in the displacement field.
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated.

The CRAC2D geometry is depicted in [Figure 4-53](#). 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.

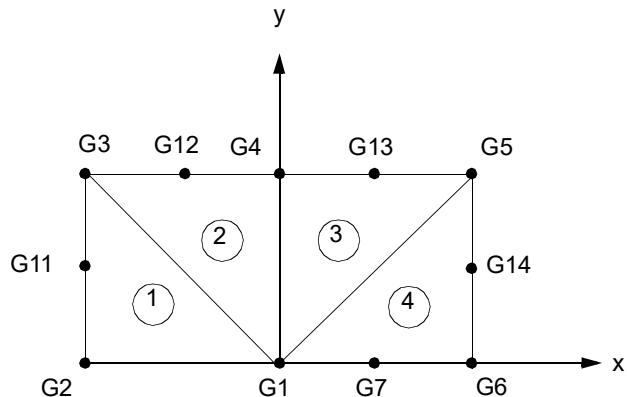
The CRAC2D element is based on a two-dimensional formulation, but it may be used for three-dimensional planar structures. Deviation from a planar element is checked and any significant deviation results in a printed error message.



Basic Triangular Element



(a) Quadrilateral Crack Element



(b) Symmetric Half-Crack Option

Figure 4-53 The CRAC2D Shell Crack Tip Element



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, CPYRAM, 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, CPYRAM 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, CPYRAM 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), CPYRAM (a pyramid element with five faces) and CTETRA tetrahedral 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, CPYRAM and CTETRA entries. For discussion, the format of the Bulk Data entry [CHEXA](#) in the *MSC Nastran Quick Reference Guide* is included below. The formats of the CPENTA, CPYRAM and CTETRA are similar to that of the CHEXA and therefore are not included.

CHEXA

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.



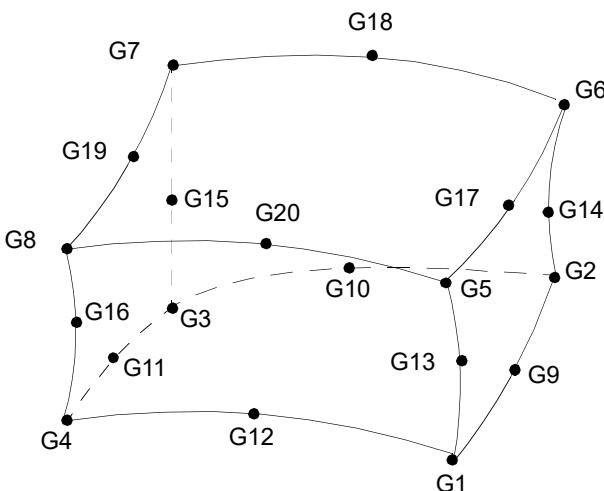


Figure 4-54 CHEXA Element Connection

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 have MSC Nastran automatically identify and constrain them, i.e., the default for the AUTOSPC Case Control command. 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](#).

The properties of the solid elements are entered on the PSOLID entry. The format of the Bulk Data entry **PSOLID** in the *MSC Nastran Quick Reference Guide* is as follows:

PSOLID

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORD M	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](#) (Ch. 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.

A laminate composite shell or solid can be analyzed by using the [PCOMPLS](#) entry referencing a CHEXA element, but only in SOL 400.

The element coordinate system of the solid elements can be affected by the skew of the elements. 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 4-55](#).



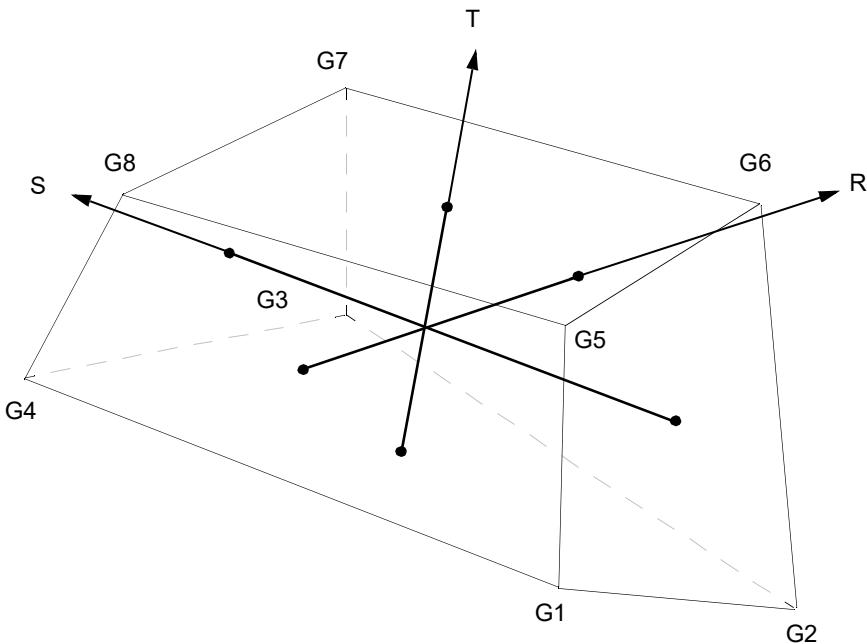


Figure 4-55 CHEXA Element Coordinate System

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 the 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 method for the element coordinate system definition 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. 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](#). The dimensions of the tapered beam are shown in [Figure 4-19](#). 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 4-56](#). 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 4-56](#) are deformed plots resulting from the end load.

The input file is not included due to its large size, but it is available on the deliver media. (See `MSC_DOC_DIR/doc/linstat/solid1.dat`.)

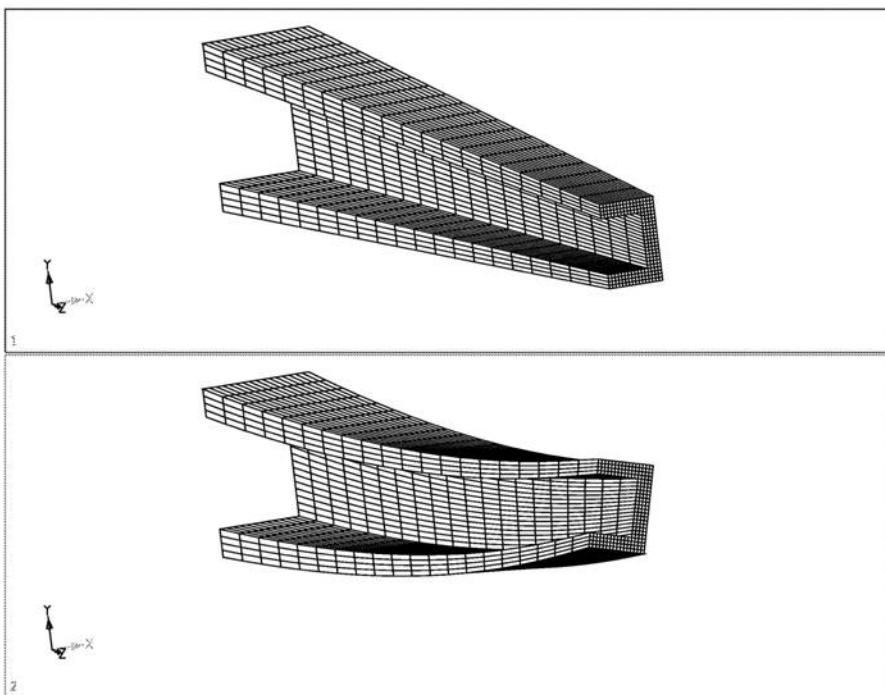


Figure 4-56 Tapered Beam Solid Element Model

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 Bulk Data entry **CTRIA6** in the *MSC Nastran Quick Reference Guide* 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 Bulk Data entries **CRAC3D** and **ADUM9** in the *MSC Nastran Quick Reference Guide* are as follows:

CRAC3D

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.

ADUM9

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 4-57](#).

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 Bulk Data entry **PRAC3D** in the *MSC Nastran Quick Reference Guide* has the following format:

PRAC3D

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.



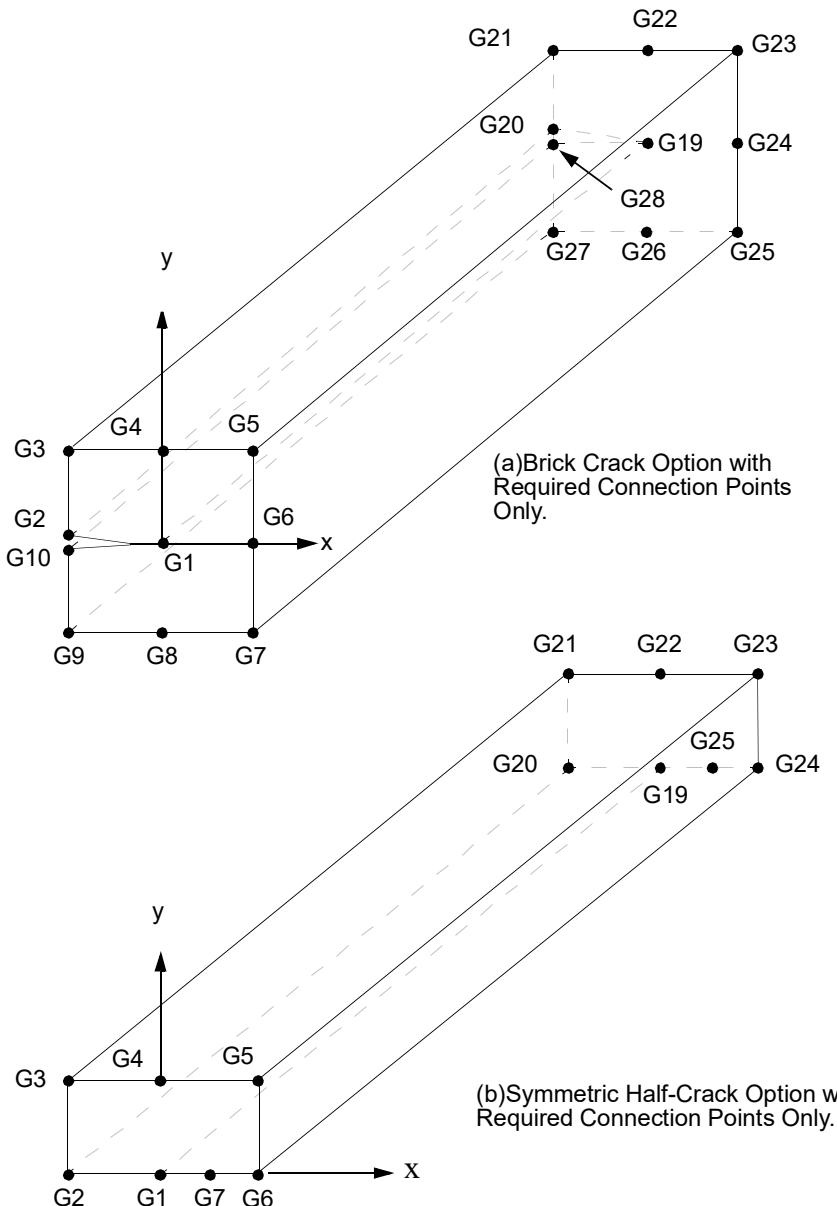


Figure 4-57

CRAC3D Solid Crack Tip Element



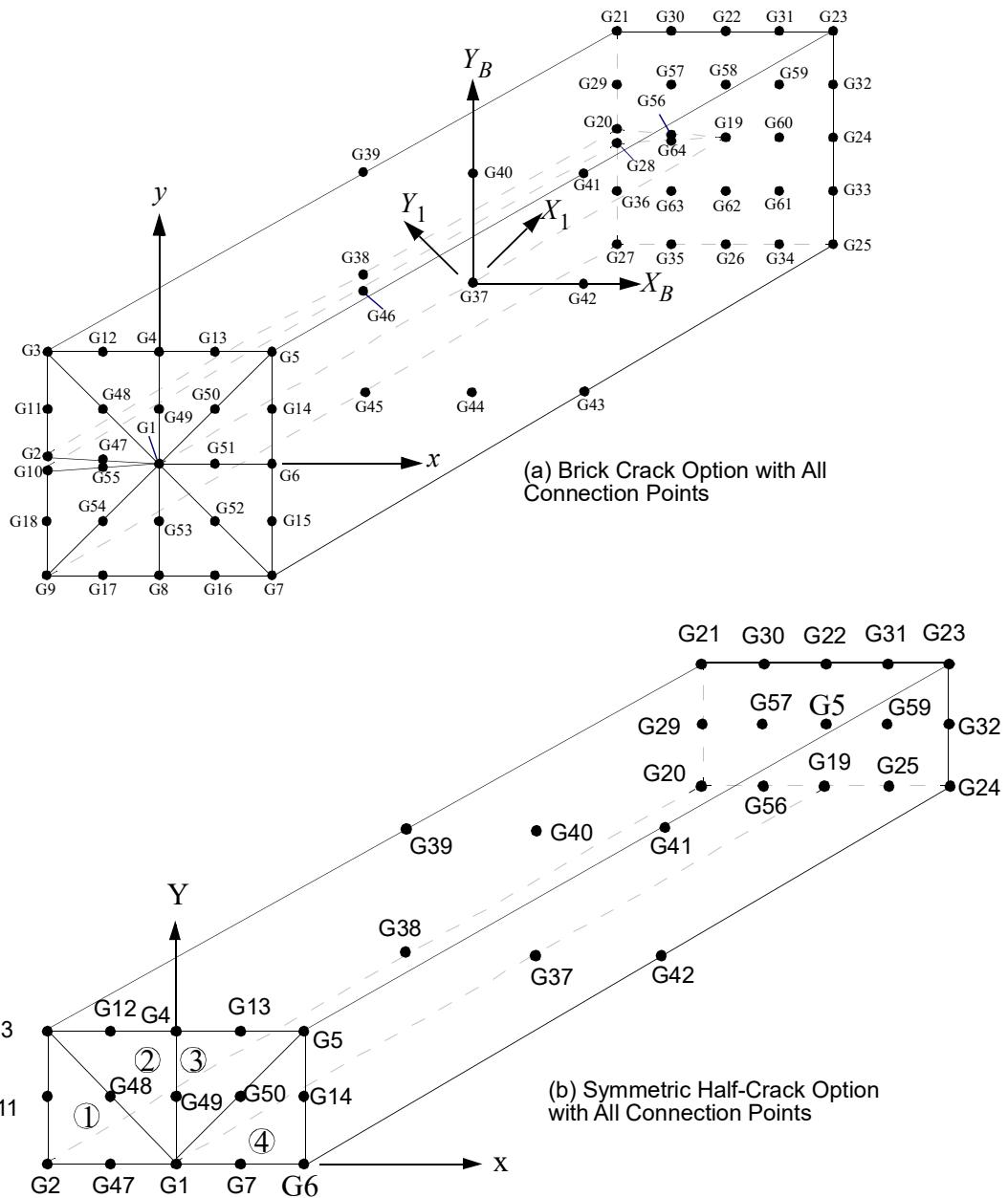


Figure 4-57 CRAC3D Solid Crack Tip Element (continued)



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:

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

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

$$KZ = \begin{bmatrix} KZ_{11} & \dots & \dots & \dots \\ KZ_{21} & KZ_{22} & \dots & \dots \\ KZ_{31} & KZ_{32} & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix}$$

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

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

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



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 Bulk Data entry [GENEL](#) in the *MSC Nastran Quick Reference Guide* is as follows:

GENEL

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

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

	"UD"		UD1	CD1	UD2	CD2	-etc.-		
--	------	--	-----	-----	-----	-----	--------	--	--

UDn - 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.-				

KZmm - 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.-		

Smn - The last item in the S matrix appears in one of fields 2 through 9.



Field	Contents
EID	Unique element identification number.
UIi, Cli, 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 Cli 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 4-58](#). 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.

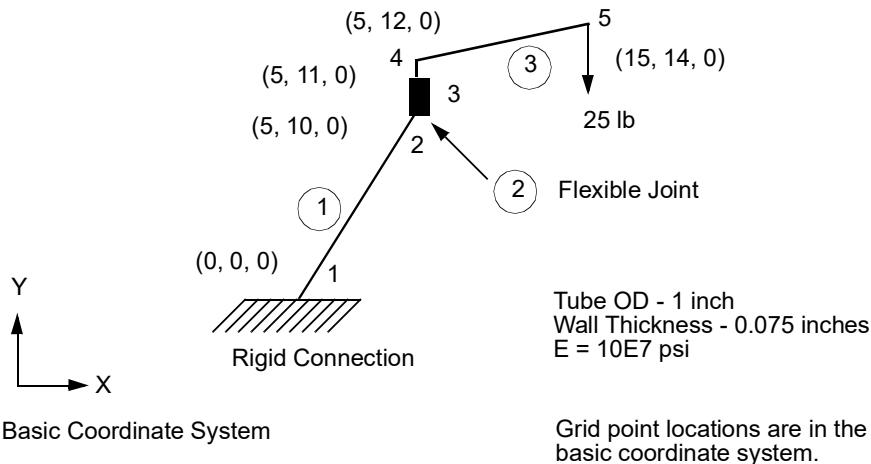
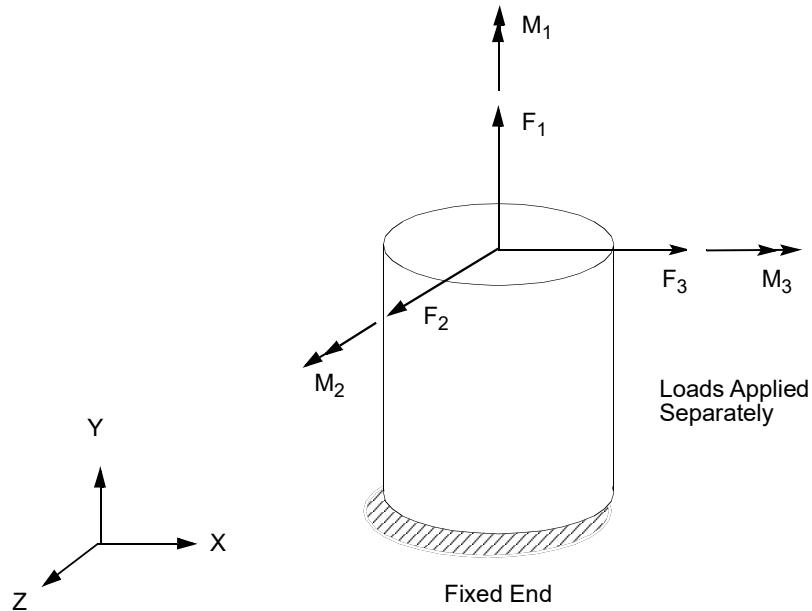


Figure 4-58 Robotic Arm with Joint

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 4-59](#).





Orientation of the Joint in the Model

Figure 4-59 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 4-11. (See sample problem MSC_DOC_DIR/doc/linstat/genel1.dat.)



Listing 4-11 GENEL Element Input File

```

$  

$FILENAME - GENEL1.DAT  

$  

CORD2R 1 0. 0. 0. 0. 1. 0. 0.  

       0. 1. 0.  

GENEL 99 2 4 2 1 2 2 2 3  

       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, 482](#) provides you with an alternative approach of inputting structure matrices using the DMIG entries.

Connector Elements

Introduction

MSC Nastran offers 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, RJOINT, 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 Connector elements address and solve such modeling issues. Connections can be established with ease between points, elements, patches, seam lines, dissimilar meshes, or any of their combinations. The



connector elements are general in purpose, easy to generate and always satisfies the condition of rigid body invariance.

The CWELD, CFAST and CSEAM elements all have a number of connectivity options. For each of these elements and their property entries (PWELD, PFAST and PSEAM) plus the SWLDPRM entry of parameters; the Quick Reference Guide should be reviewed for a complete description of the entry and use.

CWELD Element

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 through the surfaces to be attached, uniquely determine the weld elements' location and geometry. CWELD element connectivity is defined using the Bulk Data entry **CWELD** in the *MSC Nastran Quick Reference Guide*. Defines a weld or fastener connecting two surface patches or points. Large displacement and large rotational effects are supported when using MSC Nastran SOL 400. The following is a brief description of the various CWELD connectivity options:

CWELD

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB							
	XS	YS	ZS						

- Property-to-Property

PARTPAT, the user defines shell A and B by property id PSIDA and PSIDB. With PARTPAT, all shell elements (up to 3x3) are connected which fall into the cross sectional area of the spot weld.

Format ELPAT:

CWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB							
	XS	YS	ZS						

- Element-to-Element

ELPAT, where SHIDA and SHIDB are element ids of shell A and B, respectively. With ELPAT, the connectivity is extended to all neighboring shell elements (up to 3x3) which fall into the cross sectional area of the spot weld.



Format ELEMID:

CWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB							

- Patch-to-Patch

ELEMID, option defines a connection between two shell elements:

Format GRIDID:

CWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

- Point-to-Patch

GRIDID format can be used to define the connection of a point to a surface patch.

Format ALIGN:

CWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	
-------	------	------	--	---------	----	----	--	------	--

- Point-to-Point

ALIGN format defines a point-to-point connection:

CWELD element properties are defined with the Bulk Data entry PWELD in the *MSC Nastran Quick Reference Guide*, the fields can vary with the type of CWELD element selected.

PWELD

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

SWLDPRM in the *MSC Nastran Quick Reference Guide* overrides default values of parameters for CFAST, CWELD, and CSEAM connectivity search.

SWLDPRM

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

Example

In the following example, two cylindrical shell segments are connected with 4 CWELD elements at the 4 corners of the overlapping sections, see [Figure 4-60](#). The results of the patch-to-patch connection (see `MSC_DOC_DIR/doc/linstat/cweld101b.dat`) are compared to the results of the point-to-point connection. For the patch-to-patch connection (see `MSC_DOC_DIR/doc/linstat/cweld101a.dat`), 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 (1.8842 versus 1.9233). 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. Output results from the CWELD are formatted the same as the CBEAM.

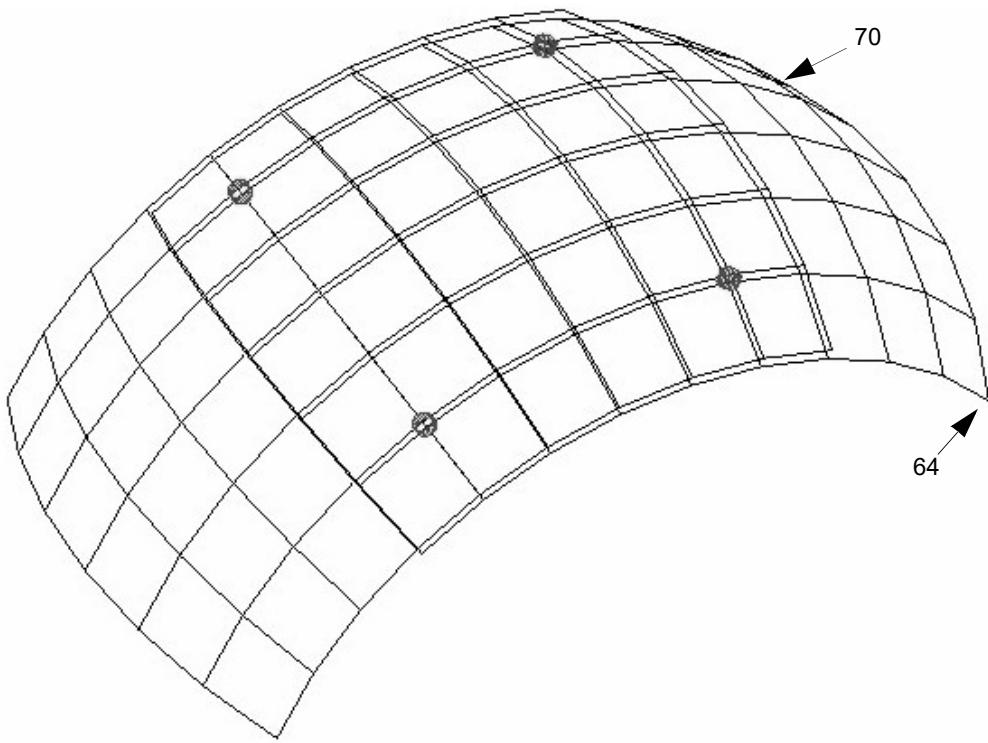


Figure 4-60 Two Spherical Segments with Spot Welds at the Four Corners

CFAST Element

The CFAST element defines a fastener with material orientation connecting two surface patches. This extends weld element capabilities by adding a flexible, user-defined connection between either two surface patches or two shell elements. The PFAST entry gives the option of specifying longitudinal and rotational stiffness, a lumped mass, and damping along a defined orientation. The format for the [CFAST](#) Bulk Data entry is as follows:



CFAST

1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

The format for the **PFAST** Bulk Data entry is as follows:

PFAST

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE	ALPHA	TREF	COINL		

Parameters for CFAST

These two parameters enable you to randomly remove a percentage of CFAST elements for failure studies.

CFDIAGP	Default = NO If YES, randomly deleted CFAST elements will be printed. (See CFRANDEL)
CFRANDEL	Default = 0. Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted.

Example

An example is two 3x3 structures made up of 9 quad4 elements each laid over each other and connected with a single CFAST element located at the center of each structure.

The diameter of the fastener was chosen so that the fastener would pick up 4 different elements for patch A and four different elements for patch B. Thus for this weld, a total of 32 different grids are involved in the connector.

Part of the input (See MSC_DOC_DIR/doc/linstat/f_qa.dat) is shown:

```
sol 101
cend
load = 10
set 7 = 777
force=7
begin bulk
swldprm,prtsw,1
$c
cfast,777,1000,elem,105,5,999
pfast,1000,30.,,1.18+8,4.53+8,4.53+8,5.09+9
,6.62+9,6.62+9
```



Figure 4-61 shows the CFAST element in red, with the new grids created and then the mpc connections between the CFAST grids and the CQUAD4 grids.

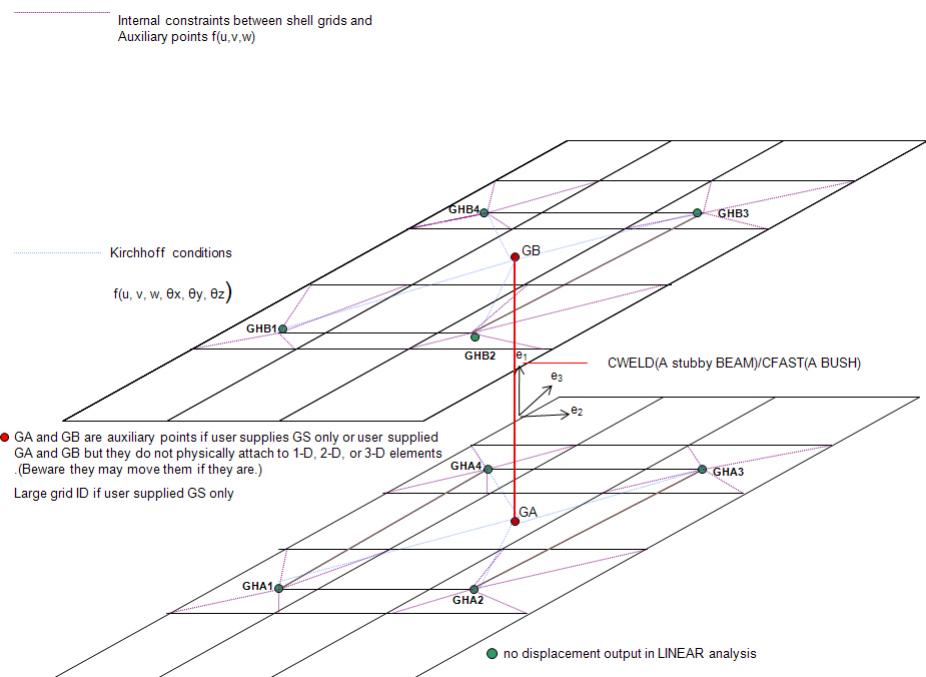


Figure 4-61 CFAST with MPC Connections

The following listings show relevant output for the CFAST element and a typical force output; and Figure 4-62 shows a displacement plot of the two fastened plates after loading. Output results from the CFAST are formatted the same as the CBUSH.

```

CFAST EID=    777 WITH FORM=ELPAT OR PARTPAT
AUXILIARY POINTS= ( 1.7066E+00, 1.7066E+00, 0.0000E+00) ( 2.8293E+01, 1.7066E+00, 0.0000E+00)
                  ( 2.8293E+01, 2.8293E+01, 0.0000E+00) ( 1.7066E+00, 2.8293E+01, 0.0000E+00)
                  ( 1.7066E+00, 1.7066E+00, 1.0000E-02) ( 2.8293E+01, 1.7066E+00, 1.0000E-02)
                  ( 2.8293E+01, 2.8293E+01, 1.0000E-02) ( 1.7066E+00, 2.8293E+01, 1.0000E-02)

NUMBER OF TIMES GS MOVES= 0
NUMBER OF TIMES DA IS REDUCED= 0
ANGLE BETWEEN TWO SHELL NORMALS= 0.00
GS=( 1.500E+01, 1.500E+01, 5.000E-03) GA=( 1.500E+01, 1.500E+01, 0.000E+00) GB=( 1.500E+01, 1.500E+01, 1.000E-02)
T_BE MATRIX: 0.0000E+00 1.0000E+00 0.0000E+00
              0.0000E+00 0.0000E+00 1.0000E+00
              1.0000E+00 0.0000E+00 0.0000E+00
PATCH A: EID= 101 GIDS= 201 202 206 205 0 0 0 0
          EID= 103 GIDS= 203 204 208 207 0 0 0 0
          EID= 109 GIDS= 211 212 216 215 0 0 0 0
          EID= 107 GIDS= 209 210 214 213 0 0 0 0
PATCH B: EID= 1 GIDS= 101 102 106 105 0 0 0 0
          EID= 3 GIDS= 103 104 108 107 0 0 0 0
          EID= 9 GIDS= 111 112 116 115 0 0 0 0
          EID= 7 GIDS= 109 110 114 113 0 0 0 0

```



FORCES IN FASTENER ELEMENTS (CFAST)						
ELEMENT_ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
777	0.0	2.000000E+03	-5.170960E-10	6.467096E-10	1.722580E-07	1.000000E+01

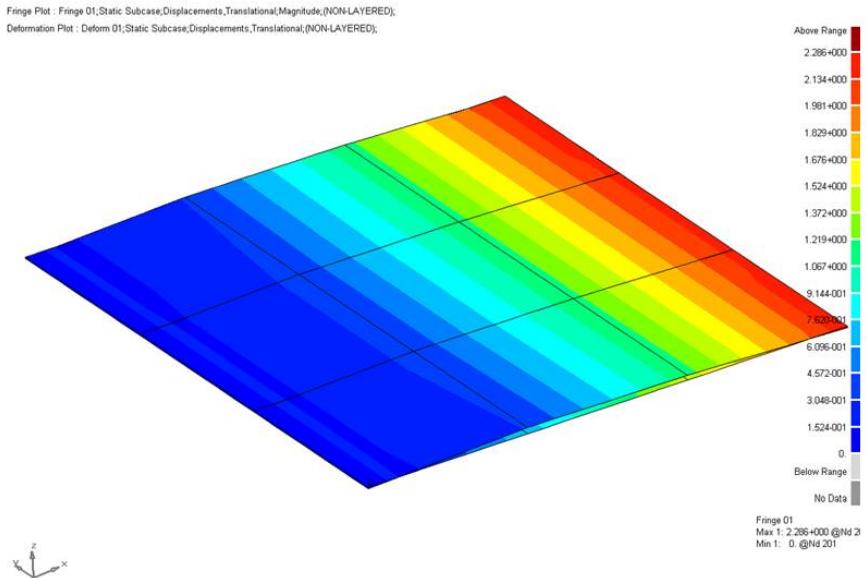


Figure 4-62 Displacement Output of CFAST Element

Example of Projected Grids

The displacements of the projected grids GA and GB for CWELD elements with GRIDID, ELEMID, ELPAT or PARTPAT format and for CFAST elements are computed to display the relationship between these elements and their connecting shell elements. If GA or GB is not specified for CFAST elements or CWELD elements with ELPAT or PARTPAT option, the program will create a grid internally, with the grid ID number starting from OSWPPT+1 (OSWPPT is a parameter specified by PARAM Bulk Data entry). The user may request a positive PRTSW parameter (1, 2, 11, or 12) in SWLDPRM Bulk Data entry to view the grid ID of GA or GB in the diagnostic output.

The displacements of GA and GB are requested using the **CONNECTOR** keyword of the **DISPLACEMENT** Case Control command. The output is integrated with the displacements of the general grid points.

This example demonstrates the various displacement output requests and their results for a small model with two CWELD elements.

Part of the input file (See **MSC_DOC_DIR/doc/linstat/swteiden.dat**) follows:

```
SOL 101
TIME 60
CEND
TITLE= two elements, identical location for GA, GB, GS
OLOAD= ALL
```



```

FORCE = ALL
SUBCASE 1
    SUBTITLE= shear the weld
    SPC= 1
    LOAD= 1
    DISP= ALL
SUBCASE 2
    SUBTITLE= in plane twist
    set 21 = 1002,1003,2011,thru,2014
    spc= 1
    LOAD= 2
    DISP(CONN=ALL)=21
SUBCASE 3
    SUBTITLE= upper bending
    set 32 = 4
    set 33 = 1012,1013,2001,thru,2004
    spc= 1
    LOAD= 3
    DISP(CONN=32)=33
BEGIN BULK
$
$ Grids of lower shell
grid, 1001, , 0., 0., 0.
grid, 1002, , 20., 0., 0.
:
grid, 1013, , 20., 10., 5.
grid, 1014, , 0., 10., 5.
$ Grids of upper shell
grid, 2001, , 0., 0., 0.0
grid, 2002, , 20., 0., 0.0
:
grid, 2013, , 20., 10., 6.0
grid, 2014, , 0., 10., 6.0
$ spot weld grid
grid, 3001, , 10.0, 5.0, 0.0
grid, 3011, , 10.0, 5.0, 10.0
$ quad4s
cquad4, 4001, 10, 1001, 1002, 1003, 1004
cquad4, 5001, 10, 2001, 2002, 2003, 2004
cquad4, 4011, 10, 1011, 1012, 1013, 1014
cquad4, 5011, 10, 2011, 2012, 2013, 2014
:
$ property and material
pshell, 10, 10, 1.0, 10
mat1, 10, 2.e+5 , , 0.3, 0.785e-8
$
$ spot welds
$cweld, 4, 4, 3001, elemid, , , , +cw4
+cw4, 4001, 5001
$cweld, 5, 4, 3011, elemid, , , , +cw5
+cw5, 4011, 5011
$pweld, 4, 10, 5.0
$enddata

```

The displacement results are shown as follows:



0

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
1001	G	0.0	0.0	0.0	0.0	0.0	0.0
1002	G	4.887498E-03	5.624979E-04	0.0	0.0	0.0	1.406245E-05
1003	G	4.887498E-03	-5.624979E-04	0.0	0.0	0.0	-1.406245E-05
1004	G	0.0	0.0	0.0	0.0	0.0	0.0
1011	G	0.0	0.0	0.0	0.0	0.0	0.0
1012	G	4.887498E-03	5.624979E-04	-7.685000E-01	-6.750000E-03	5.864999E-02	1.406245E-05
1013	G	4.887498E-03	-5.624979E-04	-7.685000E-01	6.750000E-03	5.864999E-02	-1.406245E-05
1014	G	0.0	0.0	0.0	0.0	0.0	0.0
2001	G	6.058339E-04	3.750000E-04	0.0	0.0	0.0	7.733938E-19
2002	G	5.605834E-03	3.750000E-04	0.0	0.0	0.0	1.084202E-1
2003	G	5.605834E-03	-3.750000E-04	0.0	0.0	0.0	1.191827E-18
2004	G	6.058339E-04	-3.750000E-04	0.0	0.0	0.0	7.6233363E-19
2011	G	3.907157E-02	3.750000E-04	8.148733E-04	-2.562901E-16	3.850649E-02	2.836059E-17
2012	G	4.407157E-02	3.750000E-04	-7.693148E-01	-5.232793E-15	3.850649E-02	2.851452E-17
2013	G	4.407157E-02	-3.750000E-04	-7.693148E-01	-5.356609E-15	3.850649E-02	2.956961E-17
2014	G	3.907157E-02	-3.750000E-04	8.148733E-04	-2.636780E-16	3.850649E-02	2.803595E-17
3001	G	0.0	0.0	0.0	0.0	0.0	0.0
3011	G	0.0	0.0	0.0	0.0	0.0	0.0
101000001	G	2.443749E-03	8.917563E-18	0.0	0.0	0.0	8.402567E-19
101000002	G	3.105834E-03	8.944668E-18	0.0	0.0	0.0	9.774760E-19
101000003	G	2.443749E-03	2.517788E-16	-3.842500E-01	-2.775558E-15	3.842500E-02	2.303930E-17
101000004	G	4.157158E-02	3.029749E-15	-3.842500E-01	-2.789435E-15	3.850649E-02	2.786400E-17

0

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
1002	G	1.499616E-02	3.324066E-02	0.0	0.0	0.0	2.330633E-03
1003	G	-1.499616E-02	3.324066E-02	0.0	0.0	0.0	2.330633E-03
2011	G	6.107051E-03	-8.968769E-03	-1.542189E-16	3.303767E-17	1.479144E-16	1.890160E-03
2012	G	2.110705E-02	4.220943E-02	-3.132043E-15	5.117434E-17	1.498367E-16	3.390160E-03
2013	G	-2.110705E-02	4.220943E-02	-2.615099E-15	5.202362E-17	1.383761E-16	3.390160E-03
2014	G	-6.107051E-03	-8.968769E-03	1.645747E-16	3.118436E-17	1.398878E-16	1.890160E-03
101000001	G	6.505213E-18	1.662033E-02	0.0	0.0	0.0	1.580825E-03
101000002	G	6.722053E-18	1.662033E-02	0.0	0.0	0.0	2.640160E-03
101000003	G	3.035766E-18	1.662033E-02	-1.434168E-15	4.168935E-17	1.434168E-16	1.580825E-03
101000004	G	1.461505E-16	1.662033E-02	-1.434197E-15	4.178690E-17	1.439374E-16	2.640160E-03

0

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
1012	G	3.460202E-16	9.473882E-16	-1.537000E+00	-1.350000E-02	1.173000E-01	6.294140E-17
1013	G	-4.478185E-16	9.592263E-16	-1.537000E+00	1.350000E-02	1.173000E-01	6.130147E-17
2001	G	0.0	0.0	8.021409E-04	-1.965454E-14	7.693148E-02	0.0
2002	G	0.0	0.0	-1.537827E+00	6.397660E-15	7.693148E-02	0.0
2003	G	0.0	0.0	-1.537827E+00	6.758049E-15	7.693148E-02	0.0
2004	G	0.0	0.0	8.021409E-04	-2.021495E-14	7.693148E-02	0.0
101000001	G	0.0	0.0	-7.685000E-01	-6.591949E-15	7.685000E-02	0.0
101000002	G	0.0	0.0	-7.685127E-01	-6.664110E-15	7.693148E-02	0.0

CSEAM Element

CSEAM is a connector elements which defines a seam line to connect two surface patches. To define the seam weld, you must select two surface patches by their property IDs and specify the width and the thickness of the seam. You also have to specify start point “GS” and end point “GE” of the seam segment. These two points do not have to lie on or in between the shell planes. The program will project these two points onto the selected surface patches to find the connecting shell grids.

The CSEAM element provides the following key features:

- A seam line is considered continuous between two CSEAM elements that have a common face based on either common GS/GE points or XYZ coordinates. Note that the SMLN label on a CSEAM element does not determine the definition of a seam line. It is only intended for ease of seam line visualization.



- This element can connect up to 64 shell grids, which allows the connection of higher order shell elements.
- Besides selecting the connected surface patches by property IDs, the user may define the connection by specifying shell element IDs directly.
- Each of the eight auxiliary points must have a projection onto the connected shell element. This requirement prevents the generation of ill conditioned stiffness matrix in down stream processing.
- The user can model tailored blank tapering by specifying different property IDs at the start and end points.
- This element type supports the MAT9 anisotropic material properties.

The format for the Bulk Data entry [CSEAM](#) in the *MSC Nastran Quick Reference Guide* is as follows:

CSEAM

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE			

The format for the Bulk Data entry [PSEAM](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PSEAM

1	2	3	4	5	6	7	8	9	10
PSEAM	PID	MID	TYPE	W	T				

Example:

As an example of using the CSEAM element, two 2in plates which are separated by .1in and one of the plates edges is attached to a wall will have a seam created on the middle 1in of two opposite edges. Then the unattached plate is forced away from the wall. (See [MSC_DOC_DIR/doc/linstat/smln_101.dat](#)) The resulting deformed plot shows the plates held together and high stresses in the unattached plate. See [Figure 4-63](#). Output results from the CSEAM are formatted the same as the CHEXA.



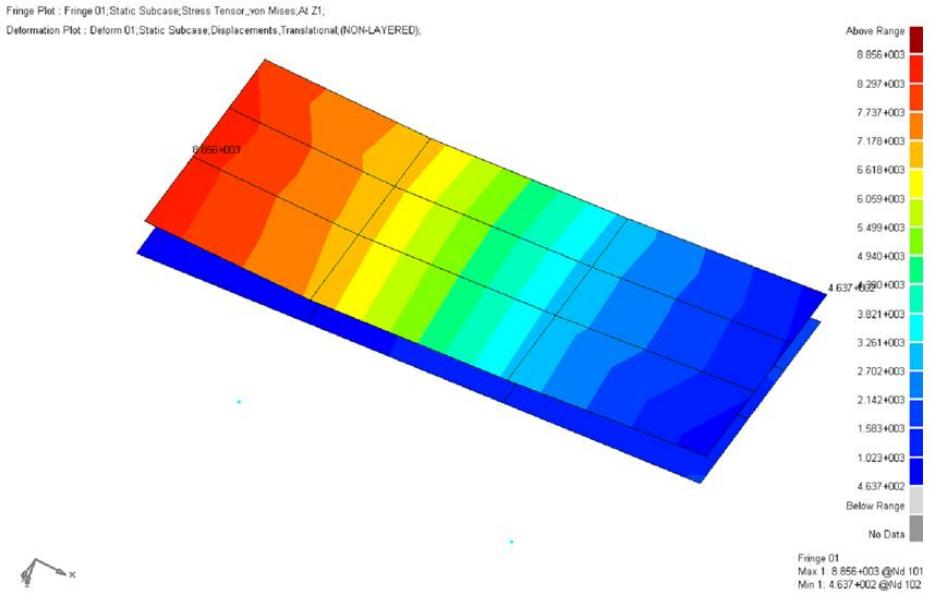


Figure 4-63 Displacement of fastened plates

CINTC Element

Line interface element is used to connect dissimilar meshes along the edges of finite element mesh subdomains. These subdomains have boundaries usually associated with either two-dimensional shell elements or one dimensional beam elements. A set of MPC (Multipoint Constraint) equations are internally generated with the interface boundary grids to enforce the compatibility of displacements and rotations across the interface.

There are perhaps three situations in which finite element analysts want to use the interface element technique to connect two or more independently meshed regions. They are:

- Different groups of engineers work almost independently in building finite element models.
- Global-local modeling is used in the analysis, where a fine mesh is required in a particular area due to high stress gradients while the rest of area only needs a coarse mesh, in consideration of the balance of computing resources and the solution accuracy.
- Some regions of a structural model need constant changes or modifications, while the rest of structure needs little or no change.

An interface element is equivalent to a glued or tied contact that connects the dissimilar meshes.

CINTC is the element connection entry which specifies the connectivity of line interface boundaries, represented by the GMBNDC Bulk Data entry.

The format for the Bulk Data entry **CINTC** in the *MSC Nastran Quick Reference Guide* is as follows:

CINTC



1	2	3	4	5	6	7	8	9	10
CINTC	EID	TYPE							
LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn))									

Bulk Data entry **GMBNDC** defines a geometric boundary consisting of edges along a curve interface.

Example

As shown in [Figure 4-64](#), two independently meshed plates are connected by a line interface element. The plate structure is constrained at one edge and subject to a pressure load. The interface element is defined by Bulk Data entries, CINTC and GMBNDC, as shown in [Listing 4-12](#). The solution results of displacement and von Mises stress distributions are presented in [Figure 4-65](#) and [Figure 4-66](#), respectively. See [MSC_DOC_DIR/doc/linstat/itfpltq4.dat](#).

Listing 4-12 Interface Element Definition

```
$-----2-----3-----4-----5-----6-----7-----8-----9-----0-----
CINTC 1001
      LIST=( 101, 102 )
GMBNDC 101   67   107
        GRID  72   77     82     87     92     97    102
GMBNDC 102    6   66
        GRID  12   18     24     30     36     42    48
      .
      54   60
```

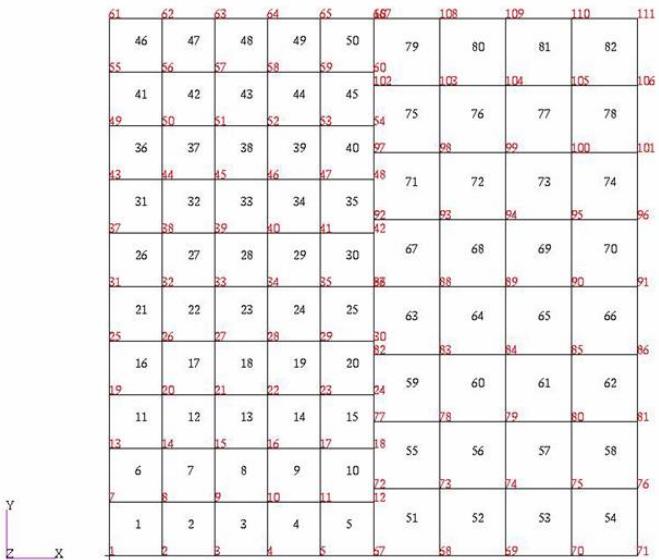


Figure 4-64 Two Plates Connected by Line Interface Element



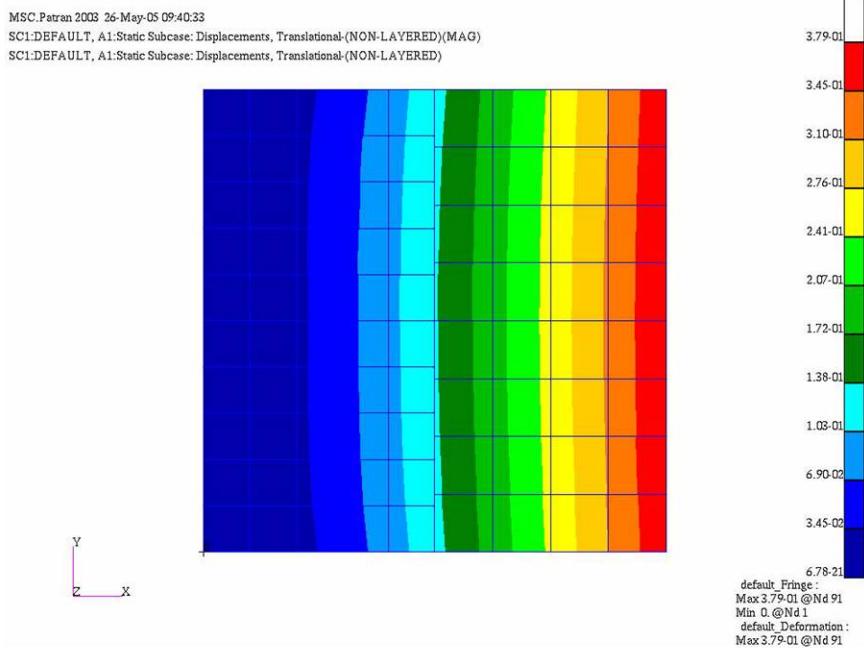


Figure 4-65 Displacement



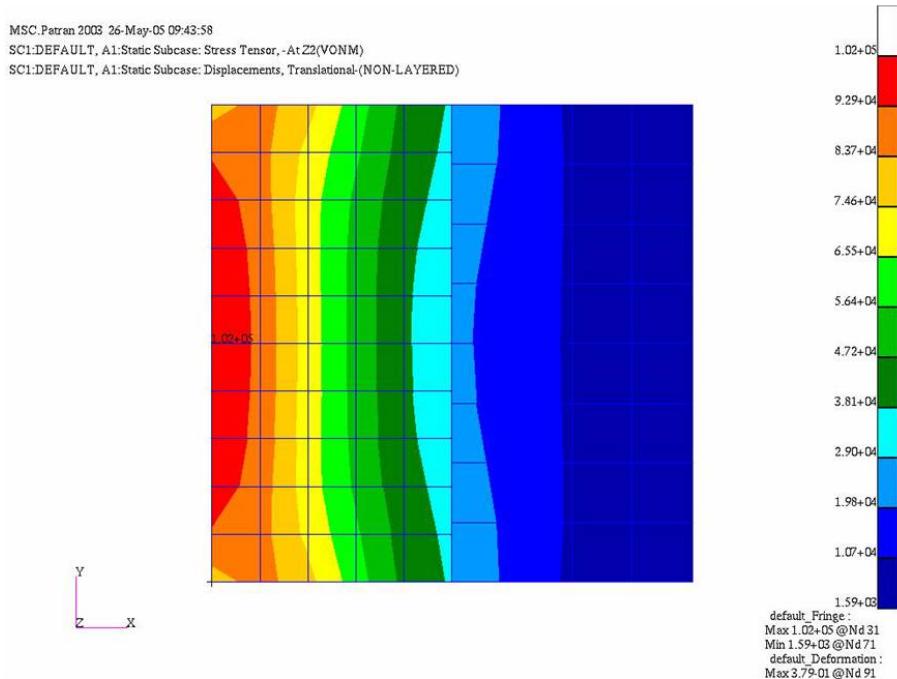


Figure 4-66 von Mises Stress







5

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 or PCOMPG)
- Equivalent Composite Honeycomb Section



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/PCOMPG entry. The input on the PCOMP/PCOMPG entry consists of the layer-by-layer material definition. MSC Nastran computes the equivalent PSHELL entries for you. An additional benefit of the PCOMP/PCOMPG 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/PCOMPG entry and the MAT9 entry for composite analysis.

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 v. 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:

$$G = \frac{E}{2(1+v)} \quad (5-1)$$



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 Bulk Data entry [MATBV](#) in the *MSC Nastran Quick Reference Guide* is as follows:

MAT1

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 Dynamic Analysis User's Guide*.

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 [Equation \(5-2\)](#) and [Equation \(5-3\)](#). 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 A_1 , A_2 , and A_3 . 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 4-42](#). The in-plane stress-strain relationship is described by [Equation \(5-2\)](#). [Equation \(5-3\)](#) defines the transverse shear stress-transverse shear strain relationship.



$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{Bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \end{Bmatrix} \quad (5-2)$$

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{Bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{Bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \quad (5-3)$$

The format of the Bulk Data entry [MAT2](#) in the *MSC Nastran Quick Reference Guide* is as follows:

MAT2

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/PCOMPG entries, MAT2 entries are generated automatically. The PCOMP/PCOMPG entry is discussed in [The Composite Element \(PCOMP or PCOMPG\)](#).

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 CTRIA6 element. The axisymmetric solid orthotropic material is defined by [Equation \(5-4\)](#).



$$\begin{Bmatrix} \epsilon_x \\ \epsilon_\theta \\ \epsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & \frac{-v_{\theta x}}{E_\theta} & \frac{-v_{zx}}{E_z} & 0 \\ \frac{-v_{x\theta}}{E_x} & \frac{1}{E_\theta} & \frac{-v_{z\theta}}{E_z} & 0 \\ \frac{-v_{xz}}{E_x} & \frac{-v_{\theta z}}{E_\theta} & \frac{1}{E_z} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - T_{REF}) \begin{Bmatrix} A_x \\ A_\theta \\ A_z \\ 0 \end{Bmatrix} \quad (5-4)$$

To preserve symmetry, the following relationships must hold:

$$\frac{v_{x\theta}}{E_x} = \frac{v_{\theta x}}{E_\theta}; \frac{v_{xz}}{E_x} = \frac{v_{zx}}{E_z}; \frac{v_{\theta z}}{E_\theta} = \frac{v_{z\theta}}{E_z} \quad (5-5)$$

The format of the Bulk Data entry **MAT3** in the *MSC Nastran Quick Reference Guide* is as follows:

MAT3

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
NUXTH, NUTHZ, NUZX	Poisson's ratios
RHO	Mass density
GZX	Shear modulus
AX, ATH, AZ	Thermal expansion coefficients
TREF	Reference temperature
GE	Structural element damping coefficient. See Remarks 9. and 11. (Real)

Two-Dimensional Orthotropic Material (MAT8)

The MAT8 entry is used to define a two-dimensional orthotropic stress-strain relationship as shown in [Equation \(5-6\)](#) and [Equation \(5-7\)](#). The MAT8 entry can only be used with the plate and shell elements. [Equation](#)



(5-6) defines the in-plane stress-strain relationship. The transverse shear stress-transverse shear strain relationship is defined by Equation (5-7).

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & \frac{-v_{12}}{E_1} & 0 \\ \frac{-v_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} + (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ 0 \end{Bmatrix} \quad (5-6)$$

$$\begin{Bmatrix} \tau_{1z} \\ \tau_{2z} \end{Bmatrix} = \begin{bmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{bmatrix} \begin{Bmatrix} \gamma_{1z} \\ \gamma_{2z} \end{Bmatrix} \quad (5-7)$$

The format of the Bulk Data entry MAT8 in the *MSC Nastran Quick Reference Guide* is as follows:

MAT8

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 ($\varepsilon_2/\varepsilon_1$ 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.



Field	Contents
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.

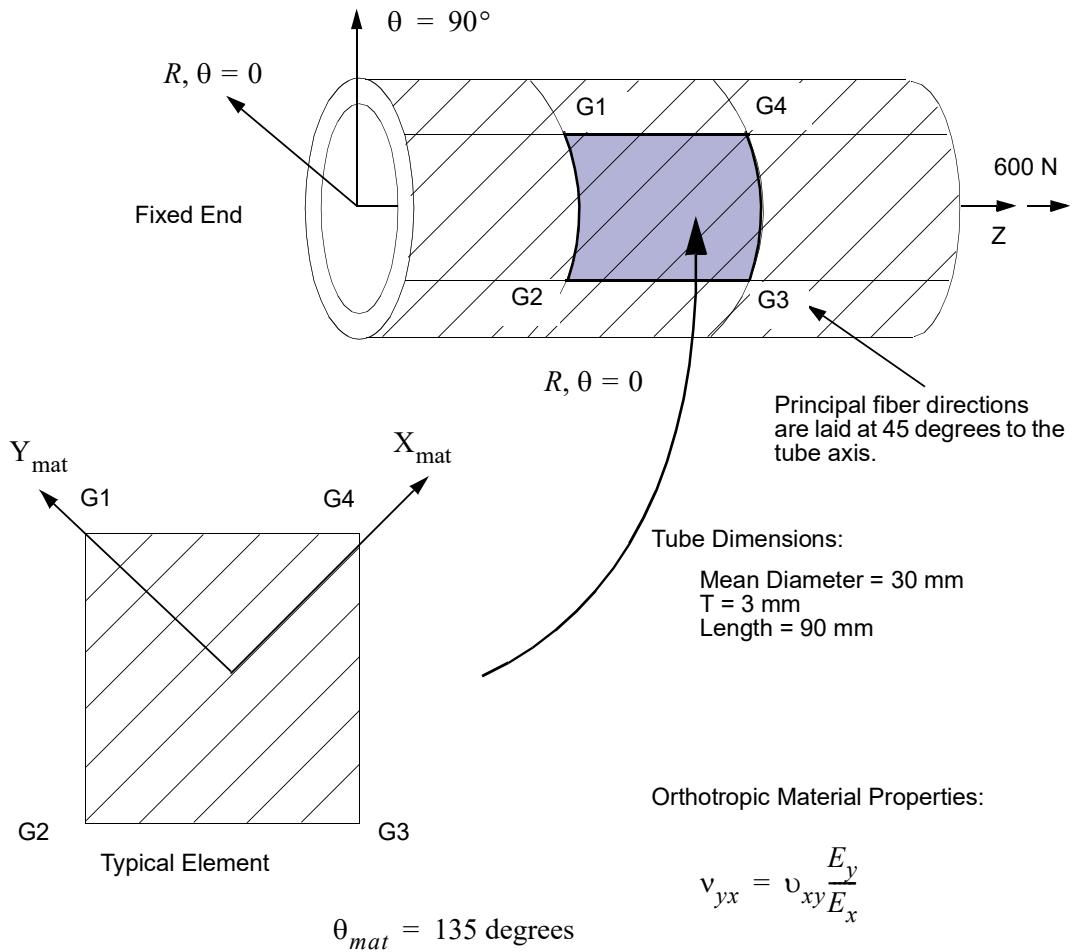


Figure 5-1 Composite Tube with Orthotropic Properties



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  

$  

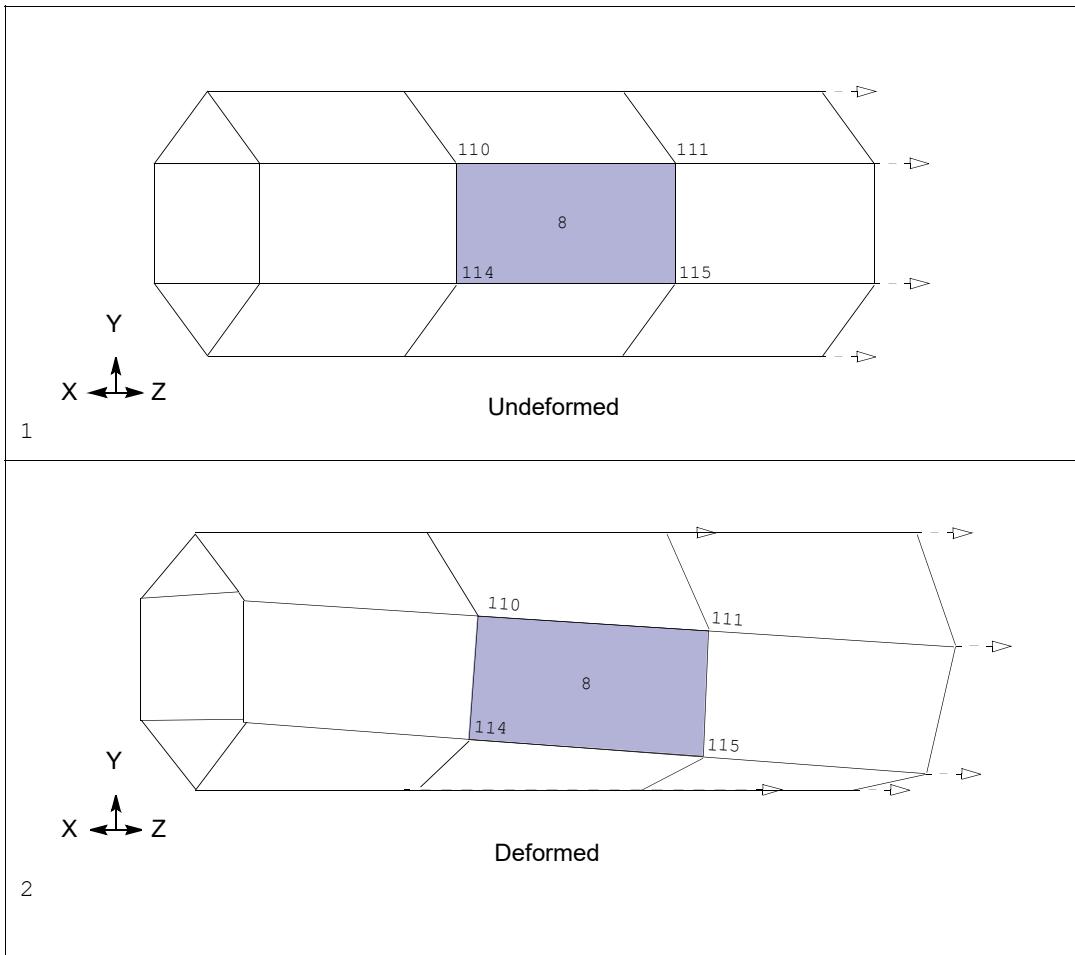
MAT8    1      250000. 100000. .4      80000.  80000.  80000.  

$
```

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

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 [Equation \(5-8\)](#).



$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} = [G] \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} - (T - T_{REF}) \begin{pmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{pmatrix} \quad (5-8)$$

where:

$$[G] = \begin{bmatrix} G11 & & & & & \\ G12 & G22 & & & & \\ G13 & G23 & G33 & & & \\ G14 & G24 & G34 & G44 & & \\ G15 & G25 & G35 & G45 & G55 & \\ G16 & G26 & G36 & G46 & G56 & G66 \end{bmatrix} \quad \text{symmetric}$$

The format of the Bulk Data entry [MAT9](#) in the *MSC Nastran Quick Reference Guide* is as follows:

MAT9

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.
GE	Structural element damping coefficient. See Remarks 6. and 8. (Real)



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 [Equation \(5-9\)](#):

$$\begin{aligned}
 G_{11} &= \frac{1 - \nu_{yz}\nu_{zy}}{E_y E_z \Delta} & G_{12} &= \frac{\nu_{yx} + \nu_{zx}\nu_{yz}}{E_y E_z \Delta} & G_{13} &= \frac{\nu_{zx} + \nu_{yx}\nu_{zy}}{E_y E_z \Delta} \\
 G_{22} &= \frac{1 - \nu_{xz}\nu_{zx}}{E_x E_z \Delta} & G_{23} &= \frac{\nu_{zy} + \nu_{xy}\nu_{zx}}{E_x E_z \Delta} \\
 G_{33} &= \frac{1 - \nu_{xy}\nu_{yx}}{E_x E_y \Delta}
 \end{aligned} \tag{5-9}$$

where:

ν_{ij}	=	Poisson's ratios where $\nu_{ij}/E_i = \nu_{ji}/E_j$
E_x, E_y, E_z	=	Young's modulus in the x-, y- and z-directions
G_{xy}, G_{yz}, G_{zx}	=	shear moduli
Δ	=	$\frac{1 - \nu_{xy}\nu_{yx} - \nu_{yz}\nu_{zy} - \nu_{zx}\nu_{xz} - 2\nu_{yx}\nu_{zy}\nu_{xz}}{E_x E_y E_z}$

$$|\nu_{yx}| < \sqrt{\frac{E_y}{E_x}} \quad |\nu_{zy}| < \sqrt{\frac{E_z}{E_y}} \quad |\nu_{xz}| < \sqrt{\frac{E_x}{E_z}}$$

$$|\nu_{xy}| < \sqrt{\frac{E_x}{E_y}} \quad |\nu_{yz}| < \sqrt{\frac{E_y}{E_z}} \quad |\nu_{zx}| < \sqrt{\frac{E_z}{E_x}}$$

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. (See [MSC_DOC_DIR/doc/linstat/anis1.dat](#)) 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.



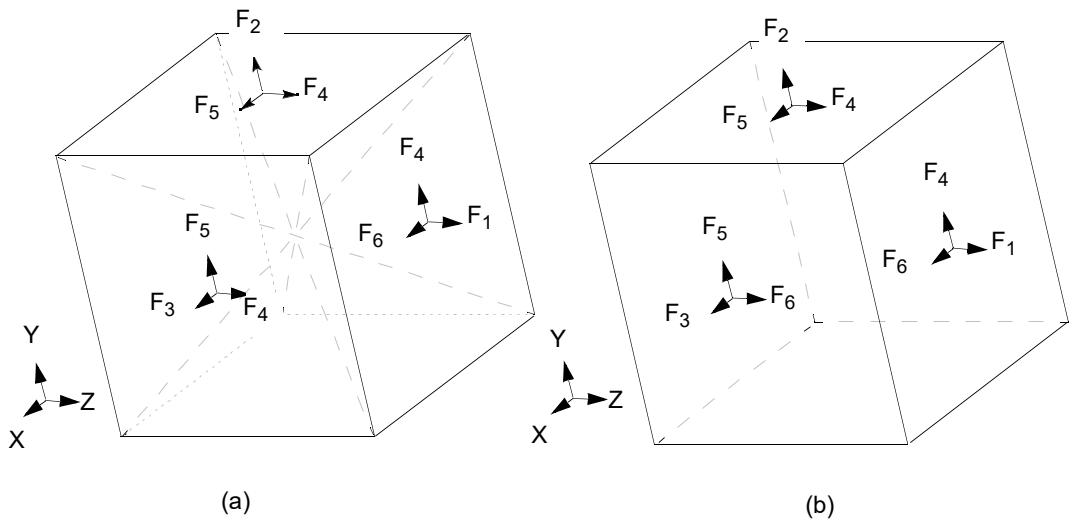


Figure 5-3 Pseudo-Anisotropic Model

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					
		F_X	F_Y	F_Z	F_{XY}	F_{YZ}	F_{ZX}
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 [Equation \(5-8\)](#). 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 [Equation \(5-10\)](#).



$$[G] = \begin{bmatrix} 10.67 & 6.91 & 6.89 & 1.25 & -0.09 & 0.86 \\ 6.91 & 19.03 & 6.90 & 1.27 & -0.12 & 0.83 \\ 6.89 & 6.90 & 14.28 & 1.23 & -0.15 & 0.82 \\ 1.25 & 1.27 & 1.23 & 5.27 & 0.21 & -0.11 \\ -0.09 & -0.12 & -0.15 & 0.21 & 4.63 & 0.52 \\ 0.86 & 0.83 & 0.82 & -0.11 & 0.52 & 5.72 \end{bmatrix} \cdot 10^6 \quad (5-10)$$

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 $1 \times 1 \times 1$ inch cube, but this time the analysis will not include the CBAR entries. 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](#). (See MSC_DOC_DIR/doc/linstat/anis2.dat)

Listing 5-2 Anisotropic Verification Input File

```
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
```

In the 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](#).



ELEMENT-ID	GRID-ID	STRAINS IN HEXAHEDRON SOLID			ELEMENTS (HEXA)			VON MISES
		CORNER NORMAL	CENTER AND CORNER POINT SHEAR	STRAINS PRINCIPAL	DIR. -A-	COSINES -B-	MEAN -C-	
13								SUBCASE 1
CENTER	X	1.549806E-07	XY -1.567790E-08	A 1.554435E-07	LX 1.00 0.02-0.04	-2.147629E-08	1.345509E-07	
	Y	-3.437740E-08	YZ 2.221636E-09	B -5.634964E-08	LY-0.04-0.04-1.00			
	Z	-5.617430E-08	ZX -1.076345E-08	C -3.466499E-08	LZ-0.03 1.00-0.04			
								SUBCASE 2
CENTER	X	-3.437740E-08	XY -5.186951E-09	A 7.192971E-08	LX-0.02 1.00 0.09	-6.659116E-09	6.601998E-08	
	Y	7.186195E-08	YZ 1.194498E-09	B -3.456629E-08	LY 1.00 0.02 0.01			
	Z	-1.750721E-08	ZX -2.957457E-09	C -1.738608E-08	LZ 0.01 0.09-1.00			
								SUBCASE 3
CENTER	X	-5.617430E-08	XY -7.530764E-09	A 1.065883E-07	LX 0.01 1.00 0.10	-1.095211E-08	9.830458E-08	
	Y	-1.750721E-08	YZ 2.773456E-09	B -5.656718E-08	LY 0.01 0.10-1.00			
	Z	1.065378E-07	ZX -4.683692E-09	C -1.716476E-08	LZ 1.00 0.01 0.01			
								SUBCASE 4
CENTER	X	-1.567790E-08	XY 1.970898E-07	A 8.825833E-08	LX 0.69 0.72-0.05	9.465206E-09	1.142492E-07	
	Y	-5.186951E-09	YZ -1.062715E-08	B -1.095836E-07	LY 0.73-0.69 0.04			
	Z	-7.530764E-09	ZX 8.945698E-09	C -7.070323E-09	LZ-0.01-0.07-1.00			
								SUBCASE 5
CENTER	X	2.221636E-09	XY -1.062715E-08	A 1.126186E-07	LX-0.10-0.03-0.99	-2.063197E-09	1.271659E-07	
	Y	1.194498E-09	YZ 2.189892E-07	B -1.076343E-07	LY 0.70 0.71-0.09			
	Z	2.773456E-09	ZX -2.101742E-08	C 1.205294E-09	LZ 0.71-0.71-0.05			
								SUBCASE 6
CENTER	X	-1.076345E-08	XY 8.945698E-09	A 8.237462E-08	LX 0.69 0.71-0.12	6.134867E-09	1.046499E-07	
	Y	-2.957457E-09	YZ -2.101742E-08	B -9.874503E-08	LY-0.05-0.11-0.99			
	Z	-4.683692E-09	ZX 1.796267E-07	C -2.034187E-09	LZ 0.72-0.69 0.04			

Figure 5-4

Verification Model Strain Output

Note: Only the center strains are shown here to reduce space. The complete output also gives the corner strains.

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.



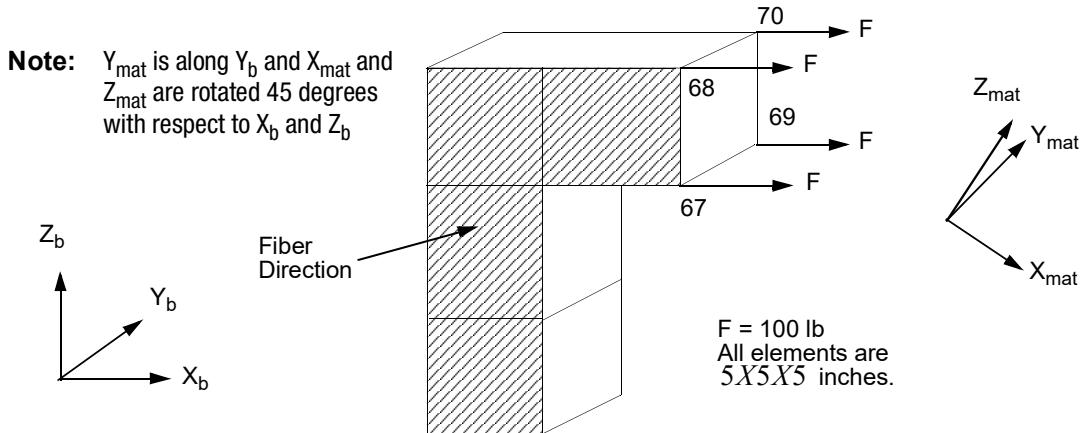


Figure 5-5 Anisotropic Solids Model

The input file is shown in [Listing 5-3](#). (See `MSC_DOC_DIR/doc/linstat/anis3.dat`) Note the local coordinate system used to define the material orientation. The displacement results are shown in [Figure 5-6](#).

Listing 5-3 Part of the Structure with Anisotropic Material Input File

```
$  
$ MATERIAL COORDINATE SYSTEM  
$  
CORD2R 1 0 0.0 0.0 0.0 .70710680.0 .7071068+  
.70710680.0 -.707107  
$  
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  
$
```

POINT	ID.	TYPE	DISPLACEMENT			VECTOR		
			T1	T2	T3	R1	R2	R3
51	G	0.0	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	0.0
53	G	1.232738E-04	3.251065E-07	8.268995E-05	0.0	0.0	0.0	0.0
54	G	1.204173E-04	1.783952E-06	-8.797066E-05	0.0	0.0	0.0	0.0
55	G	0.0	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	0.0
57	G	1.002035E-04	2.286116E-05	7.448530E-05	0.0	0.0	0.0	0.0
58	G	9.797833E-05	-2.485534E-05	-7.772400E-05	0.0	0.0	0.0	0.0
59	G	3.605538E-04	1.530884E-05	1.199663E-04	0.0	0.0	0.0	0.0
60	G	3.610890E-04	-1.436989E-05	-1.294236E-04	0.0	0.0	0.0	0.0
61	G	3.284150E-04	1.772471E-05	1.189002E-04	0.0	0.0	0.0	0.0
62	G	3.284143E-04	-1.659684E-05	-1.265683E-04	0.0	0.0	0.0	0.0
63	G	6.271876E-04	1.717129E-05	1.256454E-04	0.0	0.0	0.0	0.0
64	G	6.300415E-04	-1.526032E-05	-1.351714E-04	0.0	0.0	0.0	0.0
65	G	5.944516E-04	1.910164E-05	1.262980E-04	0.0	0.0	0.0	0.0
66	G	5.975869E-04	-1.427919E-05	-1.336644E-04	0.0	0.0	0.0	0.0
67	G	3.671078E-04	-5.154453E-05	-4.031157E-04	0.0	0.0	0.0	0.0
68	G	6.358424E-04	-4.903843E-05	-4.012304E-04	0.0	0.0	0.0	0.0
69	G	3.339844E-04	-4.717636E-05	-4.023418E-04	0.0	0.0	0.0	0.0
70	G	6.039479E-04	-4.626662E-05	-3.997554E-04	0.0	0.0	0.0	0.0

Figure 5-6 Output for Structure with Anisotropic Material



The PSHELL Entry

The PSHELL entry was introduced in [MSC Nastran Elements](#) 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 Bulk Data entry [PSHELL](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PSHELL

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.
MID _i	Material identification numbers.
T	Default membrane thickness for T _i on the connection entry or average thickness if TFLAG = 1 (Field 13 of plate element), then $T_i = T_i \cdot T$.
$12I/T^3$	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/PCOMPG 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.

The Composite Element (PCOMP or PCOMPG)

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 Bulk Data entry [PCOMP](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PCOMP

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	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.
MID _i	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer.
T _i	Thicknesses of the various plies.



Field	Contents
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element.
SOUTi	Stress or strain output request.

An alternative to the PCOMP entry is the PCOMPG entry. The PCOMPG entry includes a global ply ID, so it is easier to track the output for the same ply across the model. The format for the Bulk Data entry [PCOMPG](#) in the *MSC Nastran Quick Reference Guide* is as follows:

PCOMPG

1	2	3	4	5	6	7	8	9	10
PCOMP G	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYID1	MID1	T1	THETA1	SOUT1				
	GPLYID2	MID2	T2	THETA2	SOUT2				

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark 10. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 7. “HILL” for the Hill theory. “HOFF” for the Hoffman theory. “TSAI” for the Tsai-Wu theory. “STRN” for the Maximum Strain theory.
TREF	Reference temperature. See Remark 4. (Real; Default = 0.0)
GE	Damping coefficient. See Remark 5. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 13. and 14. “Blank” All plies must be specified and all stiffness terms are developed. “MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed. “BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.



Field	Contents
	“SMEAR” All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero).
	“SMCORE” All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.
GPLYID _i	User-defined Global (External) Ply ID. See Remark 1. (Integer > 0)
MID _i	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 2. and 15. (Integer > 0 or blank, except MID1 must be specified.)
T _i	Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)
THETA _i	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUT _i	Stress or strain output request. See Remarks 6. and 7. (Character: “YES” or “NO”; Default = “NO”)

Remark:

1. The global ply identification number should be unique with respect to other plies in the entry. The plies are defined in stacking sequence starting with the bottom layer.

Additional properties can be entered on the **PSHLN1** entry for use in SOL 400 STATICS.

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 each lamina oriented in a particular direction as shown in [Figure 5-7](#).



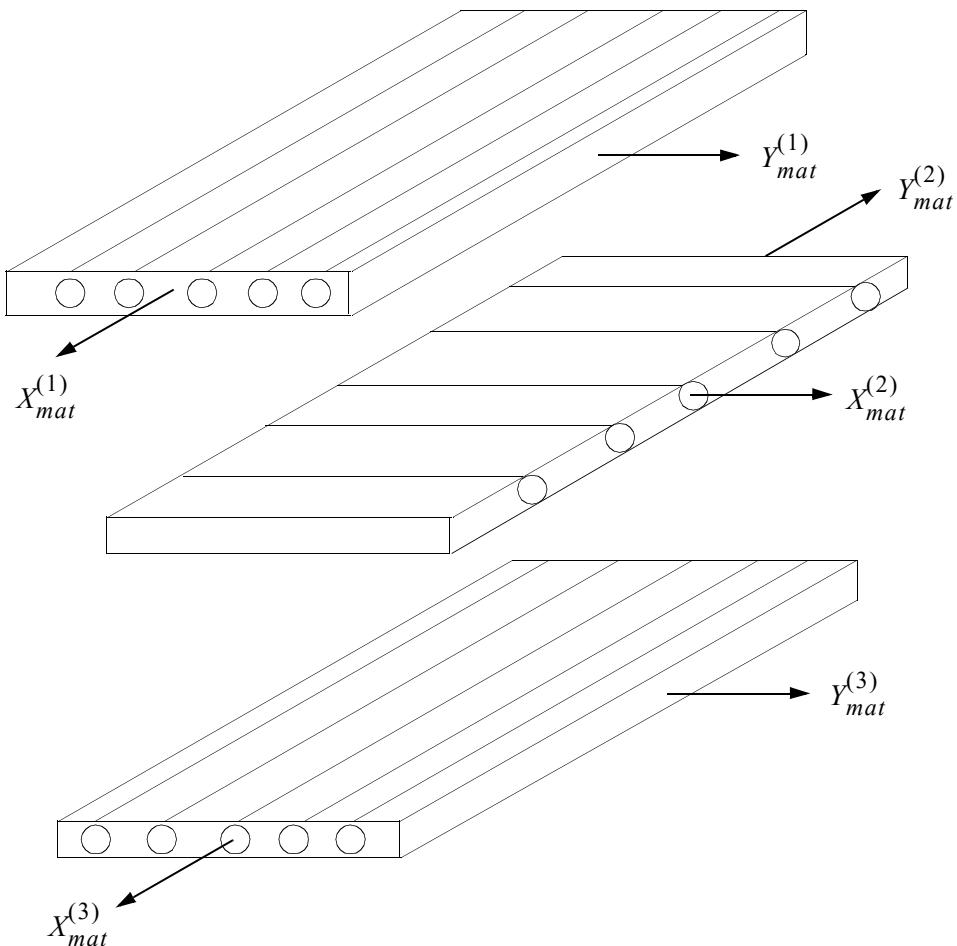


Figure 5-7 Laminae Arranged to Form a Laminate

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](#).



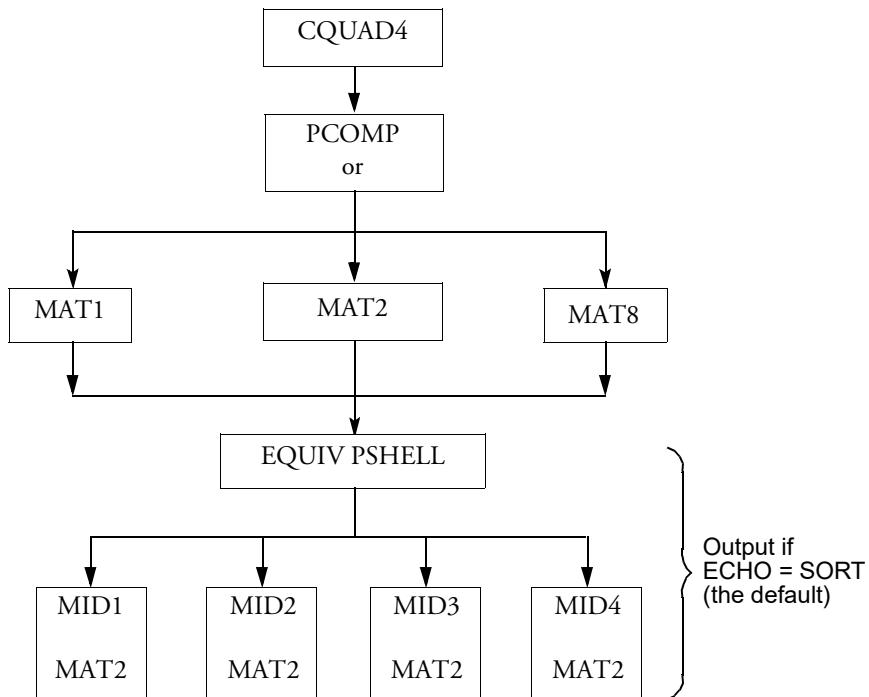


Figure 5-8 Equivalent PSHELL and MAT2 Entries Are Generated

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/PCOMPG entries.

As shown in Figure 5-8, each PCOMP/PCOMPG (together with the material entries for each lamina) is processed to form an equivalent PSHELL and four MAT2 entries. To print the equivalent PSHELL and MAT2 entries in the output file, use NASTRAN [PRTPCOMP \(361\)=1](#) provided that ECHO=NONE is not set. If you wish to use them for future runs, they may be punched to an ASCII file. If you are using the equivalent properties in a future analysis instead of using the PCOMP/PCOMPG entries 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 PCOMPG 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 PCOMPG entry. The top GPLYID will be 103, center 102 and the bottom 101.

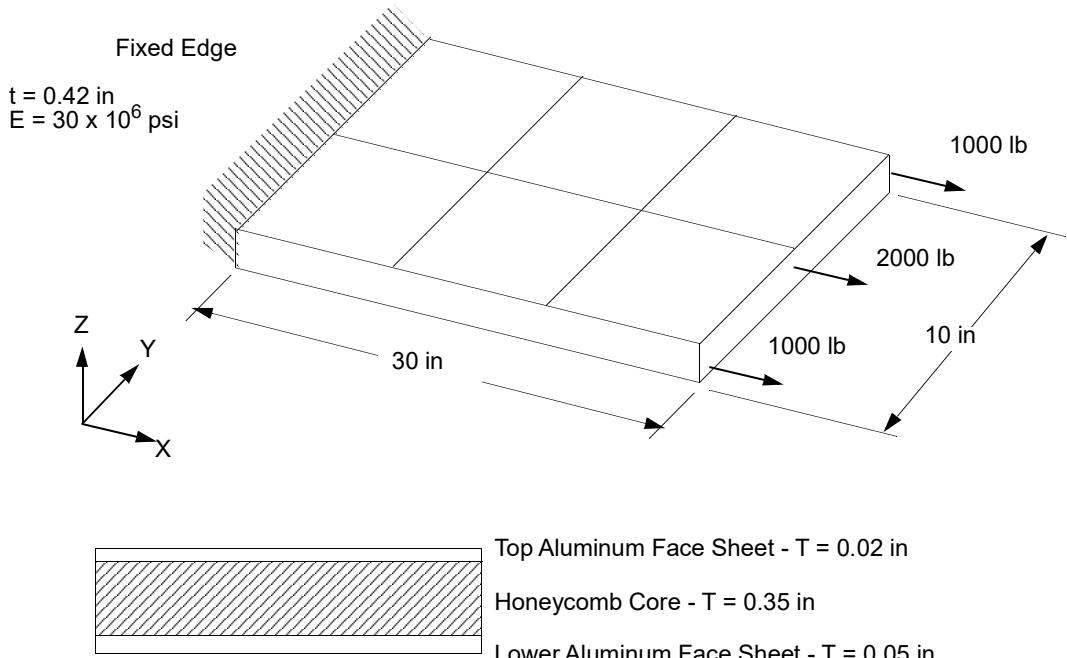


Figure 5-9 Honeycomb Cantilever Plate

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](#) (See [MSC_DOC_DIR/doc/linstat/pcomp1.dat](#)). The load on the plate is a 4000-pound uniform load acting in the plane of the plate. Using the PCOMPG entry and the [GPRSORT \(Case\)](#) Case Control command, sorted output of composite ply results can be printed. The output requests include the element force, laminae stress, laminae strain, and failure index table for both element 4 and element 4's plys by GPLYID.

A selected portion of the output is shown in [Figure 5-10](#). If PCOMP's had been used the ply IDs would have printed as "1,2,3".

Listing 5-4 Honeycomb Plate

```
NASTRAN PRTPCOMP=1

SET 1 = 4
SET 2 = 8
DISP = 2
STRESS = 1
GPRSORT = 1

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
$
PCOMPG 100          100.    STRN
          101      120     0.05   0.      YES
          102      130     0.35   0.0    YES
          103      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 (IFP6CD)
THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING PSHELL AND MAT2 CARDS.
REFER TO REMARK 12 OF THE MAT2 DESCRIPTION IN THE MSC NASTRAN QUICK REFERENCE GUIDE.

PSHELL      100   10000001  4.2000E-01    10000002  1.0000E+00    10000003  1.0000E+00  0.0000E+00
          -2.1000E-01  2.1000E-01  10000004
MAT2      10000001  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
MAT2      10000002  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
MAT2      10000003  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
MAT2      10000004  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

DISPLACEMENT VECTOR

POINT ID. TYPE      T1        T2        T3        R1        R2        R3
  8       G     1.004615E-02 -1.431147E-17 -3.156609E-01  1.103718E-16  2.090650E-02  0.0

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  2.748534E+01 -9.314405E+00  5.329071E-15 -3.737749E-03  4.785021E-03  4.475587E-16 -1.003042E-03

STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRESSES IN FIBER 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     101  2.08187E+03  2.85912E+02 -9.67098E+01  1.15645E-15 -2.59177E-03 -3.07  2.08707E+03  2.80719E+02  9.03174E+02
  4     102  3.49114E+02  4.79933E-03 -1.62792E-03  1.15638E-15 -2.59162E-03 -3.09  3.49991E-02  4.71158E-03  1.51438E-02
  4     103  4.79470E+03  6.59403E+02 -2.23917E+02  1.60252E-23 -3.59147E-11 -3.09  4.80679E+03  6.47314E+02  2.07974E+03

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  101   0.0570   -1           0.0000
                    102   0.0007   -1           0.0000
                    103   0.1313   -1           0.0000
                                         0.1313

STRESSES IN LAYERED COMPOSITE ELEMENTS
GLOBAL ELEMENT STRESSES IN FIBER AND MATRIX DIRECTIONS INTER-LAMINAR STRESSES PRINCIPAL STRESSES (ZERO SHEAR) MAX
PLY ID ID      NORMAL-1  NORMAL-2  SHEAR-12  SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MINOR  SHEAR
  101  4     2.08187E+03  2.85912E+02 -9.67098E+01  1.15645E-15 -2.59177E-03 -3.07  2.08707E+03  2.80719E+02  9.03174E+02
  102  4     3.49114E+02  4.79933E-03 -1.62792E-03  1.15638E-15 -2.59162E-03 -3.09  3.49991E-02  4.71158E-03  1.51438E-02
  103  4     4.79470E+03  6.59403E+02 -2.23917E+02  1.60252E-23 -3.59147E-11 -3.09  4.80679E+03  6.47314E+02  2.07974E+03

FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS REFERENCED BY GLOBAL PLY
GLOBAL FAILURE ELEMENT FP=FAILURE INDEX FOR PLY FB=FAILURE INDEX FOR BONDING MAX OF FP,FB FOR ALL ELEMENTS FLAG
PLY ID THEORY ID      (DIRECT STRESSES/STRAINS) (INTER-LAMINAR STRESSES) REFERENCED BY GLOBAL PLY
  101  STRAIN  4      0.0570   -1           0.0000
                                         0.0570
  102  STRAIN  4      0.0007   -1           0.0000
                                         0.0007
  103  STRAIN  4      0.1313   -1           0.0000
                                         0.1313

STRAINS IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRAINS IN FIBER 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     101  1.99610E-04 -3.38650E-05 -2.51446E-05  3.00677E-22 -6.73859E-10 -3.07  2.00285E-04 -3.45401E-05  2.34825E-04
  4     102  3.34716E-04 -5.67407E-05 -4.23258E-05  3.00660E-17 -6.73821E-05 -3.09  3.35856E-04 -5.78815E-05  3.93738E-04
  4     103  4.59688E-04 -7.79008E-05 -5.82185E-05  4.16654E-30 -9.33781E-18 -3.09  4.61260E-04 -7.94724E-05  5.40732E-04

STRAINS IN LAYERED COMPOSITE ELEMENTS
GLOBAL ELEMENT STRAINS IN FIBER AND MATRIX DIRECTIONS INTER-LAMINAR STRAINS PRINCIPAL STRAINS (ZERO SHEAR) MAX
PLY ID ID      NORMAL-1  NORMAL-2  SHEAR-12  SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MINOR  SHEAR
  101  4     1.99610E-04 -3.38650E-05 -2.51446E-05  3.00677E-22 -6.73859E-10 -3.07  2.00285E-04 -3.45401E-05  2.34825E-04
  102  4     3.34716E-04 -5.67407E-05 -4.23258E-05  3.00660E-17 -6.73821E-05 -3.09  3.35856E-04 -5.78815E-05  3.93738E-04
  103  4     4.59688E-04 -7.79008E-05 -5.82185E-05  4.16654E-30 -9.33781E-18 -3.09  4.61260E-04 -7.94724E-05  5.40732E-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.

Another example using the PCOMP entry, consider the model shown in [Figure 5-11](#).

[Figure 5-12](#) 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.



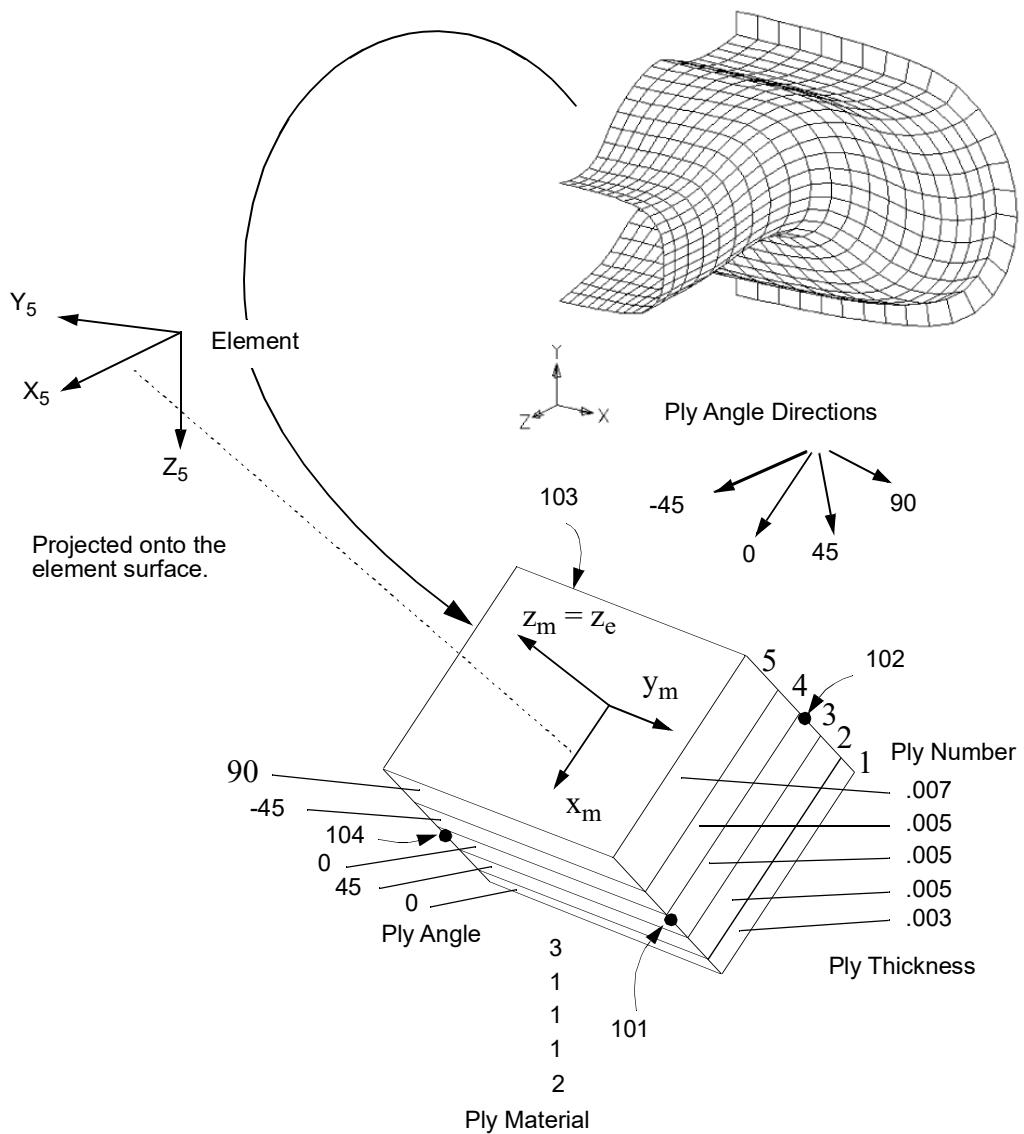


Figure 5-11 Typical Composite Layup

```
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, 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 5-12

In the previous example, 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 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.

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 or PCOMPG/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 5-13](#) 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 5-14](#).

Aluminum Facesheets	Honeycomb Core
E1=E2=1.E7	E1=E2=1000.
G12=G13=G23=3.85E6	G12=100., G13=1.E5, G23=5.E4

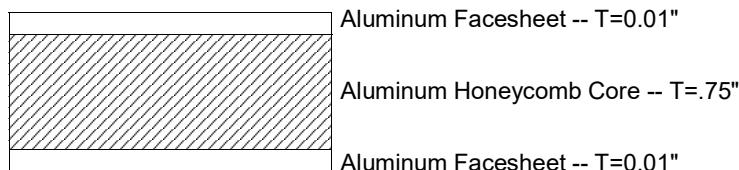


Figure 5-13 Honeycomb Cantilever Plate Represented by PSHELL/MAT1



$$I = 2y^2 A = 2(.38)^2 (.01) = 2.888 \cdot 10^{-3}$$

$$T = 0.02$$

$$\frac{12I}{T^3} = \frac{12(2.88 \cdot 10^{-3})}{(0.02)^3} = 4332$$

$$T_s \approx .75 + .02 = .77$$

$$\frac{T_s}{T} = \frac{.77}{.02} = 38.5$$

$$G_{core} \approx \frac{1 \cdot 10^5 + 5 \cdot 10^4}{2} = 7.5 \cdot 10^4$$

Figure 5-14 Calculation of the Equivalent Properties

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.

As a comparison of PCOMP/MAT8 (see MSC_DOC_DIR/doc/linstat pcomp2.dat) and PSHELL/MAT1 (see MSC_DOC_DIR/doc/linstat pshell2.dat) methods, the abridged output files in [Figure 5-15](#) and [Figure 5-16](#) show the displacements calculated by using the PSHELL/MAT1 combination are comparable with the displacements obtained using the PCOMP/MAT8 combination.



AXIAL LOAD							SUBCASE 1		
POINT ID.	TYPE	DISPLACEMENT			VECTOR			R3	
		T1	T2	T3	R1	R2			
1	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
2	G	9.962895E-03	3.228006E-03	0.0	0.0	0.0	0.0	0.0	
3	G	1.950692E-02	3.060445E-03	0.0	0.0	0.0	0.0	0.0	
4	G	2.949229E-02	2.925501E-03	0.0	0.0	0.0	0.0	0.0	
5	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
6	G	9.140318E-03	-3.035766E-18	0.0	0.0	0.0	0.0	0.0	
7	G	1.961452E-02	-1.301043E-17	0.0	0.0	0.0	0.0	0.0	
8	G	2.955693E-02	-2.602085E-17	0.0	0.0	0.0	0.0	0.0	
9	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
10	G	9.962895E-03	-3.228006E-03	0.0	0.0	0.0	0.0	0.0	
11	G	1.950692E-02	-3.060445E-03	0.0	0.0	0.0	0.0	0.0	
12	G	2.949229E-02	-2.925501E-03	0.0	0.0	0.0	0.0	0.0	

VERTICAL SHEAR LOAD							SUBCASE 2		
POINT ID.	TYPE	DISPLACEMENT			VECTOR			R3	
		T1	T2	T3	R1	R2			
1	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
2	G	0.0	0.0	8.073865E-03	2.400982E-04	-1.522475E-03	0.0	0.0	
3	G	0.0	0.0	2.972733E-02	2.395836E-04	-2.691690E-03	0.0	0.0	
4	G	0.0	0.0	5.865972E-02	1.254657E-04	-2.983529E-03	0.0	0.0	
5	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
6	G	0.0	0.0	9.123428E-03	1.948176E-20	-1.696776E-03	0.0	0.0	
7	G	0.0	0.0	3.092414E-02	4.336809E-18	-2.559687E-03	0.0	0.0	
8	G	0.0	0.0	5.930084E-02	1.768605E-18	-3.006703E-03	0.0	0.0	
9	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
10	G	0.0	0.0	8.073865E-03	-2.400982E-04	-1.522475E-03	0.0	0.0	
11	G	0.0	0.0	2.972733E-02	-2.395836E-04	-2.691690E-03	0.0	0.0	
12	G	0.0	0.0	5.865972E-02	-1.254657E-04	-2.983529E-03	0.0	0.0	

VERTICAL TWIST LOAD							SUBCASE 3		
POINT ID.	TYPE	DISPLACEMENT			VECTOR			R3	
		T1	T2	T3	R1	R2			
1	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
2	G	0.0	0.0	-1.261490E-03	1.146153E-04	2.239877E-04	0.0	0.0	
3	G	0.0	0.0	-3.607047E-03	3.615262E-04	2.259625E-04	0.0	0.0	
4	G	0.0	0.0	-5.963150E-03	5.777780E-04	2.258924E-04	0.0	0.0	
5	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
6	G	0.0	0.0	1.619103E-18	1.428758E-04	-3.159433E-19	0.0	0.0	
7	G	0.0	0.0	6.965999E-18	3.529766E-04	-7.318365E-19	0.0	0.0	
8	G	0.0	0.0	1.478072E-17	5.823313E-04	-7.860466E-19	0.0	0.0	
9	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
10	G	0.0	0.0	1.261490E-03	1.146153E-04	-2.239877E-04	0.0	0.0	
11	G	0.0	0.0	3.607047E-03	3.615262E-04	-2.259625E-04	0.0	0.0	
12	G	0.0	0.0	5.963150E-03	5.777780E-04	-2.258924E-04	0.0	0.0	

Figure 5-15 Displacement Outputs for Honeycomb Sandwich Using PCOMP



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	0.0	0.0	0.0
2	G	1.000260E-02	3.251481E-03	0.0	0.0	0.0	0.0
3	G	1.958364E-02	3.083884E-03	0.0	0.0	0.0	0.0
4	G	2.960999E-02	2.947241E-03	0.0	0.0	0.0	0.0
5	G	0.0	0.0	0.0	0.0	0.0	0.0
6	G	9.172846E-03	-2.818926E-18	0.0	0.0	0.0	0.0
7	G	1.969241E-02	-7.806256E-18	0.0	0.0	0.0	0.0
8	G	2.967541E-02	-1.474515E-17	0.0	0.0	0.0	0.0
9	G	0.0	0.0	0.0	0.0	0.0	0.0
10	G	1.000260E-02	-3.251481E-03	0.0	0.0	0.0	0.0
11	G	1.958364E-02	-3.083884E-03	0.0	0.0	0.0	0.0
12	G	2.960999E-02	-2.947241E-03	0.0	0.0	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	0.0	0.0	0.0
2	G	0.0	0.0	8.076960E-03	2.402719E-04	-1.525178E-03	0.0
3	G	0.0	0.0	2.978118E-02	2.410396E-04	-2.695777E-03	0.0
4	G	0.0	0.0	5.876278E-02	1.257192E-04	-2.987175E-03	0.0
5	G	0.0	0.0	0.0	0.0	0.0	0.0
6	G	0.0	0.0	9.157884E-03	-1.699656E-17	-1.697873E-03	0.0
7	G	0.0	0.0	3.097864E-02	-3.220080E-17	-2.562250E-03	0.0
8	G	0.0	0.0	5.939800E-02	-4.227033E-17	-3.011073E-03	0.0
9	G	0.0	0.0	0.0	0.0	0.0	0.0
10	G	0.0	0.0	8.076960E-03	-2.402719E-04	-1.525178E-03	0.0
11	G	0.0	0.0	2.978118E-02	-2.410396E-04	-2.695777E-03	0.0
12	G	0.0	0.0	5.876278E-02	-1.257192E-04	-2.987175E-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	0.0	0.0	0.0
2	G	0.0	0.0	-1.283220E-03	1.158808E-04	2.232562E-04	0.0
3	G	0.0	0.0	-3.635008E-03	3.642552E-04	2.244724E-04	0.0
4	G	0.0	0.0	-5.990866E-03	5.833288E-04	2.238659E-04	0.0
5	G	0.0	0.0	0.0	0.0	0.0	0.0
6	G	0.0	0.0	-3.672608E-17	1.440272E-04	6.528930E-18	0.0
7	G	0.0	0.0	-1.145731E-16	3.556043E-04	8.429672E-18	0.0
8	G	0.0	0.0	-2.037080E-16	5.878134E-04	9.229271E-18	0.0
9	G	0.0	0.0	0.0	0.0	0.0	0.0
10	G	0.0	0.0	1.283220E-03	1.158808E-04	-2.232562E-04	0.0
11	G	0.0	0.0	3.635008E-03	3.642552E-04	-2.244724E-04	0.0
12	G	0.0	0.0	5.990866E-03	5.833288E-04	-2.238659E-04	0.0

Figure 5-16 Displacement Outputs for Honeycomb Sandwich Using PSHELL/MAT1



6 Constraints

- Introduction to Constraints
- Single-Point Constraints (SPC, SPC1)
- Automatic Application of Single-Point Constraints (AUTOSPC)
- Enforced Displacements at Grid Points (SPCD, SPC)
- Multipoint Constraints (MPC)
- Nonhomogeneous Multipoint Constraint (MPCY, MPCD)



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](#)). However, this level of understanding is not necessary for the majority of the applications.

Single-Point Constraints (SPC, SPC1)

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](#).
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, 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 \(AUTOSPC\)](#).

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 6-1](#), that permit unrestrained motion at point G in any direction perpendicular to the axis of the rods.



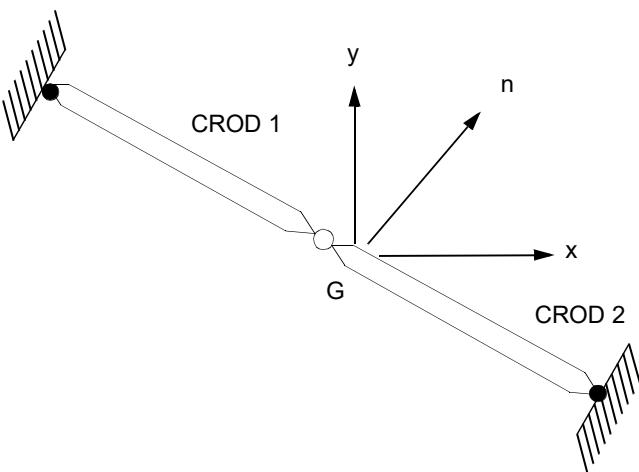


Figure 6-1 Pin-Connected Rods

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 \(SPCD, SPC\)](#).

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 Bulk Data entry [SPC](#) in the *MSC Nastran Quick Reference Guide* is as follows:

SPC

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 Bulk Data entry [SPC1](#) in the *MSC Nastran Quick Reference Guide* is as follows:



SPC1

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 AUTOSPC, which is described in [Automatic Application of Single-Point Constraints \(AUTOSPC\)](#).

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 Bulk Data or the Case Control command [AUTOSPC \(Case\)](#) in the *MSC Nastran Quick Reference Guide*. In Solution 101, the Grid Point Singularity Processor (GPSP) module is invoked to identify potential singularities. AUTOSPC defaults 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 3×3 matrices representing the three translations and three rotations of each grid point. For each 3×3 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 [Equation \(6-1\)](#):

$$R_i = \frac{K_i}{|K_{MAX}|} \quad \text{if } |K_{MAX}| > 0$$

$$i = 1, 2, 3$$

$$R_i = 0.0 \quad \text{if } |K_{MAX}| = 0 \tag{6-1}$$



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 $|R_{ij}| < EPZERO$ 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 α -set are not constrained (α -sets are discussed in [Matrix Operations](#)).

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 6-2](#).



Note: Local coordinate system 1
 is rotated -45° about the X_b axis.

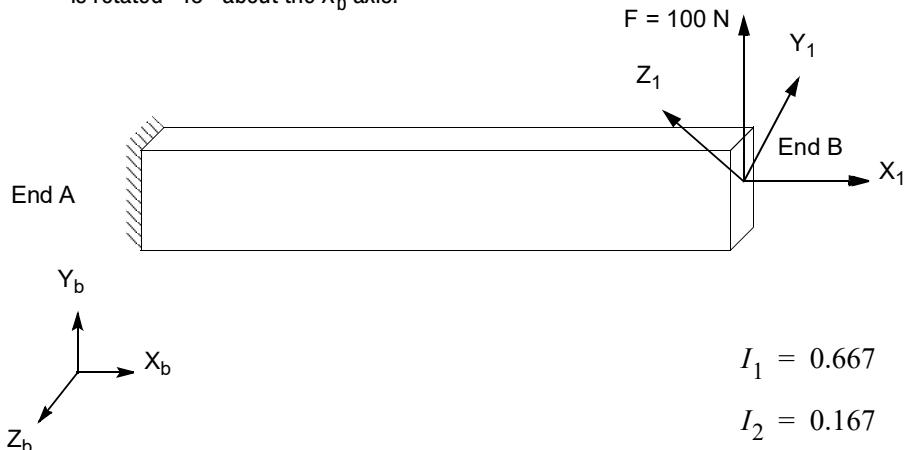


Figure 6-2 Single Element with Vertical Load

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 6-1](#). (See MSC_DOC_DIR/doc/linstat/bar3a.dat) 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 6-1 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 6-3](#) confirm this assumption.

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE UNION	NEW USET EXCLUSIVE UNION	SB	S	*	
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		
ELEMENT FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT PLANE 1	END-A	BEND-MOMENT PLANE 2	END-B	- SHEAR -	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 6-3](#) 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. (See MSC_DOC_DIR/doc/linstat/bar36.dat) The resulting grid point singularity table, element forces, and displacements are shown in [Figure 6-4](#).

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE UNION	NEW USET EXCLUSIVE UNION	SB	S	*	
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	.0		1.060130E-03
ELEMENT FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT PLANE 1	END-A	BEND-MOMENT PLANE 2	END-B	- SHEAR -	AXIAL FORCE	TORQUE		
1	1.000000E+03	.0	4.263256E-13	.0	1.000000E+02	.0	.0		.0

[Figure 6-4](#) 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_1 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.

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 Bulk Data entry **SPCD** in the *MSC Nastran Quick Reference Guide* is as follows:

SPCD

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.

The SPC/SPCD combination is more efficient than SPC only 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 selected in the Case Control Section with the LOAD command.

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.



Multipoint Constraints (MPC)

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

$$\sum_j R_j u_j = 0 \quad (6-2)$$

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 Constraint Elements](#).
- 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 Bulk Data entry [MPC](#) in the *MSC Nastran Quick Reference Guide* is as follows:

MPC

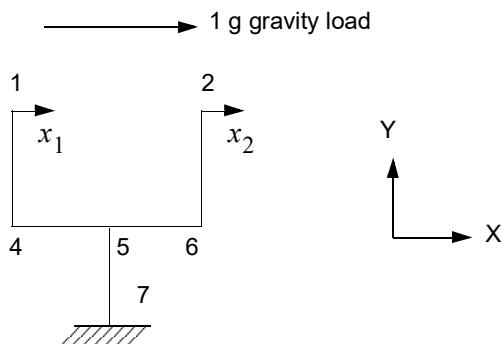
1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				



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 6-5](#). 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 6-5](#) 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 as x_2 given by [Equation \(6-3\)](#). The difference is represented by scalar point 3 defined on an SPOINT entry.

$$x_3 = x_2 - x_1 \quad (6-3)$$

The equation of constraint can be in any of the forms shown in [Equation \(6-4\)](#).

$$\begin{aligned} x_3 - x_2 + x_1 &= 0 & (a) \\ x_2 - x_1 - x_3 &= 0 & (b) \\ x_1 - x_2 + x_3 &= 0 & (c) \end{aligned} \quad (6-4)$$

[Listing 6-2](#) shows the implementation of [Equation \(6-4\)\(a\)](#) to obtain the relative displacement. (See MSC_DOC_DIR/doc/linstat/reldiff.dat) 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 [Equation \(6-4\)](#).

Listing 6-2 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
```

The MPCY bulk data entry may be used to specify a nonhomogenous constraint.

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](#).
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 [Equation \(6-4\)](#) 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 [Equation \(6-5\)](#).

$$R_1 u_1 + R_2 u_2 + R_3 u_3 = 0 \quad (6-5)$$

The work done by the corresponding forces of constraint is given by [Equation \(6-6\)](#).

$$W = 1/2(F_1 u_1 + F_2 u_2 + F_3 u_3) = 0 \quad (6-6)$$

In order for both [Equation \(6-5\)](#) and [Equation \(6-6\)](#) to be valid, it is necessary that the following equation also be valid.

$$\frac{F_1}{R_1} = \frac{F_2}{R_2} = \frac{F_3}{R_3} \quad (6-7)$$

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.,

$$u_5 = 1/4(u_1 + u_2 + u_3 + u_4) \quad (6-8)$$

From [Equation \(6-8\)](#), the forces of constraint are given by:

$$F_1 = F_2 = F_3 = F_4 = 1/4F_5 \quad (6-9)$$

It is clear, from [Equation \(6-9\)](#), 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).

Nonhomogeneous Multipoint Constraint (MPCY, MPCD)

With MPC's, if a nonhomogeneous multipoint constraint was desired, then the method of "SLACK" variable was required. That is, the nonhomogeneous right hand side of the MPC equation was written using a scalar or grid point, with an SPC or SPCD specifying the right hand side value. With an MPCY Bulk Data entry the right hand side value is entered directly into the MPC equation. The entry defines an equation of the form

$$A_m u_m + \sum_i A_i u_i = Y_m$$

See the description of the MPC Bulk Data entry in the *MSC Nastran Quick Reference Guide* for a description of the terms on the left hand side of the preceding equation.

The Bulk Data entry **MPCY** in the *MSC Nastran Quick Reference Guide* is used to define a nonhomogeneous multipoint constraint directly. This entry can also be used to define a standard homogenous multipoint constraint.



MPCY

1	2	3	4	5	6	7	8	9	10
MPCY	SID	GM	CM	AM	YM				
		G1	C1	A1	G2	C2	A2		
		G3	C3	A3	...				

Field	Contents
SID	Set identification number of a multi-point constraint set. (Integer > 0)
GM	Identification number of grid or scalar point. (Integer > 0)
CM	Component number. (Any one of the Integers 1 through 6 for grid point; blank or zero for scalar points.)
AM	Coefficient. (Real > 0.0)
YM	Right hand side value. (Real; Default = 0.0)
Gi	Identification number of grid or scalar point. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid point; blank or zero for scalar points.)
Ai	Coefficient. (Real; Default = 0.0)

The Bulk Data entry **MPCD** in the *MSC Nastran Quick Reference Guide* is used to define a load selectable value for Y_m in a nonhomogeneous multipoint constraint.

MPCD

1	2	3	4	5	6	7	8	9	10
MPCD	SID	GM1	CM1	YM1	GM2	CM2	YM2		

Field	Contents
SID	Set identification number of the MPCD entry. (Integer > 0)
GMi	Grid or scalar point identification number. Along with Ci it identifies the Equation of a MPCY. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid point, blank or zero for scalar points)
YMi	Right hand side value of MPC equation. (Real)

The unconstrained stiffness matrix equation in MSC Nastran is

$$[K_{gg}]\{U_g\} = \{P_g\} \quad (6-10)$$

If we apply a nonhomogeneous MPC constraint



$$[R_{\mu g}]\{U_g\} = \{Y_\mu\} \quad (6-11)$$

then [Equation \(6-10\)](#) becomes

$$\begin{bmatrix} \bar{K}_{NN} & K_{NM} \\ K_{MN} & K_{MM} \end{bmatrix} \begin{Bmatrix} U_N \\ U_M \end{Bmatrix} = \begin{Bmatrix} \bar{P}_N + Q_N \\ P_M + Q_M \end{Bmatrix} \quad (6-12)$$

where Q_N and Q_M are the forces of constraint necessary to impose the conditions described in [Equation \(6-11\)](#).

We partition [Equation \(6-11\)](#) as

$$\begin{bmatrix} R_{\mu N} & R_{\mu M} \end{bmatrix} \begin{Bmatrix} U_N \\ U_M \end{Bmatrix} = \{Y_\mu\} \quad (6-13)$$

and solve for U_M to obtain

$$\{U_M\} = \{\theta_M + G_{MN}U_N\} \quad (6-14)$$

where $\theta_M = R_{\mu M}^{-1}Y_\mu$ and $G_{MN} = -R_{\mu M}^{-1}R_{\mu N}$.

Substitution of [Equation \(6-14\)](#) into [Equation \(6-12\)](#) yields

$$[K_{MN} + K_{MM}G_{MN}]\{U_N\} = \{P_M + Q_M - K_{MM}\theta_M\} \quad (6-15)$$

or

$$\{Q_M\} = \underbrace{[K_{MN} + K_{MM}G_{MN}]\{U_N\} - \{P_M\}}_{\text{MPC}} + \underbrace{\{K_{MM}\theta_M\}}_{\text{NHMPC}} \quad (6-16)$$

Since the θ terms of [Equation \(6-14\)](#) are prescribed, any virtual variation of [Equation \(6-14\)](#) results, by definition, in $\delta\theta_M = 0$. Hence, the standard conjugate force transformation holds.

Or,

$$\{Q_N\} = \left\{ -G_{MN}^T Q_M \right\} \quad (6-17)$$



where the minus sign comes from the fact that we are imposing constraints, and the Q_N forces are reactive to the Q_M constraint forces.

Combining the first row of [Equation \(6-12\)](#) with [Equation \(6-17\)](#) yields

$$\{\bar{K}_{NN}U_N + K_{NM}[G_{MN}U_N + \theta_M]\} = \left\{ \bar{P}_N - G_{MN}^T Q_M \right\}$$

and using [Equation \(6-16\)](#) to expand the result, we obtain

$$\begin{aligned} [\bar{K}_{NN} + K_{NM}G_{MN} + G_{MN}^T K_{MN} + G_{MN}^T K_{MM}G_{MN}]\{U_N\} = \\ \left\{ \bar{P}_N + G_{MN}^T P_M \right\} - [K_{NM} - G_{MN}^T K_{MM}]\{\theta_M\} \end{aligned}$$

or

$$\underbrace{[K_{NN}]\{U_N\}}_{\text{MPC}} = \underbrace{\{P_N\} - [K_{NM} + G_{MN}^T K_{MM}]\{\theta_M\}}_{\text{NHMPC}} \quad (6-18)$$

As a simple example, consider a horizontal rod structure. The left and right ends are clamped. However, as [Listing 6-3](#) shows (see MSC_DOC_DIR/doc/linstat/mpcy.dat), there is a break in the structure of 0.1 units of length. If the grid point ID at the left end of the break is A, and the grid point ID at the right end of the break is B, we can tie the structure by the MPC equation

$$U_B - U_A = 0.1$$

In the example, we show how to write the MPC equation in two ways. The first way uses the standard “SLACK” variable method. The second way uses the MPCY entry.

Listing 6-3 Example Rod Structure for MPCY

```
$
$ filename=mpcy.dat
$
ID, MSC, MPCY
SOL 101
CEND
TITLE = DEMONSTRATION USE OF MPCY
SPC = 300
MPC = 300
LOAD = 300
DISP = ALL
OLOAD = ALL
MPCF = ALL
SPCF = ALL
ELFOR = ALL
BEGIN BULK
$
PROD,1,1,1.
```



```

MAT1,1,1.+7,,0.
GRDSET,,,,,,3456
$  

$      "SLACK" VARIABLE METHOD
$  

$ | 1          2          3          4          5 |
$ | 0-----0-----0          0-----0 |
$ |           1           2           3           |  

$  

$      GRIDS 3 and 4 have a 0.1 unit gap between them
$      We wish to impose relationship in x-direction
$          U4 = U3 + 0.1
$      when GRID 3 has a x-direction load of 1.+5
$  

$      GRID,1,,0.,0.,,123456
$      GRID,2,,100.,0.,0.,,23456
$      GRID,3,,200.,0.,0.,,23456
$      GRID,4,,200.1,0.,0.,,23456
$      GRID,5,,300.1,0.,0.,,123456
$  

$      SPOINT,6
$  

$      CROD,1,1,1,2
$      CROD,2,1,2,3
$      CROD,3,1,4,5
$  

$      FORCE,300,3,,1.+5,1.
$  

$      SPC,300,6,1,0.1
$      MPC,300,4,1,-1.,3,1,1.
$,6,1,1.
$  

$      SPOINT with .1 in displacement gap
$  

$  

$      MPCY METHOD
$  

$ | 11          12          13          14          15 |
$ | 0-----0-----0          0-----0 |
$ |           11          12           13           |  

$  

$      GRIDs 13 and 14 have a .1 unit gap between them
$      We wish to impose relationship in x-direction
$          U14 = U13 + 0.1
$      When GRID 13 has a x-direction load of 1.+5
$  

$      GRID,11,,0.,1.,0.,,123456
$      GRID,12,,100.,1.,0.,,23456
$      GRID,13,,200.,1.,0.,,23456
$      GRID,14,,200.1,1.,0.,,23456
$      GRID,15,,300.1,1.,0.,,123456
$  

$      CROD,11,1,11,12
$      CROD,12,1,12,13
$      CROD,13,1,14,15
$  

$      FORCE,300,13,,1.+5,1.
$
```



```
MPCY,300,14,1,1.,0.1
,,13,1,-1.
$ENDDATA
```

The results are:

		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	3.000000E-01	0.0	0.0	0.0	0.0	0.0
	3	G	6.000000E-01	0.0	0.0	0.0	0.0	0.0
	4	G	7.000000E-01	0.0	0.0	0.0	0.0	0.0
	5	G	0.0	0.0	0.0	0.0	0.0	0.0
	6	S	1.000000E-01					
	11	G	0.0	0.0	0.0	0.0	0.0	0.0
	12	G	3.000000E-01	0.0	0.0	0.0	0.0	0.0
	13	G	6.000000E-01	0.0	0.0	0.0	0.0	0.0
	14	G	7.000000E-01	0.0	0.0	0.0	0.0	0.0
	15	G	0.0	0.0	0.0	0.0	0.0	0.0
LOAD VECTOR								
POINT	ID.	TYPE	T1	T2	T3	R1	R2	R3
	3	G	1.000000E+05	0.0	0.0	0.0	0.0	0.0
	6	S	0.0					
	13	G	1.000000E+05	0.0	0.0	0.0	0.0	0.0
FORCES OF SINGLE-POINT CONSTRAINT								
POINT	ID.	TYPE	T1	T2	T3	R1	R2	R3
	1	G	-3.000000E+04	0.0	0.0	0.0	0.0	0.0
	5	G	-7.000000E+04	0.0	0.0	0.0	0.0	0.0
	6	S	7.000000E+04					
	11	G	-3.000000E+04	0.0	0.0	0.0	0.0	0.0
	15	G	-7.000000E+04	0.0	0.0	0.0	0.0	0.0
ELEMENT ELEMENTS (CROD)								
ELEMENT	ID.	AXIAL FORCE	TORQUE	ELEMENT ID.		AXIAL FORCE	TORQUE	
	1	3.000000E+04	0.0	2		3.000000E+04	0.0	
	3	-7.000000E+04	0.0	11		3.000000E+04	0.0	
	12	3.000000E+04	0.0	13		-7.000000E+04	0.0	
FORCES OF MULTIPONT CONSTRAINT								
POINT	ID.	TYPE	T1	T2	T3	R1	R2	R3
	3	G	-7.000000E+04	0.0	0.0	0.0	0.0	0.0
	4	G	7.000000E+04	0.0	0.0	0.0	0.0	0.0
	6	S	-7.000000E+04					
	13	G	-7.000000E+04	0.0	0.0	0.0	0.0	0.0
	14	G	7.000000E+04	0.0	0.0	0.0	0.0	0.0

* * * END OF JOB * * *





7

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
- Initial Strains (DEFORM)
- Load Combinations
- Thermal Loads (TEMP and TEMPD)
- Export of Static Loads
- Fatigue Analysis/Output Request
- STATSUB and BENDL Support in SOL101 Linear Statics



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, CBEAM or CBEND).
- 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, ACCEL, ACCEL1 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.

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 Bulk Data entry [FORCE](#) in the *MSC Nastran Quick Reference Guide* allows you to specify the magnitude and direction of a force vector in any coordinate system as shown below.

FORCE

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 Bulk Data entries [FORCE1](#) and [FORCE2](#) in the *MSC Nastran Quick Reference Guide* is as follows:

FORCE1

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

FORCE2

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 7-1](#). 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.

Part of the input file is shown in [Listing 7-1](#). (See `MSC_DOC_DIR/doc/linstat/edge.dat`) 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 select them using two separate subcases.

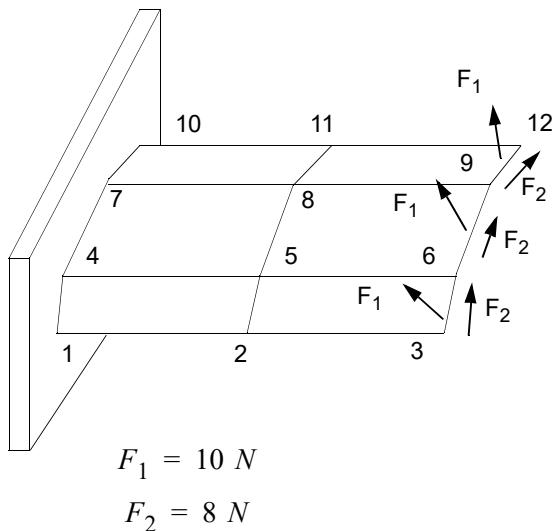


Figure 7-1 Edge Forces on a Plate Element

Listing 7-1

```
$  
$ 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

```

Note that two or more loads can be applied at the same grid points. This is perfectly acceptable, since 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 7-2](#).

LOAD VECTOR								
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 7-2 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 Bulk Data entry **SLOAD** is as follows:

SLOAD

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.

The DAREA entry normally used in dynamics can also be used in statics to create equivalent FORCE, MOMENT and/or SLOAD entries. This is useful for applying loads at grid points in the displacement (or local) coordinate system and switching between static and dynamic analyses. The creation of the equivalent load entries will only happen if there are no LSEQ entries in the Bulk Data section.

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

DAREA

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Field	Contents
SID	Identification number. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point.)
Ai	Scale (area) factor. (Real)

Remarks:

- One or two scale factors may be defined on a single entry.
- Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.
- Component numbers refer to the displacement coordinate system.



4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.
5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.
6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE/MOMENT/SLOAD entries (as appropriate) *if there are no LSEQ Bulk Data entries*.
7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements *if there are no LSEQ Bulk Data entries*.
8. In static analysis, DAREA entries may be used *only if there are no LSEQ Bulk Data entries*. They are ignored if there are any LSEQ Bulk Data entries.

Distributed Loads on Line Elements

PLOAD1

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 Bulk Data entry **PLOAD1** is as follows:

PLOAD1

1	2	3	4	5	6	7	8	9	10
PLOAD1	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.



Field	Contents
	"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.

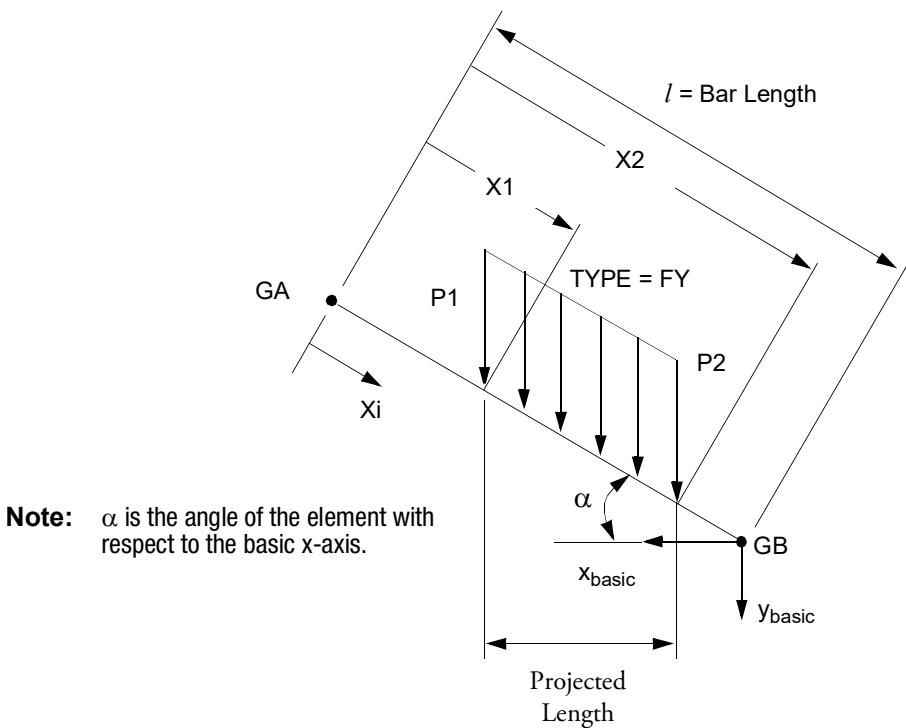


Figure 7-3 PLOAD1 Convention on Beam or Bar Elements

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 7-3](#), 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 7-3](#). 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 7-3](#)). 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 7-4](#).

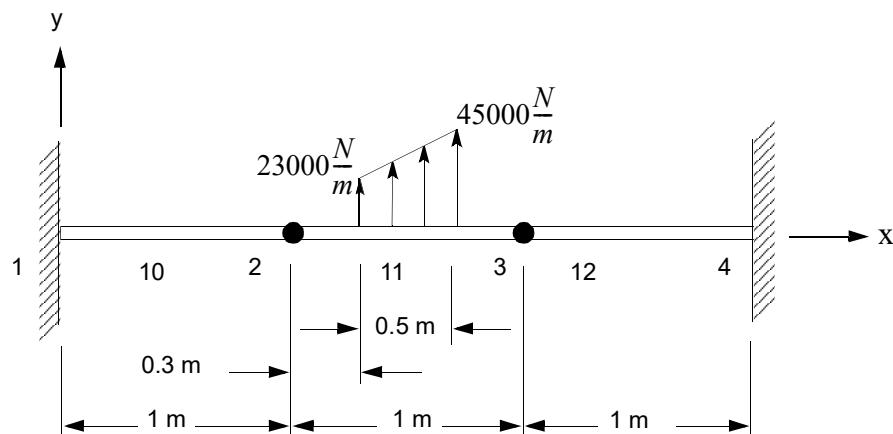


Figure 7-4 Distributed Load on CBAR Elements

The input file for this example is shown in [Listing 7-2](#). (See MSC_DOC_DIR/doc/linstat/distrib.dat)

Listing 7-2 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 the 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 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 7-5](#).

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)		A X I A L	T O R Q U E
E L E M E N T	S T A T I O N	B E N D - M O M E N T		P L A N E	1	P L A N E	1	P L A N E	2	F O R C E	
I D .	(P C T)			P L A N E	1	P L A N E	1	P L A N E	2		
10	.000	.0	-5.982597E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
10	1.000	.0	1.868857E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.000	.0	1.868857E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.050	.0	2.261429E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.100	.0	2.654002E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.150	.0	3.046575E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.200	.0	3.439148E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.250	.0	3.831720E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.300	.0	4.224293E+06	.0	-7.851454E+03	.0	-7.851454E+03	.0	.0	.0	
11	.350	.0	4.587198E+06	.0	-6.646454E+03	.0	-6.646454E+03	.0	.0	.0	
11	.400	.0	4.887105E+06	.0	-5.331454E+03	.0	-5.331454E+03	.0	.0	.0	
11	.450	.0	5.118511E+06	.0	-3.906454E+03	.0	-3.906454E+03	.0	.0	.0	
11	.500	.0	5.275917E+06	.0	-2.371454E+03	.0	-2.371454E+03	.0	.0	.0	
11	.550	.0	5.353823E+06	.0	-7.264532E+02	.0	-7.264532E+02	.0	.0	.0	
11	.600	.0	5.346729E+06	.0	1.028546E+03	.0	1.028546E+03	.0	.0	.0	
11	.650	.0	5.249135E+06	.0	2.893546E+03	.0	2.893546E+03	.0	.0	.0	
11	.700	.0	5.055540E+06	.0	4.868546E+03	.0	4.868546E+03	.0	.0	.0	
11	.750	.0	4.760446E+06	.0	6.953546E+03	.0	6.953546E+03	.0	.0	.0	
11	.800	.0	4.358353E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
11	.850	.0	3.900926E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
11	.900	.0	3.443498E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
11	.950	.0	2.986070E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
11	1.000	.0	2.528644E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
12	.000	.0	2.528644E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	
12	1.000	.0	-6.619903E+06	.0	9.148546E+03	.0	9.148546E+03	.0	.0	.0	

Figure 7-5 CBARAO Output for the Center Element



The shear and moment diagrams are shown in [Figure 7-6](#). 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.

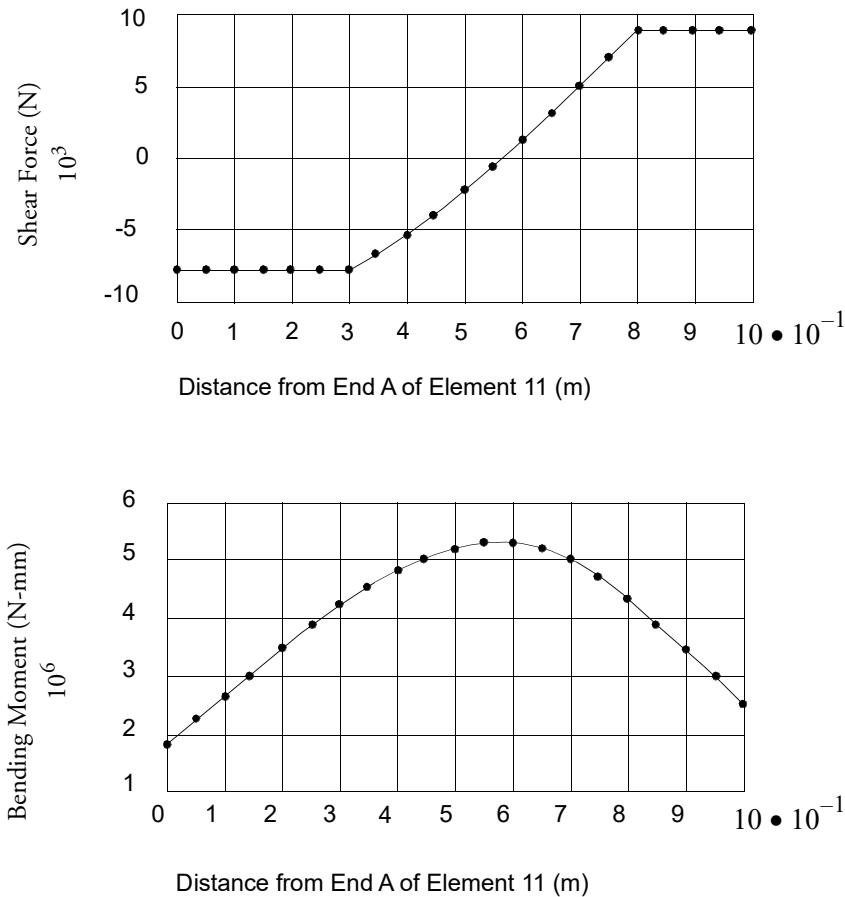


Figure 7-6 Shear and Moment Diagram Generated from CBARAO Output

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 7-7](#).



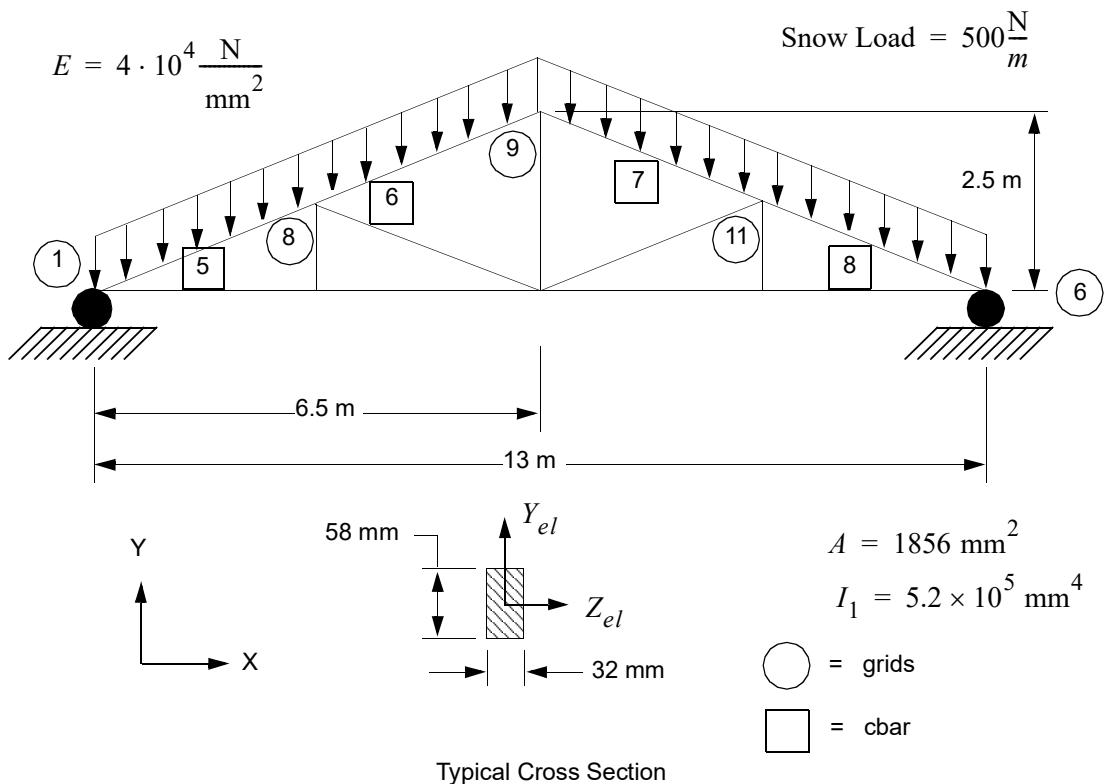


Figure 7-7 Planar Truss Structure with a Snow Loading

The snow load on the truss is 500 N/m acting in the basic Y-direction. Part of the input file containing the grid points and PLOAD1 entries is shown in [Listing 7-3](#). (See MSC_DOC_DIR/doc/linstat/pload1.dat)

Listing 7-3 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  

CBAR   6           1       8       9       1.0    1.      0.0  

CBAR   7           1       9       11      1.0    1.      0.0  

CBAR   8           1       11      6       1.0    1.      0.0  

$  

PLOAD1 1           5       FY     FRPR    0.0   -.5    1.     -.5  

PLOAD1 1           6       FY     FRPR    0.0   -.5    1.     -.5

```



```
PLOAD1 1      7      FY      FRPR    0.0     -.5     1.     -.5
PLOAD1 1      8      FY      FRPR    0.0     -.5     1.     -.5
```

To see the effect of using projected forces, the resulting SPC forces are shown in [Figure 7-8](#). 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.

FORCES OF SINGLE-POINT CONSTRAINT								
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 7-8 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.

PLOADB3

The Bulk Data entry, [PLOADB3](#) defines a distributed load over the entire length of a CBEAM3 element axis. The load is applied along the line of the shear center.

PLOADB3

1	2	3	4	5	6	7	8	9	10
PLOADB3	SID	EID	CID	N1	N2	N3	TYPE	SCALE	
	P(A)	P(B)	P(C)						

Field	Contents
SID	Load set identification number. (Integer > 0; Required)
EID	CBEAM3 element identification number. (Integer > 0, Required)
CID	Coordinate system for load definition. (Character or Integer; Default = "BASIC")
	"LOCAL" Local coordinate system;
	"ELEMENT" Element coordinate system;
	"BASIC" or 0 Basic coordinate system;
n (n>0):	Any user-specified coordinate system identification number.
N1, N2, N3	Load vector components measured in coordinate system specified by CID. (Real; at least one Ni ≠ 0.0)



Field	Contents
TYPE	Type of applied load. (Character = “FORCE”, “MOMENT” or “BIMOMENT”; Required)
SCALE	Load vector scale factor. (Real; Default = 1.0)
P(j)	Magnitudes of load at j ($j=A, B$ and C). (Real; Default = 0.0)

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 PLOAD*i* 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 Bulk Data entry **PLOAD** is as follows:

PLOAD

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 format for the Bulk Data entry PLOAD2 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 format of Bulk Data entry **PLOAD2** is as follows:

PLOAD2

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

Alternate Format and Example:



PLOAD2	SID	P	EID1	“THRU”	EID2				
PLOAD2	1	30.4	16	THRU	48				

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

The 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 Bulk Data entry **PLOAD4** is as follows:

PLOAD4

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	SORL	LDIR			

Alternate Format

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

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0; for the “THRU” option, EID1 < EID2)
EID1	
EID2	



Field	Contents
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1.)
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA, CPENTA or CPYRAM element. Required data for quadrilateral faces of CHEXA, CPENTA and CPYRAM elements only. For CPYRAM element triangle faces, G1 and G3 are adjacent corner nodes on the quadrilateral face, and the load is applied on the triangular face which includes those grids. G3 must be omitted for a triangular surface on a CPENTA element.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)
CID	Coordinate system identification number. See Remark 2. (Integer ≥ 0 ; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark 2. (Real)
SORL	The character string SURF or LINE. SURF means the surface load acting on the surface of the element and LINE means the consistent edge loads acting on the edges of the element. The default is SURF. See Remark 13.
LDIR	Denote the direction of the line load (SORL=LINE), character string X, Y, Z, TANG, or NORM. The default is NORM. See Remark 14.

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
4. For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)



5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
6. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
7. G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.
8. The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.
9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.
10. For the CQUADR and CTRIAR element, only pressure that acts normal to the element is computed properly. Surface tractions are not resolved into moments normal to the element.
11. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
12. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
13. The SORL field is ignored by all elements except QUADR and TRIAR. For QUADR or TRIAR only, if SORL=LINE, the consistent edge loads are defined by the PLOAD4 entry. P1, P2, P3 and P4 are load per unit length at the corner of the element. If all four Ps are given, then the line loads along all four edges of the element are defined. If any P is blank, then the line loads for only two edges are defined. For example, if P1 is blank, the line loads of the two edges connecting to G1 are zero. If two Ps are given, then the line load of the edge connecting to the two grid points is defined. If only one P is given, the second P value default to the first P value. For example, P1 denotes that the line load along edge G1 and G2 has the constant value of P1.
14. The direction of the line load (SORL=LINE) is defined by either (CID, N1, N2, N3) or LDIR. Fatal error will be issued if both methods are given. TANG denotes that the line load is in tangential direction of the edge, pointing from G1 to G2 if the edge is connecting G1 and G2. NORM denotes that the line load is in the mean plan, normal to the edge, and pointing outward from the element. X, Y, or Z denotes the line load is in the X, Y, or Z direction of the element coordinate system. If both (CID, N1, n2, N3) and LDIR are blank, then the default is LDIR=NORM.



15. For SOL 400, with large displacement (PARAM,LGDISP), higher-order 6-node triangles (CTRIA6), 8-node quadrilateral (CQUAD8), 20-node hexagonal (CHEXA), or 15-node pentahedral (CPENTA), the load is not calculated correctly. Use lower-order elements if distributed loads are required in a large displacement analysis.

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 7-9](#). PLOAD4 entries are to be used to apply a normal pressure to each of the six CQUAD4 elements.

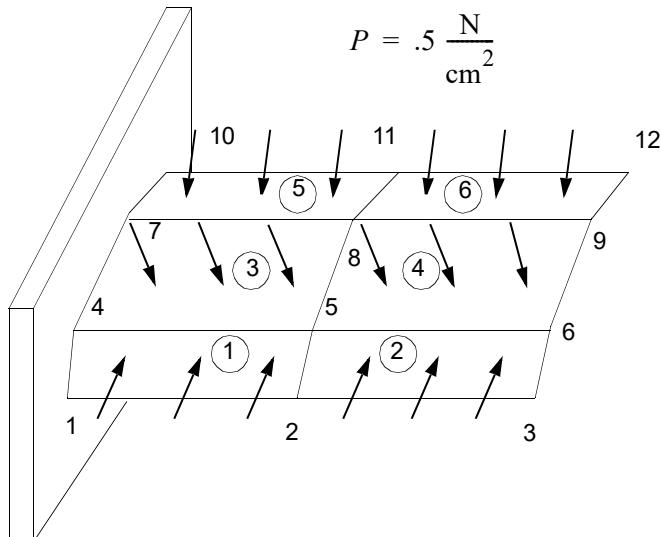


Figure 7-9 Pressure Loads on the CQUAD4s

Part of the input file is shown in [Listing 7-4](#). (See MSC_DOC_DIR/doc/linstat/press.dat) 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 7-4 CQUAD4 Elements with Pressure Loads

```
$  
$ FILENAME - PRESS.DAT  
$  
$ UNITS ARE CM, N
```

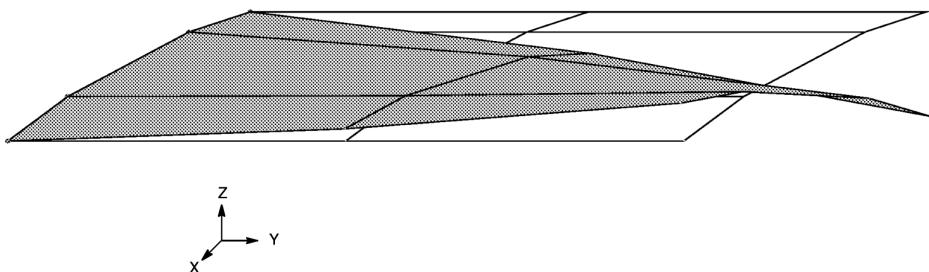


```

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      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 7-10](#). 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](#).



[Figure 7-10](#) Displacements Due to Pressure Load

Gravity and Centrifugal Force

Gravity (GRAV)

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 Bulk Data entry **GRAV** is as follows:

GRAV

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.

Centrifugal (RFORCE)

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 Bulk Data entry **RFORCE** is as follows:

RFORCE

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB	IDRF						

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer ≥ 0)
CID	Coordinate system defining the components of the rotation vector. See Remark 16. (Integer ≥ 0 ; Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real)



Field	Contents
R1, R2, R3	Rectangular components of rotation vector \vec{R} . The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$ unless A and RACC are both zero)
METHOD	Method used to compute centrifugal forces due to angular velocity. For angular acceleration, see Remark 13. (Integer = 1 or 2; Default = 1)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 15. (Integer; Default = 0)
IDRF	ID indicating to which portion of the structure this particular RFORCE entry applies. IDRF corresponds to a SET3 entry specifying the elements with this acceleration. A BRKSQ entry may also be specified with a matching IDRF entry. (Integer; Default = 0)

Remark:

1. The forces that are created with the RFORCE entry act on the structure as follows: the forces that are defined with the RFORCE entry for a constant angular velocity (A), act in the positive radial direction. These forces represent the inertia forces on the structure due to a constant angular velocity. The forces that are defined with the RFORCE entry for a constant angular acceleration (RACC), act in the same direction as the angular acceleration. These forces would be opposite to the inertia forces on the structure due to a constant angular acceleration. In Figure 7-11, the force vector at grid point Gi is given by

$$\{\vec{F}\}_i = [m]_i [\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)] \quad (7-1)$$

where:

angular velocity	=	$\vec{\omega} = 2\pi A \cdot \vec{R}$ (radians/unit time)
angular acceleration	=	$\vec{\alpha} = 2\pi RACC \cdot \vec{R}$ (radians/unit time)
$[m]_i$	=	3 × 3 translational mass matrix at grid point Gi

Note: The equation for \vec{F}_i will have additional terms if the mass is offset or I_{23}^i, I_{13}^i terms exist relative to the rotation axes and METHOD = 1 is selected.



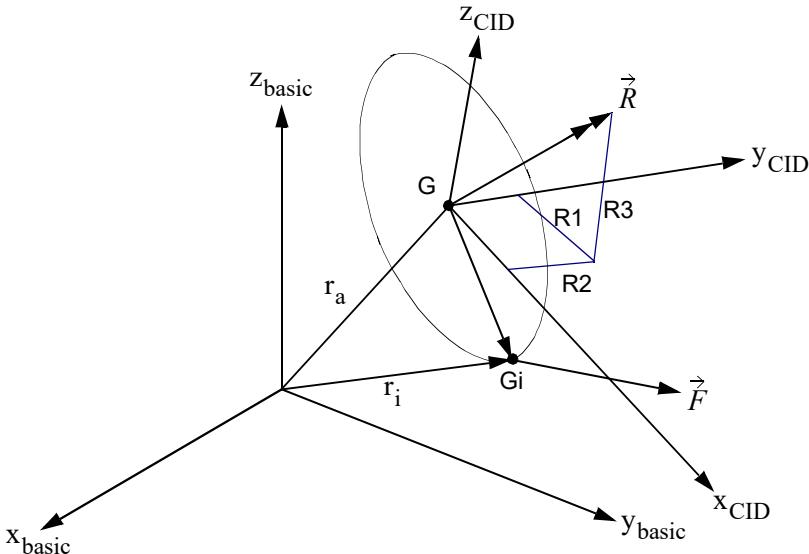


Figure 7-11 RFORCE Vector at Grid Point Gi

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 7-1](#).

Table 7-1 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, CTRIAX6, CHEXA, CPENTA, CPYRAM 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.

Note for PARAM,COUPMASS=-1 (the default) the generation of lumped mass matrices that contains only translational components for the elements listed above. Notable exceptions to this are the CBAR and CBEAM elements, both of which will yield rotational and coupling terms in order to preserve the mass center when element offsets are defined. This offset mass is 'lumped' in the sense that it has low matrix rank, and



is ‘coupled’ in the sense that there are non-zero off diagonal terms in the mass matrix. The CBEAM element will also yield a mass moment of inertia about the local X axis of the element, and if NASTRAN BARMASS > 0, then this is also true of the CBAR element.

In order to yield a lumped mass matrix containing translational components only for the CBAR and CBEAM elements, set SYSTEM(414)=1, along with the default value for PARAM,COUPMASS (-1). The default value (0) for SYSTEM(414) produces the coupled mass matrices for CBAR and CBEAM.

Set Selection of GRAV and RFORCE (GRAVSET, RFORSET)

Above, GRAV and RFORCE loading was described as a global loading condition. However, in certain application, there is a need to apply these forces only to certain portions of the model and not to the complete model.

GRAVSET

Default = 0

GRAVSET=n, where n is a Bulk Data SET1 entry, allows the user to apply gravity loading (GRAV) to just a portion of the structure. GRAVSET like GRAV loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the gravity loading is to be applied.

GRAVSET can appear in either the Bulk Data or ABOVE or IN the 1st Subcase. A GRAVSET appearing in any other Subcase will be ignored and may cause incorrect results. A GRAVSET appearing in or above the first subcase takes precedence over a GRAVSET appearing in Bulk Data. See also the companion RFORSET entry.

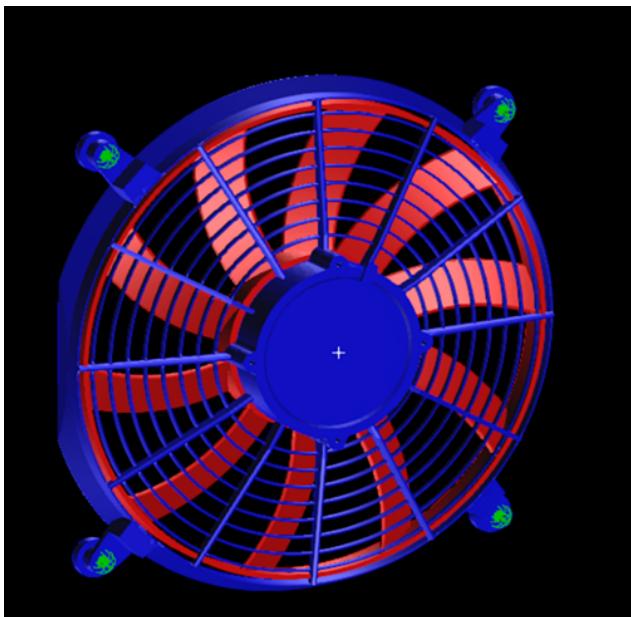
RFORSET

Default = 0

RFORSET=n, where n is a Bulk Data SET1 entry, allows the user to apply rotational force loading (RFORCE) to just a portion of the structure. RFORSET like RFORCE loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the rotational force loading is to be applied. RFORSET can appear in either the Bulk Data or ABOVE or IN the 1st Subcase. A RFORSET appearing in any other Subcase will be ignored and may cause incorrect results. A RFORSET appearing in or above the first subcase takes precedence over a RFORSET appearing in Bulk Data. See also the companion GRAVSET entry.



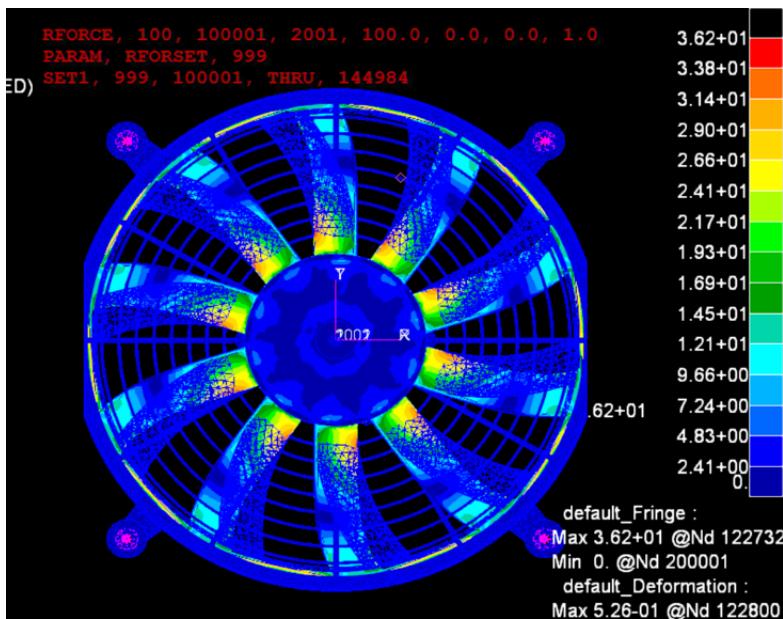
As an example of usage, consider an Automotive cooling fan assembly shown in figure.



The housing (blue) is fixed in place while the fan blade (red) rotates via an electric motor.

RFROCE loading applied to the housing will lead to incorrect loading and unrealistic stress in the housing. Only RFORCE loading should be applied to the rotating blade via PARAM, RFORSET, n; where n is the ID of a SET1 Bulk Data entry. SET1, n lists the GRID ID that will have the load vector terms for the RFORCE. This gives the distribution:





Acceleration Loads (ACCEL and ACCEL1)

ACCEL and ACCEL1 Bulk Data entries are used to apply an acceleration load that varies across the structure. The ACCEL entries apply acceleration loads which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry. The ACCEL1 entry applies static acceleration load at individual grid points. Bulk Data entries **ACCEL** and **ACCEL1** in the *MSC Nastran Quick Reference Guide* are used in the same way as other load entries (such as GRAV, FORCE, and MOMENT, etc.) through the MSC Nastran Case Control commands.

ACCEL

1	2	3	4	5	6	7	8	9	10
ACCEL	SID	CID	N1	N2	N3	DIR			
	LOC1	VAL1	LOC2	VAL2	Continues in Groups of 2				

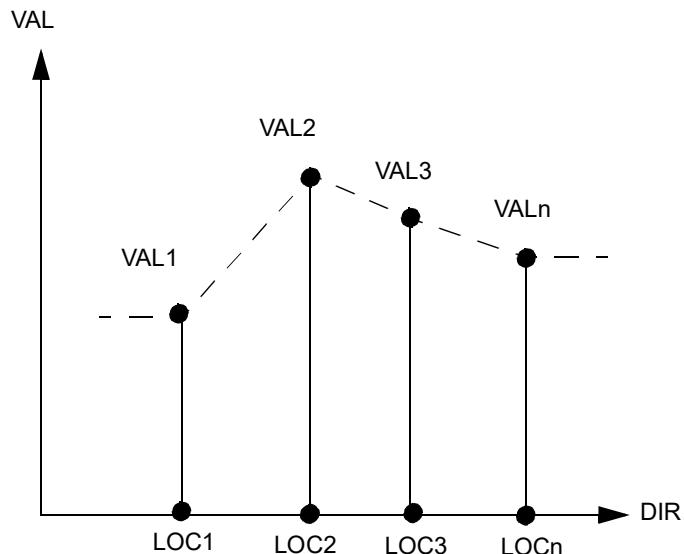
Field	Contents
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer > 0; Default = 0)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
DIR	Component direction of acceleration variation. (Character; one of X, Y, and Z)



Field	Contents
LOCi	Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)
VALi	The load scale factor associated with location LOCi. (Real)

Remarks:

1. For all grids of the model, the acceleration vector is defined by $\vec{a} = VAL \cdot \vec{N}$, where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of \vec{a} is equal to VAL times the magnitude of \vec{N} . The scale factor VAL for each grid is found linearly interpolating the DIR coordinate of the grid between table values $LOCi/VALi$. If the GRID point coordinate in coordinate system CID is outside the range of the table, VAL is determined either from $VAL1$ or $VALn$ (the last value, see the following figure).
2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL1 loads, by specification on a LOAD entry. The SID on an ACCEL entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.
5. The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.
6. A minimum of two pairs of {LOCi, VALi} data must be specified.



ACCEL1

1	2	3	4	5	6	7	8	9	10
ACCEL1	SID	CID	A	N1	N2	N3			
	GRIDID1	GRIDID2	-etc.-						

Field	Contents
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer > 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
GRIDIDI LIST	List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer > 0)

Remarks:

1. The acceleration vector is defined by $\vec{a} = A \cdot \vec{N}$, where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of \vec{a} is equal to A times the magnitude of \vec{N} .
2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL loads, by specification on a LOAD entry. The SID on an ACCEL1 entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.

Defining Mass (CMASSi and CONMi)

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 $7.32 \cdot 10^{-4}$ lb·sec²/in⁴.

The format of the Bulk Data entry CMASS1 is as follows:

CMASS1

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			



Field	Contents
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PMASS entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid or scalar point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; blank or zero if scalar point)

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

PMASS

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

Field	Contents
PID _i	Property identification number. ($\text{Integer} > 0$)
M _i	Value of scalar mass. (Real)

The format of the Bulk Data entry CONM1 entry is as follows:

CONM1

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

Field	Contents
EID	Unique element identification number. ($\text{Integer} > 0$)
G	Grid point identification number. ($\text{Integer} > 0$)
CID	Coordinate system identification number for the mass matrix. ($\text{Integer} \geq 0$)
M _{ij}	Mass matrix values. (Real)

The other concentrated mass entries are **CMASS2**, **CMASS3**, **CMASS4** and **CONM2**.

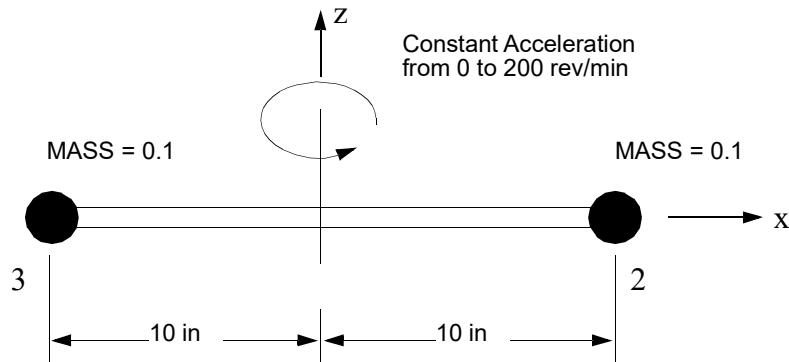
The nonstructural mass defined on the property entries is mass that is added to the structure in addition to the structural mass from the elements. For one-dimensional elements, the units are mass/unit length. For two-dimensional and three-dimensional elements, the units are mass/unit area and mass/unit 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/in³ 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](#) and [Glossary](#).)

Consider the two rotating masses shown in [Figure 7-12](#). The masses are accelerated at constant angular acceleration of 20 rev/sec² from 0 to 200 rev/min. The goal is to determine the axial force in the CBAR as a function of angular velocity.



The input file is shown in [Listing 7-5](#). (See MSC_DOC_DIR/doc/linstat/rotate.dat) 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 7-5 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
$
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     1.33333 0.0   0.0   1.0   1
20.
RFORCE 4          1     2.0   0.0   0.0   1.0   1
20.
RFORCE 5          1     2.66667 0.0   0.0   1.0   1
20.
RFORCE 6          1     3.33333 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 7-13](#). 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 .



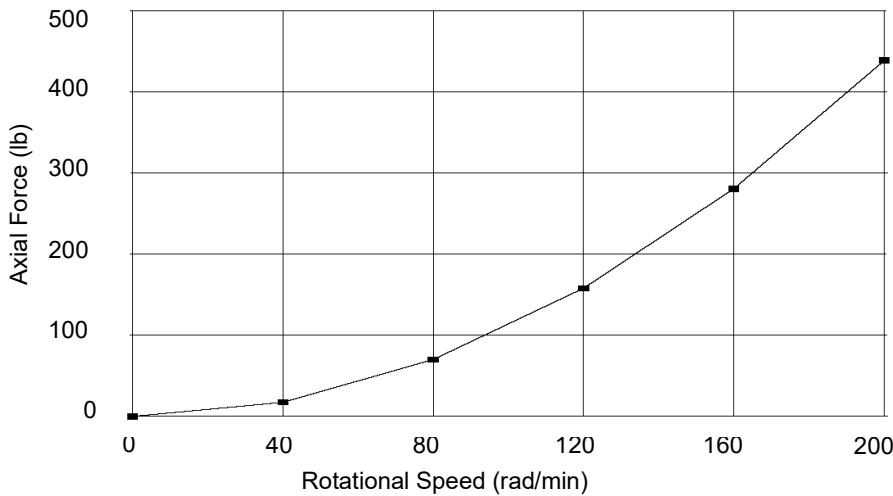


Figure 7-13 Axial Force in the Bar as a Function of Angular Velocity

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 \(TEMP and TEMPD\)](#).

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

DEFORM

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.
EID _i	Element number.
D _i	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 7-14. The goal is to impose an initial strain equal to 100 $\mu\text{cm}/\text{cm}$ in the center member.

$$E = 7.1 \times 10^6 \frac{\text{N}}{\text{cm}^2}$$

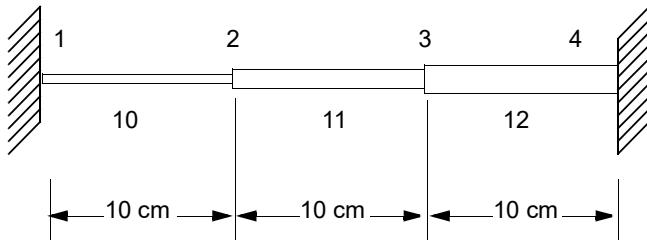


Figure 7-14 Three-Bar Structure with Initial Strain

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 $\mu\text{m}/\text{cm}$ strain.

$$\begin{aligned}\sigma_{11} &= E\varepsilon \\ &= (7.1 \times 10^6)(100 \times 10^{-6}) = 710 \frac{\text{N}}{\text{cm}^2}\end{aligned}$$

$$\begin{aligned}P &= A\sigma_{11} \\ &= (1.5)(710) = 1065 \text{ N}\end{aligned}$$

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.

$$\sigma_{10} = \frac{P}{A_{10}} = \frac{1065}{1} = 1065 \frac{\text{N}}{\text{cm}^2}$$

$$\sigma_{12} = \frac{P}{A_{12}} = \frac{1065}{2} = 532.5 \frac{\text{N}}{\text{cm}^2}$$

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.



$$\varepsilon_{10} = \frac{\sigma_{10}}{E} = 150 \cdot 10^{-6}$$

$$\varepsilon_{12} = \frac{\sigma_{12}}{E} = 75.0 \cdot 10^{-6}$$

Therefore,

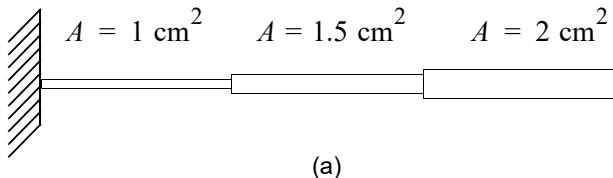
$$\Delta L_{10} = L_{10}\varepsilon_{10} = (10)(150 \cdot 10^{-6}) = 1.5 \cdot 10^{-3} \text{ cm}$$

$$\Delta L_{11} = L_{11}\varepsilon_{11} = (10)(150 \cdot 10^{-6}) = 1.0 \cdot 10^{-3} \text{ cm}$$

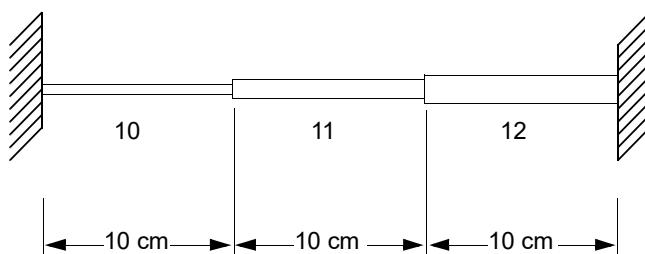
$$\Delta L_{12} = L_{12}\varepsilon_{12} = (10)(75 \cdot 10^{-6}) = 0.75 \cdot 10^{-3} \text{ cm}$$

These values are entered on the DEFORM Bulk Data entry as shown in [Listing 7-6](#). (See [MSC_DOC_DIR/doc/linstat/strain.doc](#)) 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 7-15\(a\)](#). 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 7-15\(b\)](#), is the constrained configuration that is of interest. If the calculations are correct, the strain in the center element should be the required 100 $\mu\text{cm}/\text{cm}$.



(a)



(b)

Figure 7-15 Three-Bar Structures with Initial Strain



The input file is shown in [Listing 7-6](#). (See MSC_DOC_DIR/doc/linstat/strain.dat)

Listing 7-6 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      0.0      23456
GRID     3          20.0      0.0      0.0      23456
GRID     4          30.0      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 7-16](#). 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/cm².

Since $E = 7.1 \times 10^6$, the strain in element 11 is equal to $\epsilon = E/\sigma = 100 \mu\text{cm}/\text{cm}$ 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.



DISPLACEMENT		VECTOR								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3			
1	G	.0	.0	.0	.0	.0	.0			
2	G	1.50000E-03	.0	.0	.0	.0	.0			
3	G	2.50000E-03	.0	.0	.0	.0	.0			
4	G	3.25000E-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										
ELEMENT ID.	SA1 SB1	S A T R E S S E S I N B A R	SA2 SB2	SA3 SB3	SA4 SB4	E L E M E N T S	(C B A R)	SA-MAX	SA-MIN	M.S.-T M.S.-C
10	.0	.0	.0	.0	.0	AXIAL STRESS	SA-MAX SB-MAX	.0	.0	
11	.0	.0	.0	.0	.0	.0	.0	.0	.0	
12	.0	.0	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0	.0		.0	.0	.0	
SUBCASE 2										
ELEMENT ID.	SA1 SB1	S T R E S S E S I N B A R	SA2 SB2	SA3 SB3	SA4 SB4	E L E M E N T S	(C B A R)	SA-MAX	SA-MIN	M.S.-T M.S.-C
10	.0	.0	.0	.0	.0	-1.065000E+03	-1.065000E+03 -1.065000E+03	-1.065000E+03	-1.065000E+03	
11	.0	.0	.0	.0	.0	-7.100000E+02	-7.100000E+02 -7.100000E+02	-7.100000E+02	-7.100000E+02	
12	.0	.0	.0	.0	.0	-5.325000E+02	-5.325000E+02 -5.325000E+02	-5.325000E+02	-5.325000E+02	
	.0	.0	.0	.0	.0		.0	.0	.0	

Figure 7-16 Stress and Displacement Output of the Three Bar Structure

Thermal Loads (TEMP and TEMPD)

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 (Bulk Data entries TEMP and TEMPD in the *MSC Nastran Quick Reference Guide* and interpolated to grid points within elements.

TEMP

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Field	Contents
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ti	Temperature. (Real)



TEMPD

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

Field	Contents
SID _i	Temperature set identification number. (Integer > 0)
T _i	Default temperature value. (Real)

Alternatively, temperature data can be specified on an element-by-element basis as shown in [Table 7-2](#).

Table 7-2 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.	TEMPPRB
CBAR, CBEAM, CBEND	Average temperature and cross-sectional temperature gradients at ends A and B.	TEMPPRB
CBEAM3	Temperature field and gradients along the beam	TEMPPB3
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 [TEMPERATURE \(Case\)\(INIT\)](#) and [TEMPERATURE \(Case\)\(LOAD\)](#) 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 selected in the SUBCOM subcase; however, for thermal loads they do need to be selected. 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:



$$T_{subcom} = T_0 + \sum_{i=1}^n a_i(T_i - T_0) \quad (7-2)$$

where:

T_0	= reference temperature
T_i	= temperature applied in Subcase i
a_i	= subseq factor for Subcase i

The TEMP(LOAD) must be supplied through the standard temperature Bulk Data entries such as TEMPD, TEMP, etc., and selected in the SUBCOM Case Control Section. This is best illustrated with part of an example problem shown in [Listing 7-7](#).

Listing 7-7 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 7-17](#). The tent is subjected to thermal radiation from partial exposure to the sun. The ground is held at 75 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](#).

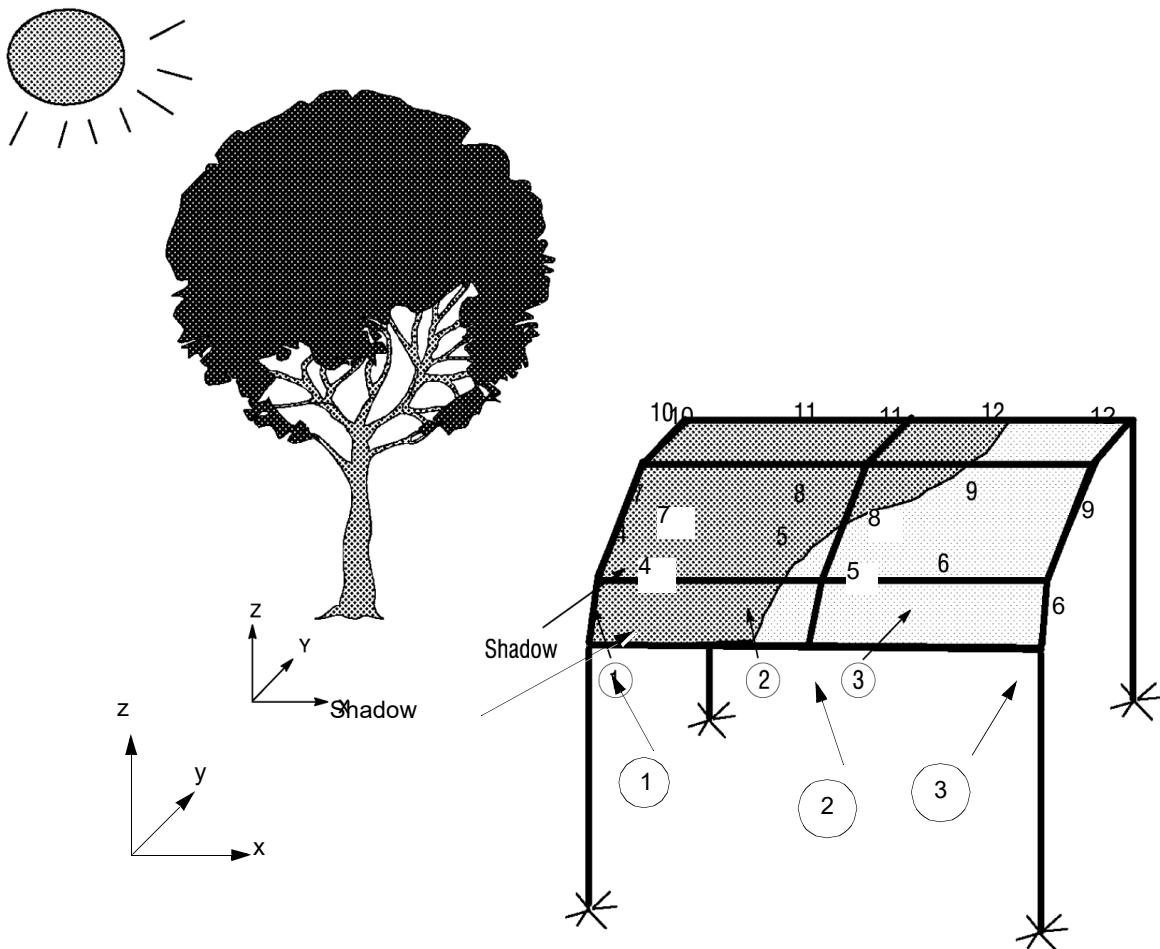


Figure 7-17 Umbrella Tent With Thermal Loads

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. Part of a Solution 153 input file is shown for this model and is included on the delivery media with the name 'heat.dat'. (See `MSC_DOC_DIR/doc/linstat/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. Part of the Solution 101 input file used for the stress analysis is shown in [Listing 7-8](#). (See `MSC_DOC_DIR/doc/linstat/thermal.dat`) The temperature file from the Solution 153 run



is shown in [Listing 7-9](#). Note the use of the `INCLUDE` statement, which inserts the temperature input file at that location when the job is submitted. See [Database Management](#) for a further discussion of the `INCLUDE` statement.

Listing 7-8 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(LOAD) = 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.        .
```

Listing 7-9 Temperature Distribution Generated with Solution 153

\$TITLE = TENT THERMAL ANALYSIS SUBJECTED TO RADIATION	
1	
\$SUBTITLE=THE BASE TEMPERATURE AT 75 DEGREE F	2
\$LABEL =	3
\$TEMPERATURE	4
\$REAL OUTPUT	5
\$SUBCASE ID = 1	6
\$LOAD FACTOR = 1.0000000E+00	6
TEMP* 1 1 5.300640E+02	
TEMP* 1 2 5.557228E+02	
TEMP* 1 3 5.300640E+02	
TEMP* 1 4 5.288076E+02	
TEMP* 1 5 5.326366E+02	
TEMP* 1 6 5.288076E+02	
TEMP* 1 7 4.691609E+02	
TEMP* 1 8 4.726500E+02	
TEMP* 1 9 4.691609E+02	
TEMP* 1 10 4.074296E+02	
TEMP* 1 11 4.273944E+02	
TEMP* 1 12 4.074296E+02	
TEMP* 1 99 7.500000E+01	
TEMP* 1 101 7.500000E+01	
TEMP* 1 103 7.500000E+01	
TEMP* 1 110 7.500000E+01	
TEMP* 1 112 7.500000E+01	
TEMP* 1 999 0.000000E+00	



The OLOAD and stress results for this model are shown in [Figure 7-18](#). The OLOAD output shows the applied loads generated by the imposed temperature. 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		-6.588088E+04	-7.761009E+04	1.587762E+04	0.0	0.0	0.0		
2	G		0.0	-1.179019E+05	-4.716078E+04	0.0	0.0	0.0		
3	G		6.588088E+04	-7.761009E+04	1.587762E+04	0.0	0.0	0.0		
4	G		-1.035648E+05	-2.895253E+01	3.104404E+04	0.0	0.0	0.0		
5	G		0.0	1.484770E+03	4.716078E+04	0.0	0.0	0.0		
6	G		1.035648E+05	-2.895253E+01	3.104404E+04	0.0	0.0	0.0		
7	G		-9.263741E+04	1.559617E+04	2.481715E+04	0.0	0.0	0.0		
8	G		-3.637979E-12	2.217270E+04	3.769779E+04	0.0	0.0	0.0		
9	G		9.263741E+04	1.559617E+04	2.481715E+04	0.0	0.0	0.0		
10	G		-4.937203E+04	6.204287E+04	9.459667E+03	0.0	0.0	0.0		
11	G		-3.637979E-12	9.424447E+04	-3.769779E+04	0.0	0.0	0.0		
12	G		4.937203E+04	6.204287E+04	9.459667E+03	0.0	0.0	0.0		
101	G		0.0	0.0	-4.692165E+04	0.0	0.0	0.0		
103	G		0.0	0.0	-4.692165E+04	0.0	0.0	0.0		
110	G		0.0	0.0	-3.427682E+04	0.0	0.0	0.0		
112	G		0.0	0.0	-3.427682E+04	0.0	0.0	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	7.820048E+01	1.190432E+03	-1.190432E+03	-7.820048E+01	-2.881437E+01	1.161618E+03	-1.219246E+03	2.1E+01		
	-4.433472E+02	4.515958E+02	-4.515958E+02	4.433472E+02	4.433472E+02	4.227815E+02	-4.804102E+02	2.0E+01		
102	-4.433472E+02	4.515958E+02	-4.515958E+02	4.433472E+02	-2.881437E+01	4.227815E+02	-4.804102E+02	2.1E+01		
103	7.820048E+01	1.190432E+03	-1.190432E+03	-7.820048E+01	1.619600E+01	1.161618E+03	-1.219246E+03	2.0E+01		
	3.438357E+02	-3.438357E+02	4.929828E+02	4.929828E+02	5.091788E+02	-4.767868E+02	4.8E+01			
	-1.219539E+02	1.840813E+02	-1.840813E+02	1.219539E+02	2.002773E+02	-1.678853E+02	5.1E+01			
STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)										
ELEMENT	ID.	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X NORMAL-Y SHEAR-XY			PRINCIPAL STRESSES (ZERO SHEAR) ANGLE MAJOR MINOR			VON MISES	
0	1	-2.500000E-02	-1.430361E+00	-5.638787E-01	3.251771E+00	48.7945	2.283385E+00	-4.277625E+00	5.768829E+00	
		2.500000E-02	-1.430361E+00	-5.638787E-01	3.251771E+00	48.7945	2.283385E+00	-4.277625E+00	5.768829E+00	
0	2	-2.500000E-02	-1.430361E+00	-5.638787E-01	-3.251771E+00	-48.7945	2.283385E+00	-4.277625E+00	5.768829E+00	
0	3	-2.500000E-02	-1.235332E+00	-8.364280E-02	-1.319849E+00	-31.7251	2.283385E+00	-4.277625E+00	5.768829E+00	
0	4	-2.500000E-02	1.235332E+00	-8.364280E-02	-1.319849E+00	31.7251	2.051286E+00	-8.995960E-01	2.619613E+00	
0	5	-2.500000E-02	1.235332E+00	-8.364280E-02	1.319849E+00	31.7251	2.051286E+00	-8.995960E-01	2.619613E+00	
0	6	-2.500000E-02	1.235332E+00	-8.364280E-02	1.319849E+00	31.7251	2.051286E+00	-8.995960E-01	2.619613E+00	
	2.500000E-02	-1.409853E+00	-6.633537E-01	3.347315E-01	69.0571	-5.352446E-01	-1.537962E+00	1.352267E+00		
		-1.409853E+00	-6.633537E-01	3.347315E-01	69.0571	-5.352446E-01	-1.537962E+00	1.352267E+00		

Figure 7-18 Thermally Induced Loads and Stresses in the Tent Structure

Note: Only a partial listing of the CBAR element output is included.

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 7-19](#) 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 7-19](#), 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 Bulk Data entry [LOAD](#) is as follows:

LOAD

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.



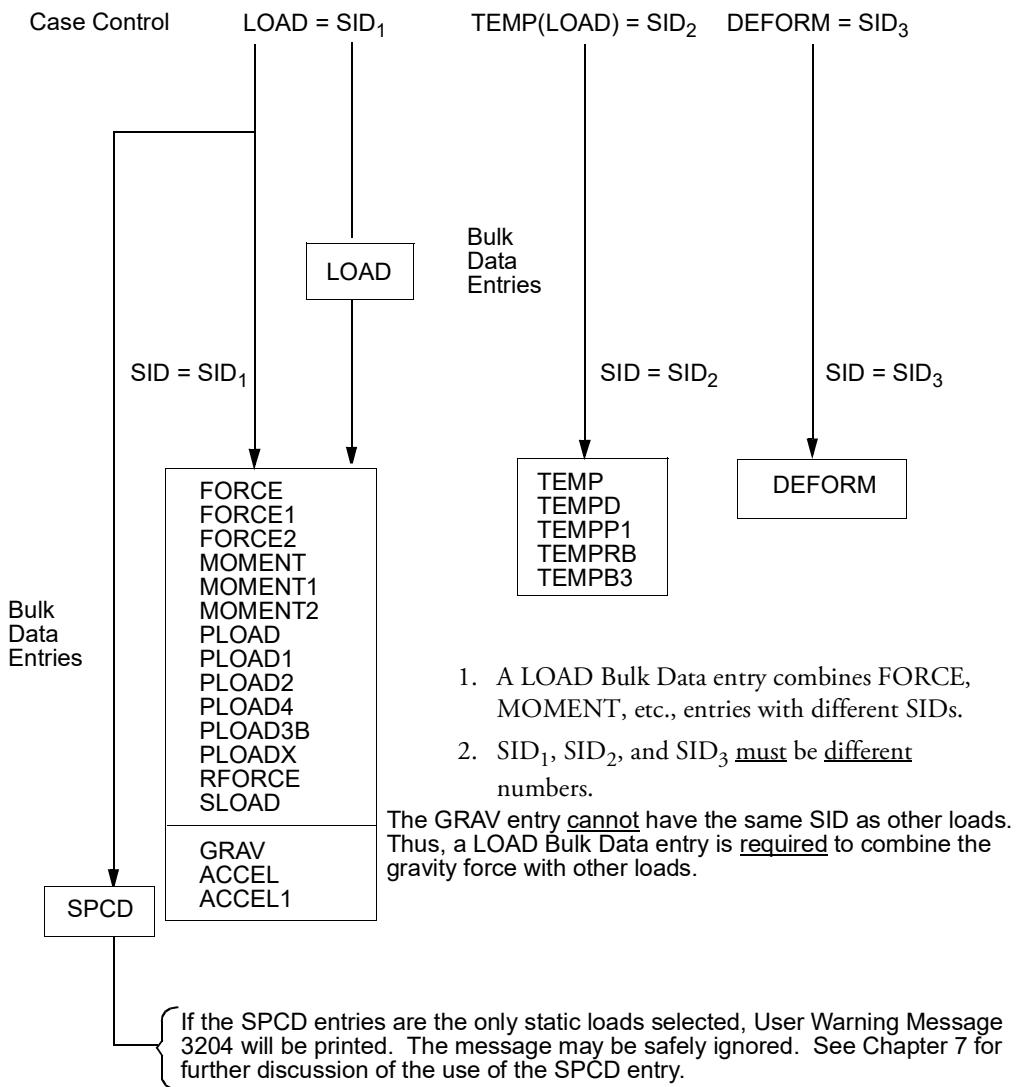


Figure 7-19 Load Combinations

As an example of the LOAD entry combining two loads, consider the umbrella tent shown in [Figure 7-20](#).



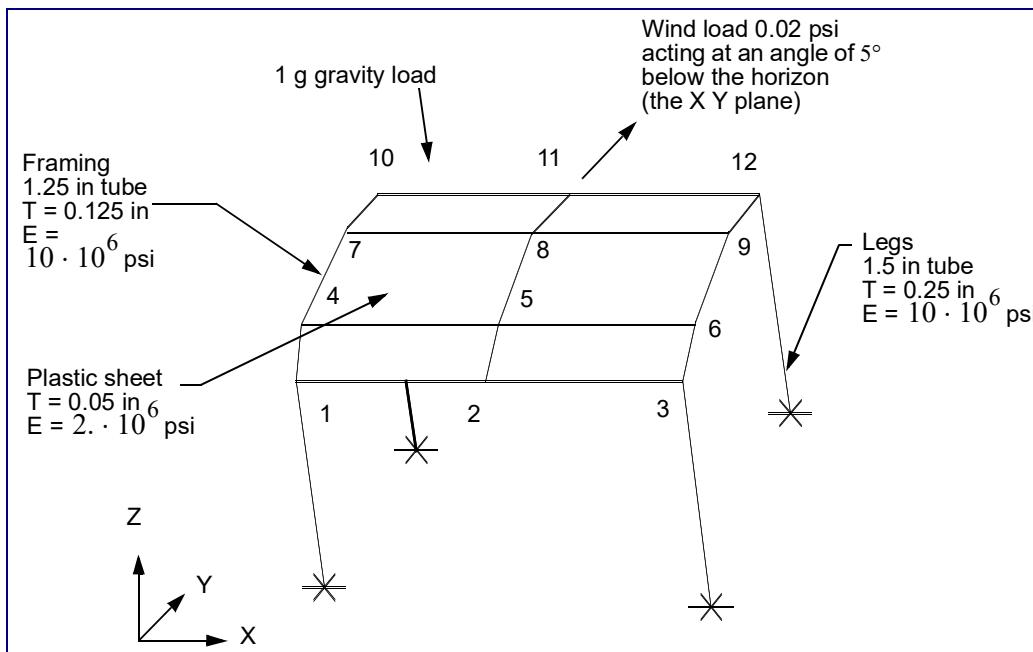


Figure 7-20 Umbrella Tent Subjected to Gravity and Wind Loads

Part of the input file containing the combined loading is shown in Listing 7-10. (See MSC_DOC_DIR/doc/linstat/tent.doc) 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 7-10 Umbrella Tent with Wind and Gravity Loads

```

$  
$FILENAME - TENT.DAT  
$  
:  
LOAD=5  
:  
BEGIN BULK  
$  
CORD2R 1 0 0. 0. 0. 0. .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.

Export of Static Loads

Functionality exists in MSC Nastran to facilitate load transfer from one run to the next in three important ways:

1. MSC Nastran can use Grid Point Force methodology to extract the load on a user defined free body. This load is output on a PG-like matrix that has an associated NAME, ID and optional label. A BGPDT data block is also produced that contains information on the grids associated with the free-body.
2. MSC Nastran can export static loads from a load case defined in the current subcase.
3. MSC Nastran can import a load from a pre-defined database (including, but not limited to, loads produced using the previous steps) to be used in the formation of the load on a structure.

The design of complex structures frequently involves joint development with a system integrator and a number of subcontractors. The development of design loads is typically the task of the system integrator based on an analysis of the entire vehicle. It is then necessary to communicate the loads on the pieces of the structure to various subcontractors. The ability to extract the load on a free body is particularly useful in aeroelasticity, where the load can be a combination of applied, rigid aerodynamic and inertial loadings. The ability to export the statically applied load, or some portion of the load, is felt to be of benefit when it is desired to apply the same loading to different representations of the same structure.

The ability to export a statically applied load is enabled by the Case Control command [EXPORTLD \(Case\)](#) in the *MSC Nastran Quick Reference Guide*. Typically, this is applied at the subcase level, but can be applied above the subcase level as well. The command results in a unique load vector (qualified by LOADID and LOADNAME) for each subcase. If a SET is part of the EXPORTLD command, only the loads on the grids associated with this grid are exported. A BGPDT (Basic Grid Point Data Table) is output with the load vector to identify the degrees of freedom associated with each of the rows in the vector.



The ability to export a free body load is done through the combination of the [FBODYLD \(Case\) Case Control command](#) and the Bulk Data entries [FBODYLD](#) and [FBODYSB](#) in the *MSC Nastran Quick Reference Guide*. The FBODYLD Case Control command is used to point to the FBODYLD Bulk Data (via the *NAMEi* on the case control command) that defines the submodel for which the freebody load will be calculated and stored. The case control command also provides an optional load ID that can be associated with the load. The FBODYLD Bulk Data entry, in turn, points to a FBODYSB Bulk Data entry. The FBODYLD entry provides a label that is intended to identify the loading condition while the FBODYSB entry has a second label that is intended to identify the component. Both labels are optional. The FBODYSB entry identifies the grids and elements that make up the free body and provides the ability to exclude certain types of grid point forces in creating the free body load.

The FBODYLD request produces informational messages from a DBDICT statement that requests output on the presence of all FBLPG and FBLBGPDT data blocks along with qualifiers as to loadid, namei (from case control and the FBODYLD Bulk Data entry), submodel name from the FBODYSB entry and the labels from the two bulk data entries.

The remarks for the EXPORTLD Case Control command indicate how the loads created with an EXPORTLD request can be imported in a subsequent run using FMS statements such as:

```
ASSIGN loads1='run1.MASTER'
DBLOCATE datablk=(EXTLD,EXTBGP) WHERE (LOADNAME='ALLCASES'),
    CONVERT(LOADID=LOADID+1000) LOGICAL=loads1
...
CEND
LOADS=1001 $ Select external load with LOADID=1001, imported
from previous run.
```

The EXTLD datablock contains the loads and the EXTBGP contains the grid point information that is used to match up the imported loads with the grid points. Clearly, it is necessary that the Grid ID's have the same meaning in the two runs.

The import of a load created using the FBODYLD Case Control command is similar except it is now necessary to rename the datablocks created in the previous run to match those required for the import of these loads:

```
ASSIGN loads1='fbrun1.MASTER'
DBLOCATE datablk=(FBLPG/EXTLD,FBLBGPDT/EXTBGP)
WHERE (LOADNAME='fb1case'), CONVERT(LOADID=LOADID+1000)
LOGICAL=loads1
...
CEND
LOADS=1001 $ Select external load with LOADID=1001, imported
from previous run.
```

The applied loads are typically excluded when using the FBODYSB Bulk Data entry so that only element forces remain at the grid points.

Import of loads has the following rules

- If a set of imported loads share a common LOADID value, then those loads will implicitly be added. The same holds true for imported loads and bulk data load sets that share a common ID.



- To explicitly combine load sets, the load IDs should be made unique. The LOAD Bulk Data entry can then be used to explicitly define the linear combination.
- Both standard exported loads (EXPORTLD) and free body loads (FBODYLD) can be imported and used together in a single run. It is up to the user to keep the load IDs unique between them.
- The BGPDT is used to link the exported loads to the grids points of the model to which they are applied. This means that the grid id's for the loaded grids must be the same in the two models.

Fatigue Analysis/Output Request

Introduction

Besides stress, engineers question is how long will it last? Fatigue life calculations can often be a tedious post-processing activity that would take place after the stresses and strains are determined, and done externally to MSC Nastran. Now users can request fatigue life and damage as an output request similar to requesting displacements, stresses, strains, and forces. You can calculate fatigue damage and fatigue life directly within linear statics SOL 101, modal analysis SOL 103, or modal transient SOL 112 runs for materials that can be defined with MAT1 bulk data (metal fatigue analysis). A brief description will be give below. For a detailed description of this capability see [MSC Nastran Embedded Fatigue User's Guide](#).

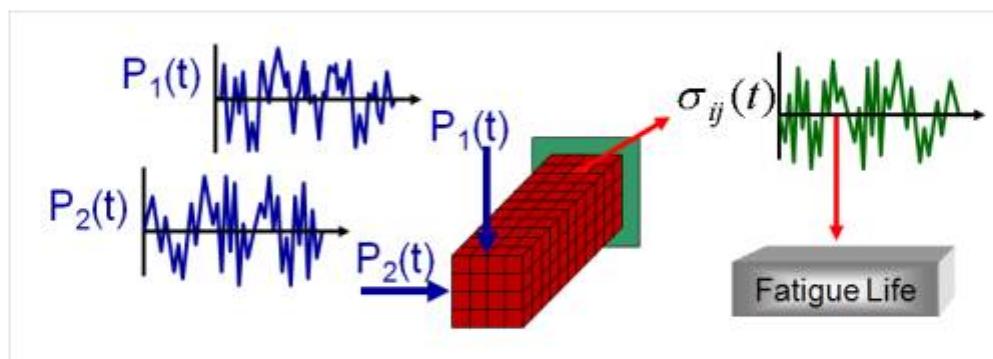
Fatigue Description

Fatigue can be described as structural failure under repeated or otherwise varying load, which never reaches a level sufficient to cause failure in a single application. Typical stress analysis is generally representative of a single application of a loading environment. Multiple or cyclic applications of the same loading environment over time can now be simulated to predict fatigue life and damage directly in MSC Nastran.

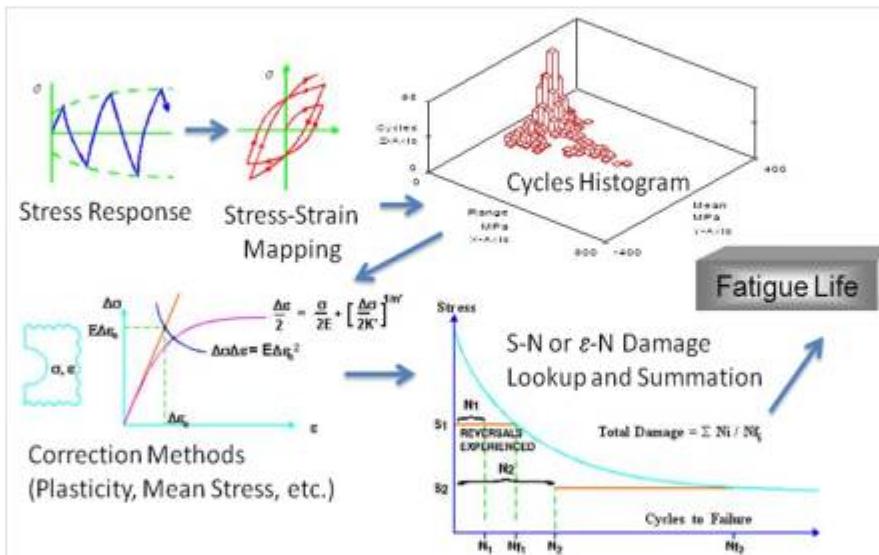
Fatigue analysis requires three main inputs: geometry, materials, and the cyclic load variations. The geometry comes from the SOL 101, 103, or 112 runs in the form of stress distributions over the entire model. Special material properties are used in the form of stress-life (S-N) or strain-life (ϵ -N) curves. The plasticity that occurs due to the cyclic loading is built into these curves and methods used as look up tables and corrections to equate linear stress or strain levels (range and mean) to life. The cyclic variations of the loading are defined in typical table format and are used to scale the stress distribution. Multiple, simultaneously applied loads are combined using the principle of linear superposition to produce the stress or strain time variations. These time histories are then processed through a "rainflow" cycle count algorithm to determine the range and mean of each stress/strain cycle. Damage is determined using the tried and true methods of the total life (S-N) or crack initiation (ϵ -N) to determine fatigue life. Damage from all cycles is summed and reported as life values. Multiple loading events can be strung together to form a sequence of events, commonly known as a duty cycle. Damage from each event is summed to give life due to the entire duty cycle.

The following illustrations gives the overall fatigue calculation process. The first illustrates the process of taking the FE loads and their time variations to combine them into stress output responses time histories at various locations of the model. Ultimately this stress variation is turned into a fatigue live prediction.





The process of converting the time varying stress responses into fatigue life predictions is a two or three step process depending on the method used. Both the stress-life (S-N) and strain-life (ϵ -N) methods employ a well known algorithm to extract cycles of stress/strain called rainflow cycle counting. An easy way to conceptualize this is by mapping the stress time history to the stress-strain space where each hysteresis loop represents a stress-strain cycle. Each cycle has its specific stress range and mean. Sometimes this is illustrated in the form of a histogram showing specific discrete *bins* of stress range vs. mean. In the case of the strain-life (ϵ -N) method, the plasticity correction is then made using techniques such as Neuber's plasticity correction method, before looking up the damage on an S-N or an ϵ -N curve. Both methods may employ mean stress corrections also. Damage from all such *bins* is then summed using Palmgren-Miner damage summation rule and fatigue life presented as the reciprocal of damage.



Overview of Case Control and Bulk Data:

One or more fatigue analyses can be selected using the FATIGUE case control entry. A SET case control entry is used if more than one analysis is to be requested and is then referenced by the FATIGUE case control entry. The [FATIGUE \(Case\)](#) in the *MSC Nastran Quick Reference Guide* case control entry must appear above all subcases. This Case Control examples indicates that three separate fatigue analyses are to be performed.

```
SET 99 = 11, 12, 13
FATIGUE (SET) = 99
```

Each Set ID selected by a FATIGUE case control entry references a set of bulk data that describe the inputs necessary for a fatigue analysis. For each fatigue analysis, a set of FTGDEF, FTGPARM, and FTGSEQ bulk data entries with the same ID need to be defined.

Fatigue Element Definitions (FTGDEF)

The [FTGDEF](#) (FaTiGue element DEFinitions) supplies the analysis with the desired locations on the model where fatigue damage is to be calculated. If no FTGDEF bulk data exists for the fatigue analysis, all the elements (solid and shells) are assumed part of the analysis. Individual elements or entire property sets of elements can be specified. Individual elements can also be excluded from the analysis. The example below simply shows that a fatigue analysis is to be performed on element 1 only and that additional properties are defined using PFTG bulk data with ID 18.

```
FTGDEF, 11, , 18
      , ELSET, 1
```

Fatigue Parameters (FTGPARM)

The [FTGPARM](#) (FaTiGue PARaMeters) defines fatigue parameters. If no FTGPARM bulk data exists for the fatigue analysis, defaults are assumed. The FTGPARM specifically selects which type of fatigue analysis is to be performed such as an S-N (total life or stress-life) or ε-N (crack initiation or strain life) analysis. S-N analysis is the default. Other parameters may also be specified on the FTGPARM to enhance the analysis, speed it up, request different correction methods, and obtain additional output. This example simply shows that a crack initiation analysis is being requested:

```
FTGPARM, 11, EN, 1.0
```

Fatigue Load Sequence (FTGSEQ)

The [FTGSEQ](#) (FaTiGue SEQuence) defines the cyclic load variation. This entry is required and if not present, a fatal error is issued. The FTGSEQ is simple, yet very powerful. It can be used to define a simple oscillating time variation of -1 to +1 scaling of the load, to very complicated sequences of the loading called a duty cycle. For a duty cycle, additional bulk data are necessary: [FTGEVNT](#) (FaTiGue EVeNT) and [FTGLOAD](#) (FaTiGue LOADing). [FTGEVNT](#) is used to define the events of the load sequence and [FTGLOAD](#) is used to define the actual time variations and associate them with a stress distribution from the analysis. This example shows the simplest request where there is only one load variation with time, thus only one event. The FTGSEQ selects FTGEVNT 21, which in turn selects FTGLOAD 101. FTGLOAD 101 references [TABLFTG](#) 201 that describes the time variation and associates it to SUBCASE 8 (in the case of SOL 101):



```
FTGSEQ, 11
      , 21
FTGEVNT, 21, 101
FTGLOAD, 101, 201, 8
TABLFTG, 201
      , 0.0, 1.0, -1.0, 0.0, ENDT
```

Fatigue Materials (MATFTG)

The **MATFTG** (MATerial FaTiGue) defines the fatigue material properties. It must be associated to an existing MAT1 entry with the same ID. In the example that has been used thus far, a crack initiation analysis has been selected by the FTGPARM entry. This means that e-N material data is necessary. This data can be derived by simply supplying the ultimate tensile strength and a material code defining the type of metal as shown or actual ϵ -N parameters can be entered as described in the user documentation.

```
MAT1, 1, 203403.0, 78231.7, 0.3, 1.0
MATFTG, 1
      , STATIC,     , 480.0,     99
```

Fatigue Properties (PFTG)

The **PFTG** (Property FaTiGue) defines other possible fatigue properties such as surface finish and other factors that can be applied to the requested entities from the FTGDEF entry. If this entry is not present and none is selected by FTGDEF, then default values are used. This example shows PFTG with ID 18 elected from the previous example of the FTGDEF entry where a polished surface finish is specified.

```
PFTG, 18, , POLISH
```

Examples

Examples of using this capability are shown in Chapters 3 through 9 of the [MSC Nastran Embedded Fatigue User's Guide](#).



STATSUB and BENDL Support in SOL101 Linear Statics

Introduction

Support for preloaded static analysis is now extended to include SOL 101 Linear Statics solution. The preload used to form the differential stiffness is selected using the STATSUB Case Control command which points to a previous SUBCASE. Examples models are available in `../tpl/extse400/ss101*` subdirectory of MSC Nastran Documentation Installation directory.

Bar END Load

A new case control function BENDL is added to calculate bar end loads in panel – stringer configurations.

BENDL

BAR/BEAM end load Subcase Selection

Function to calculate bar end loads and shear flow.

Format:

BENDL(PRINT = [YES,NO]) = [ALL, n, NONE]

Examples:

To output a table of bar end loads/shear flows.

`BENDL = ALL`

To output a table of bar end loads/shear flows and print table to F06.

`BENDL (PRINT) = ALL`

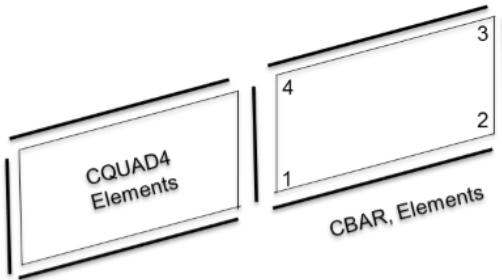
Descriptor	Meaning
PRINT	Prints bar end loads and panel shear flow to the F06 file.
YES	BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will be printed.
NO	BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will not be printed. (Default)
ALL	Select all bar/beam/rod type elements associated with "Shell" panels.
n	Identification of a previously defined SET listing "Flange/Boom" elements.
NONE	Turn off BENDL in selected subcases.

Remarks:

1. A BENDL Case Control Command is to appear in any subcase for which Bar End Loads and Panel shear flows are required. This entry is only applicable in SOL101 or ANALYSIS=STATIC.



2. The presence of BENDL in a linear statics run will automatically turn on GPFORCE=ALL as well as PARAM, NOELOF, 1.
3. Only CSHEAR, CQUAD4, CTRIA3, CQUADR, and CTRIAR panels will contribute to shear flow and bar end load calculations.
4. Include MDLPRM HDF5 to output OUTSFLOW data block to the HDF5 output file.
5. The following figure shows a typical bar end load and shear flow panel.



6. Print and table FORMAT is:

```

B A R   E N D   L O A D S   A P P R O X I M A T I O N S   ( C B A R )
ELEMENT          TOTAL END LOADS
      ID.        END 1        END 2
       6      -1.994413E+03 -1.136868E-13
      11      2.082592E+03  3.638604E-01
      16     -4.721992E+02 -7.329887E+01
      21      4.984904E+02 -4.987337E+02

B A R   E N D   L O A D S   S H E A R   F L O W   A P P R O X I M A T I O N S ( C
Q U A D 4 )
ELEMENT ID.          SHEAR 12          SHEAR 23          SHEAR 34          SHEAR 41
      1      9.972063E+01 -2.493060E+02 -9.972063E+01  3.989003E+01

```





8

R-Type Constraint Elements

- Methods for Rigid Element Solution
- Introduction to R-Type Elements
- Description of the R-Type Elements
- The RBAR Element
- The RBE2 Element
- The RBE3 Element
- The RJOINT Element



Methods for Rigid Element Solution

Linear Method

In MSC Nastran, two methods for rigid elements are available -- the linear method and the Lagrange method. The linear method is the default method in MSC Nastran. The linear method uses the linear elimination technique. The rigid elements are not real elements, they are internally represented by a set of MPC equations. By using these MPC equations, the dependent degrees-of-freedom (the m-set) are eliminated from the solution set. The rigid elements linear method has the following limitations:

- Do not compute thermal load.
- Do not have a differential stiffness matrix; therefore, the solutions are incorrect for the buckling analysis or other solution sequences where the differential stiffness matrix is required.
- Use the small rotation theory in the geometrical nonlinear analysis so that the solutions are incorrect in this type of analysis.
- Use the elimination method for solution, resulting in very dense stiffness matrices. These dense matrices cannot take advantages of the sparse matrix algorithm.

Lagrange Method

With the Lagrange method, the rigid elements become “real” finite elements, similar to, for example, a QUAD4 element. Instead of using MPC equations, the element stiffness matrix is computed for each rigid element. All the limitations for the linear method are removed, i.e., the rigid elements

- Include thermal load effect.
- Include the differential stiffness.
- Support for both the small rotation and the large rotation. Use the large rotation theory in the geometrical nonlinear analysis (PARAM,LGDISP,1). Not available for SOL 101, use SOL 400 if large displacements or large rotations are present.
- Take advantages of the sparse matrix algorithm if the augmented Lagrange multiplier method, defined below, is used.

For each Lagrange rigid element, a number of Lagrange multiplier degrees-of-freedom are created internally by MSC Nastran. For example, one to six Lagrange multiplier degrees-of-freedom are created for the RBAR and one Lagrange multiplier degree-of-freedom is created for the RROD. For each Lagrange rigid element, the independent degrees-of freedom, the dependent degrees-of-freedom, and the Lagrange multiplier degrees-of-freedom are left in the solution set (the l-set).

For the Lagrange Multiplier Method, the solution is obtained with the independent degrees-of-freedom, the dependent degrees-of-freedom, and the Lagrange multipliers degrees-of-freedom left in the solution set. Thus, the sparse characteristic of the stiffness matrix is maintained and sparse matrix algorithms can be used.

See [Rigid Element Enhancements](#) in the *MSC Nastran Reference Guide* for a theoretical discussion of these methods and the use of Rigid elements for large rotations.



Method Selection

The Control Case command, [RIGID \(Case\)](#) in the *MSC Nastran Quick Reference Guide* selects the type of rigid element. It has the following format:

$$\text{RIGID} = \left\{ \begin{array}{l} \text{LINEAR} \\ \text{LAGR} \\ \text{LGELIM} \end{array} \right\}$$

LINEAR will select the linear rigid elements, LAGR will select the Lagrange rigid element with the Lagrange multiplier method, and LGELIM will select the Lagrange element with the Lagrange elimination method (not used in Statics). If the RIGID command does not exist in the user Case Control file, the linear rigid element will be used.

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. They could also be called constraint elements. Thus, an R-type element is mathematically equivalent to one or more multipoint constraint equations (discussed in [Constraints](#)). Each constraint equation expresses one dependent degree of freedom as a linear function of the independent degrees of freedom.

The R-type elements are sometimes referred to as rigid elements; however, the name “rigid” is misleading. The R-type elements that are rigid consist of the RROD, RBAR, RBAR1, RJOINT, RBE1, RBE2, RTRPLT and RTRPLT1. 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; and a brief discussion of the RJOINT element.

The discussions below assume a Linear method selection (the default). There are some differences in the Bulk Data entries when changing from Linear to Lagrange. They are discussed in [Lagrange Method Bulk Data Entry Differences](#).

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. 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](#).



Seven R-type elements are described, 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 Elements Available in MSC Nastran

Name	Description	$m = \text{No. of Equations of Constraint Generated}$
RROD	Pin-ended rod which is rigid in extension.	$m = 1$
RBAR	Rigid bar with six degrees of freedom at each end.	$1 \leq m \leq 6$
RBAR1	Rigid bar with six degrees of freedom at each end.	$1 \leq m \leq 6$
RJOINT	Rigid joint (coincident points) 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.	$1 \leq m \leq 12$
RTRPLT1	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.	$m \geq 1$
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.	$m \geq 1$

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](#).



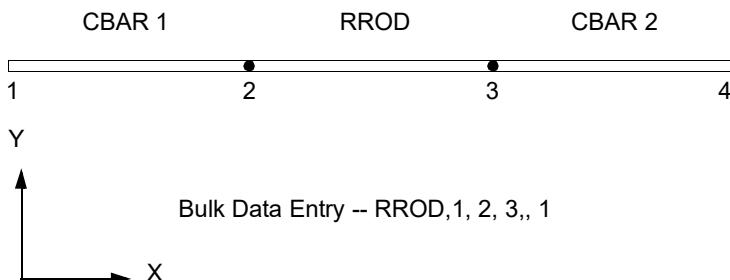


Figure 8-1 An RROD Connection

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 Bulk Data entry **RROD** 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 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.

Five of the R-type elements (RBAR, RJOINT, 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](#) 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 or RJOINT



Table 8-2 Typical Application for Rigid Element

Application	R-Type Entries
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 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 use of PARAM,CHECKOUT,YES, see [CHECKOUT](#) in the *MSC Nastran Quick Reference Guide*, is a useful way to check rigid elements and MPC's. See Chapter 7.4.1 Constraint Modules of the *MSC Nastran Reference Guide* for a detailed description of the tests performed.

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 Bulk Data entry, [RBAR](#) in the *MSC Nastran Quick Reference Guide* is as follows:

RBAR

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification number of connection points. (Integer > 0)
CNA, CNB	Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark 3. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
CMA, CMB	Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 4. and 5. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real ≥ 0.0 or blank)

The format of the Bulk Data entry, [RBAR1](#) is a follows:



RBAR1

1	2	3	4	5	6	7	8	9	10
RBAR1	EID	GA	GB	CB	ALPHA				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB, which are constrained to move as the rigid bar. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark 8. (Real ≥ 0.0 or blank)

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, i.e. the RBAR1 format. (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.

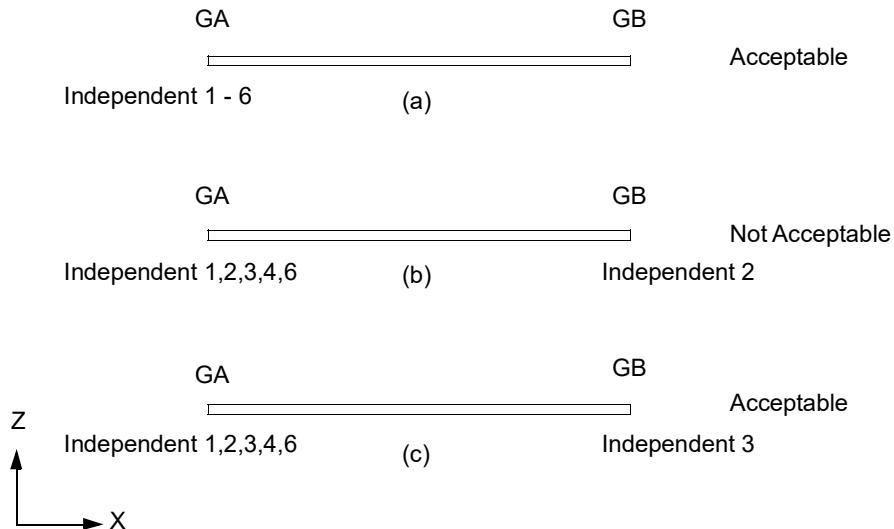


Figure 8-2 Defining Independent DOFs on the RBAR



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.

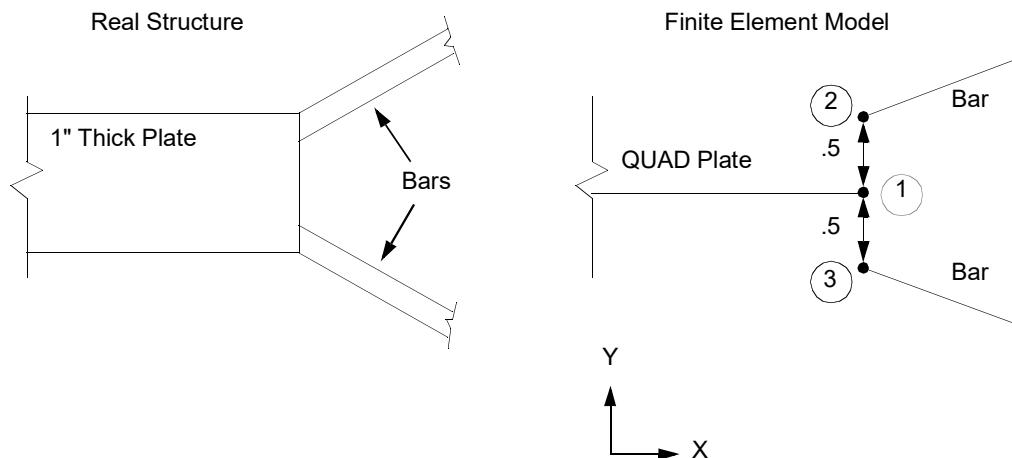


Figure 8-3 Model of a Thick Plate with Bars Attached

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 secondary 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,

$$u_{1_2} = u_{1_1} - .5 \cdot u_{6_1} \quad (u_{1_2} = \text{displacement in the 1-direction at grid point 2})$$

$$u_{1_3} = u_{1_1} + .5 \cdot u_{6_1}$$

$$u_{6_3} = u_{6_1}$$

$$u_{6_2} = u_{6_1}$$

$$u_{2_2} = u_{2_1}$$

$$u_{2_3} = u_{2_1}$$

The MPC entries for this model are as follows:

MPC



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.

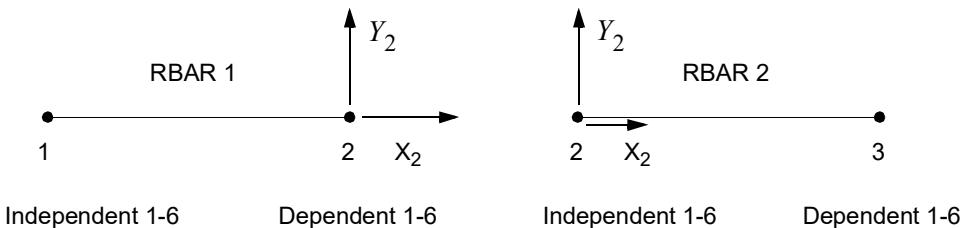
A particular area of confusion for the new user is when you need to connect R-type elements together. The important thing to remember is that you can place a degree of freedom into the *m*-set only once. Consider the two RBAR elements shown in [Figure 8-4](#) that are acting as a single rigid member.

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 RBAR 1, 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](#). (See `MSC_DOC_DIR/doc/linstat/rbar1.dat`) For clarity, CBAR 3, 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  
$
```



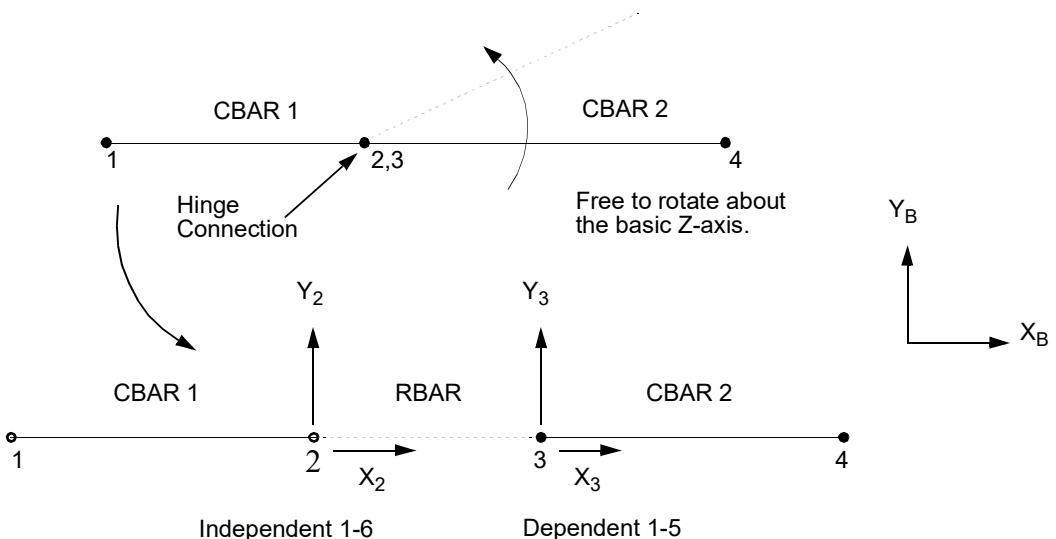
```

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 listing of the input file for this model is shown in [Listing 8-2](#). (See MSC_DOC_DIR/doc/linstat/rbar2.dat) 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.

The RBE2 Element

The RBE2 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 Bulk Data entry [RBE2](#) is as follows:

RBE2

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

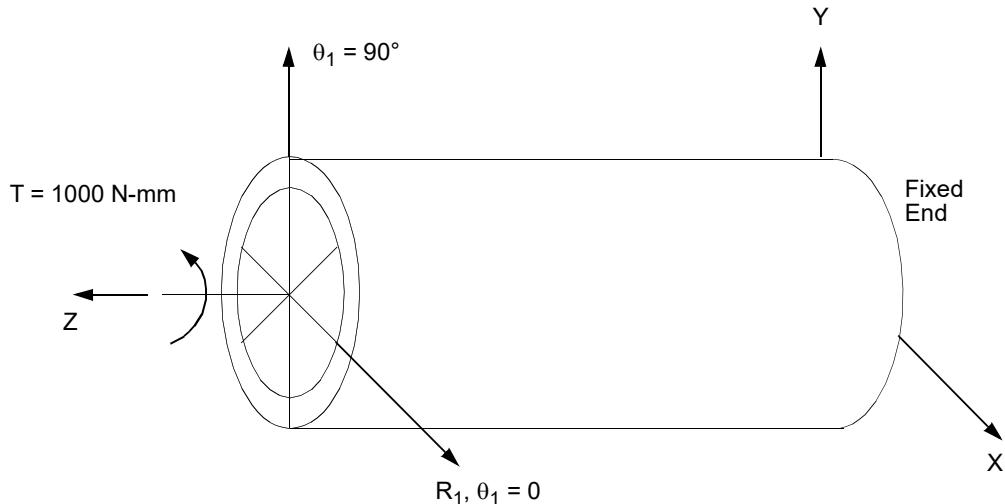
Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.)



Field	Contents
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real ≥ 0.0 or blank)

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
 $E = 25 \times 10^4 \text{ N/mm}^2$
 $v = 0.3$

Figure 8-6 Tube with an End Torque Loading

A partial input file for this model is shown in Listing 8-3. See /doc/linstat/torque.dat) 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          -10.6066 10.6066 0.0  

GRID    114          -15.     7.105-150.0  

GRID    115          -10.6066 10.6066 30.  

GRID    116          -15.     1.066-1430.  

GRID    117          -10.6066 10.6066 60.  

GRID    118          -15.     1.066-1460.  

GRID    119          -10.6066 10.6066 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
$ 
$ 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



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 Bulk Data entry **RBE3** is as follows:

RBE3

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.-			
	"ALPHA"	ALPHA							

Field	Contents
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)
Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
"UM"	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
"ALPHA"	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark 14. (Real ≥ 0.0 or blank)



The manner in which the forces are distributed is analogous to 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.

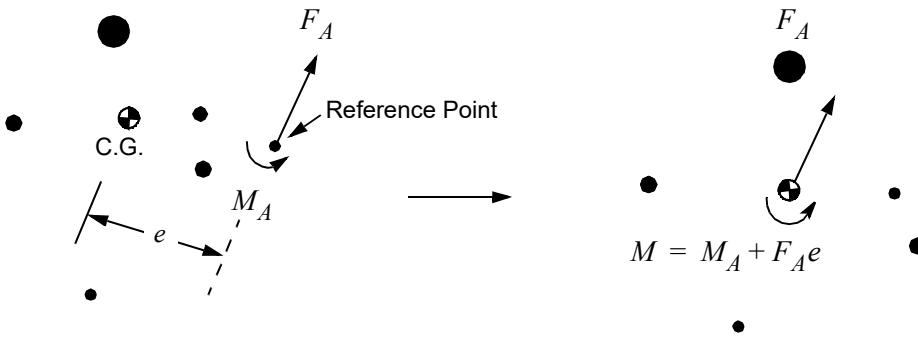
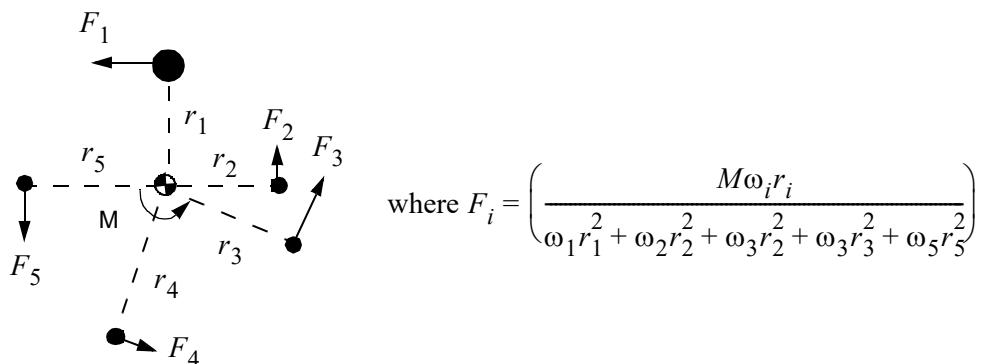
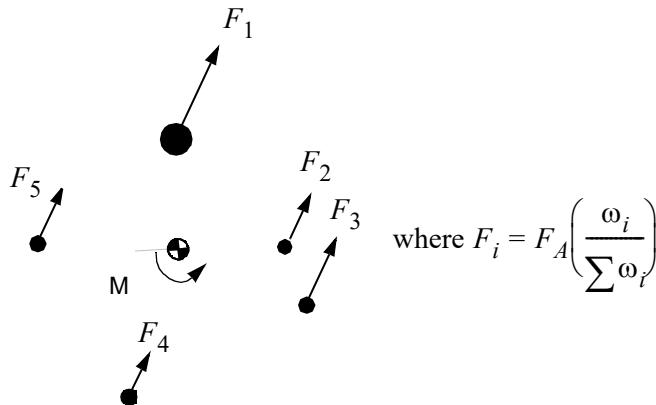


Figure 8-8 RBE3 Equivalent Force and Moment at the Reference Point

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 8-9](#). 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.



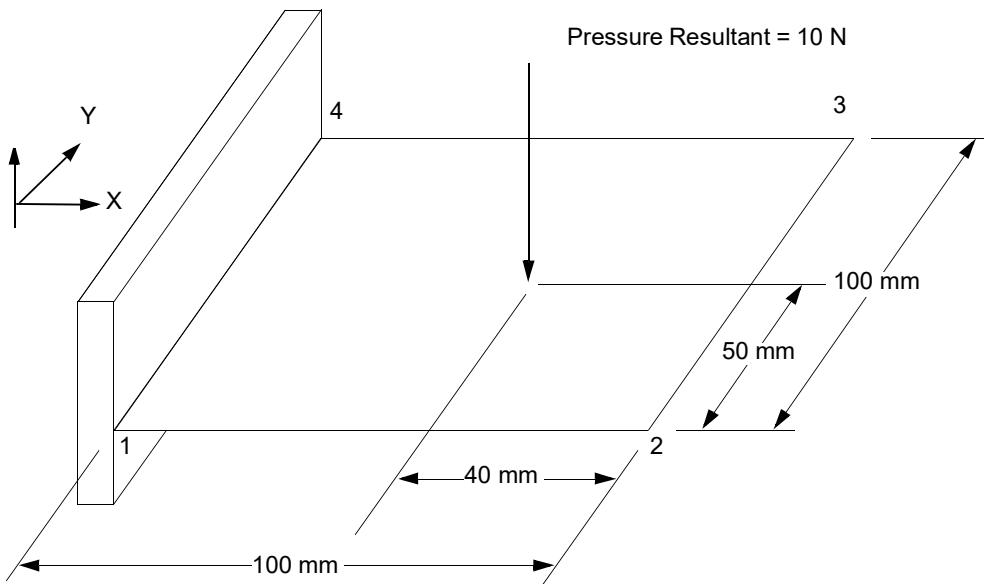


Figure 8-10 Using an RBE3 to Represent a Nonuniform Pressure Load

The input file representing this example is shown in [Listing 8-4](#). (See MSC_DOC_DIR/doc/linstat/rbe3.dat) Grid point 99 is called the REFCGRID 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 Ci 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 Ci 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](#).



DISPLACEMENT VECTOR									
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		
LOAD VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
99	G	.0	.0	1.000000E+02	.0	.0	.0		
FORCES OF SINGLE-POINT CONSTRAINT									
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		
GRID POINT FORCE BALANCE									
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.



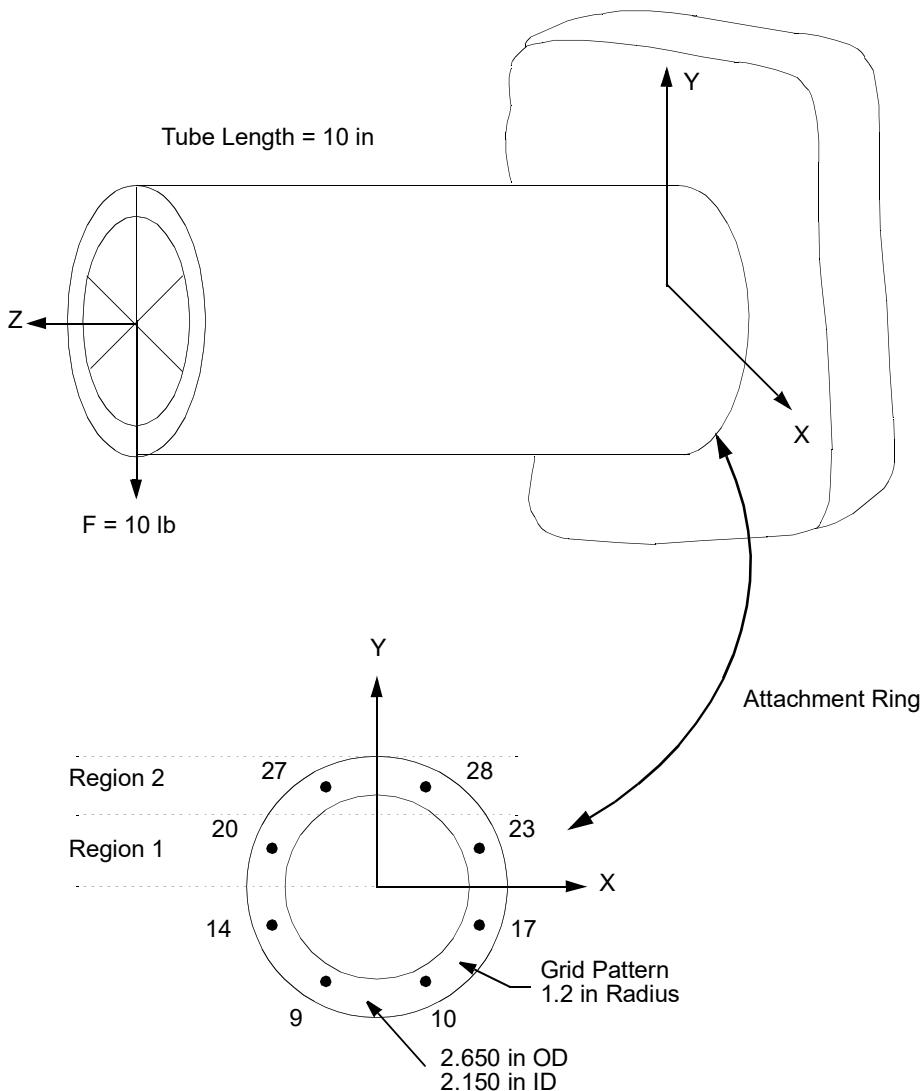


Figure 8-12 Attachment Ring

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.

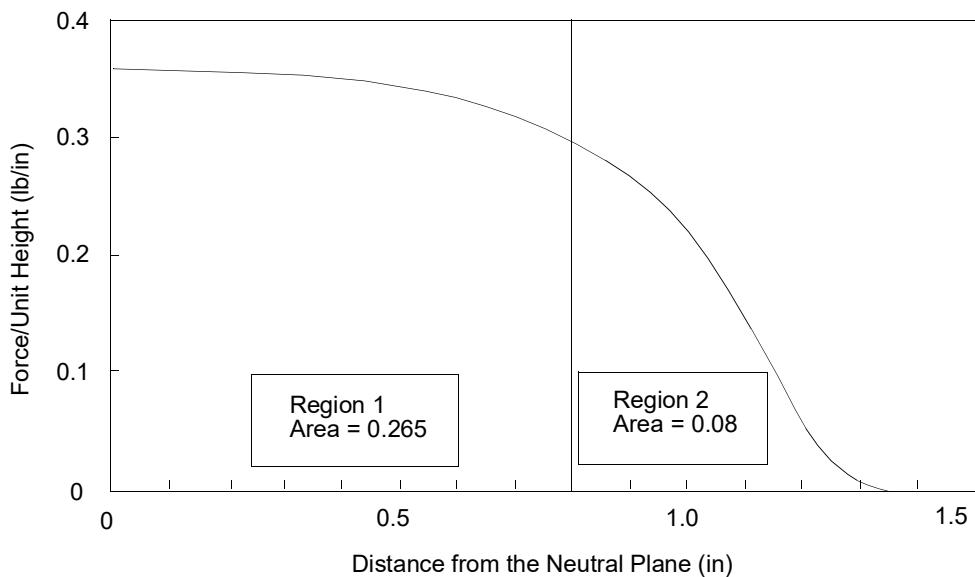


Figure 8-13 Shear Force on the Attachment Ring

A partial input file for this example is shown in [Listing 8-5](#). (See MSC_DOC_DIR/doc/linstat/shear1.dat) 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  
$
```



```

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
$ 
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

```

The most common user error in RBE3 element specification results from placing 4, 5, or 6 in the Ci (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 Ci 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 Ci field only for special cases, such as when the independent points are colinear.

The RJOINT Element

The RJOINT element defines a mechanical joint. The format for Bulk Data entry, [RJOINT](#) is as follows:

RJOINT

1	2	3	4	5	6	7	8	9	10
RJOINT	EID	GA	GB	CB					

Field	Contents
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. (Integers 1 through 6 with no embedded or blank.)



GA is the independent grid point and all six degrees-of-freedom are independent. GB is the dependent grid point. The length between points GA and GB must be zero. Because its length is zero, the thermal load effect is not applicable.

If CB=123456 or is left blank, then the grid point GB is constrained to move with the grid point GA, and two the grid points move as a single point. If any degree-of-freedom is released on CB, then RJOINT becomes a mechanical joint. A mechanical joint is a mechanical system that has two bodies jointed at a point. The two bodies can rotate relatively about one, two, or three axes of a local coordinate system at that point. For example:

- Hinge - A hinge is a mechanical joint that rotates freely about one axis about the local coordinate system. It can be simulated by RJOINT with one rotational degree of freedom released, i.e., CB = 12356, 12346, or 12345.
- Universal joint - A universal joint is a mechanical joint that rotates freely in two axes. It can be simulated by RJOINT with two rotational degrees-of-freedom released, i.e., CB = 1234, 1235, or 1236.
- Spherical joint - A spherical joint is a mechanical joint that rotates freely about all three axes. It can be simulated by a zero length RJOINT with all rotational degrees of freedom released, i.e. CB = 123.
- Prismatic joint - A prismatic joint is a mechanical system with two blocks that are constrained to have the same rotations, but translate relative with each other along a local axis. It can be simulated by RJOINT with one translational degree-of-freedom released, i.e., CB = 23456, 13456, 12456.
- Cylindrical joint - A cylindrical joint is a mechanical system that allows two grid points to have relative translation along a moving axis and, at the same, have relative rotation about the same axis. It can be simulated by RJOINT with one translational degree of freedom and one rotational degree of freedom released, i.e. CB = 2356, 1346, 1245.

Guidelines for the RJOINT element:

- The theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB. If different global coordinate systems are used, the degrees-of-freedom to be released are determined by the coordinate system at GB.
- For linear or nonlinear static analysis, the joints must be constrained by other elements in the structural model. Otherwise, a singular stiffness matrix may be produced, leading to failure during decomposition.

Lagrange Method Bulk Data Entry Differences

The R-type elements; RBAR, RBAR1, RJOINT, RBE1, RBE2, RBE3, RROD, RTRPLT and RTRPLT1; can be used as either the linear rigid element or the Lagrange element, as selected by the Case Control command, RIGID. However, the input rules are not the same for these two types of rigid elements. The following paragraphs discuss the differences.

Except for the RBE3 and RROD, the major difference in the input format between the linear rigid element and the Lagrange rigid element is the selection of the independent degrees-of-freedom. This can be illustrated by the Bulk Data entry for RBAR:

RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	
------	-----	----	----	-----	-----	-----	-----	-------	--



The independent degrees-of-freedom are selected by CNA and CNB. For the linear rigid element, the independent degrees-of-freedom can be assigned to both CNA and CNB: for example, “CNA=1236, CNB=34”, as long as the total number equals to six and they can jointly represent any general rigid body motion. However, for the Lagrange rigid element, all six independent degrees-of-freedom must be assigned to a single grid point, i.e., “CNA=123456, CNB=blank”, or “CNA=blank, CNB=123456”. The same rule applies to the RBE1, RBE2, and RTRPLT elements. The RBAR1 and RTRPLT1 entries make it easier to input the Lagrange rigid elements.

For the RBE3, the REFC degrees-of-freedom can be any combination of integers from 1 through 6 for the linear rigid element. For the Lagrange rigid element, RFEC must be 123, 456, or 123456.

For the RROD element, the user must select one dependent degree-of-freedom by inputting either CMA or CMB for the linear rigid element. However, for the Lagrange rigid element, the user can leave both fields blank and let MSC Nastran select the best component as the dependent degree-of-freedom. In fact, this is the recommended method.

For all rigid elements, the ALPHA field is the thermal coefficient of expansion. For the Lagrange rigid elements, if ALPHA is given and the thermal loads are requested by the Case Control command TEMPERATURE(INITIAL) and TEMPERATURE(LOAD), the thermal load effect will be computed for the rigid elements. The temperature loads are taken as the average temperature given by the independent grid point and the dependent grid point. For example, the temperature load for the RBAR element is taken as the average temperature of grid points GA and GB. For the linear rigid element, no temperature effect is computed and the ALPHA field is ignored.

Example - Linear Static Analysis with Thermal Loads

To illustrate the effect of thermal loads on the Lagrange rigid elements in a static analysis, a simple beam is modeled by a RBAR1 element with a length of 1.0. At left end of the beam, a very soft rotational spring is connected to the normal rotation and the remaining five degrees-of freedom are fixed. At the right end, all six degrees-of-freedom are free with a vertical force of 100.0 is acting on it. This beam is subjected to a thermal load, which will double its length to 2.0. This problem is solved by the Lagrange multiplier method (RIGID=LAGR). (See MSC_DOC_DIR/doc/linstat/nlrgd11a.dat)

The input file for this problem is shown below.

```
ID RIGID,NLRGD11A
SOL 101
CEND
TITLE = LINEAR STATIC ANALYSIS - THERMAL LOAD ON RBAR
SUBTIT= LAGRANGE MULTIPLIER METHOD
RIGID = LAGR
TEMP (INIT) = 10
SUBCASE 1
    TEMP (LOAD) = 20
    LOAD = 100
    SPC = 10
    DISP = ALL
    SPCF = ALL
    FORC = ALL
BEGIN BULK
FORCE,100,2,0,500.0,0.0,1.0,0.0
```



```
CELAS2,101,1000.0,1,6
GRID,1,,1.0,0.0,0.0
GRID,2,,2.0,0.0,0.0
RBAR1,3,1,2,123456,2.0-2
TEMP,10,1,0.0
TEMP,10,2,0.0
TEMP,20,1,50.0
TEMP,20,2,50.0
SPC1,10,12345,1
ENDDATA
```

Results for nlrgdlla.dat:

Description	Value
Vertical displacement	0.5
Horizontal displacement	1.0
Angle of rotation	28.6°
Initial beam length	1.0
Final beam length	2.06

These results show that horizontal thermal expansion is correctly computed as 1.0. However, due to the soft rotational spring on the left end of beam, this problem is a large displacement problem. Therefore, the vertical and horizontal displacements, angle of rotation, and the final beam length are incorrect.



9

Modeling Guidelines

- Introduction to Modeling Guidelines
- Choosing the Right Element
- Mesh Density
- Mesh Transitions
- Grid Point Stresses
- Consistent Loading
- Symmetry



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](#).

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.

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.

CBEAM3 elements favor a structure with initial curvatures and with high order shell elements.

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 (CQUADi, CTRIAi) 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.

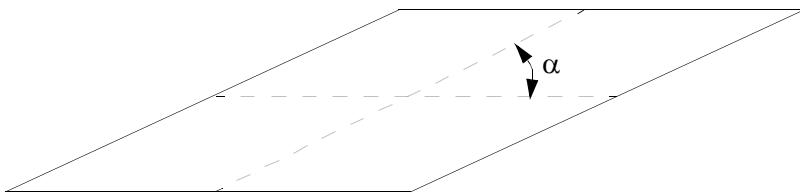


Figure 9-1 Highly Skewed Element

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.

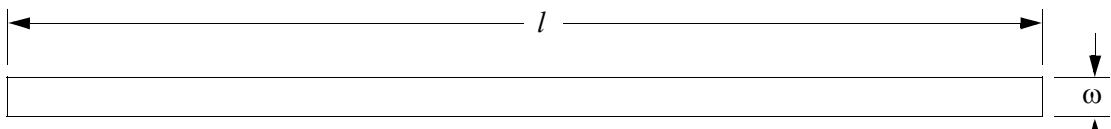


Figure 9-2 Element with High Aspect Ratio

Warping is a measure of the amount the element deviates from being planar (see [Figure 9-3](#)). Element warping should be minimized.

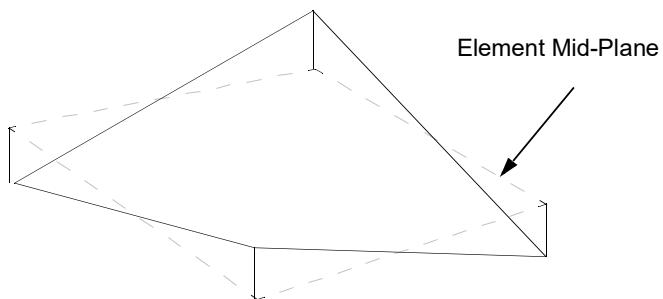


Figure 9-3 Highly Warped Element

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.



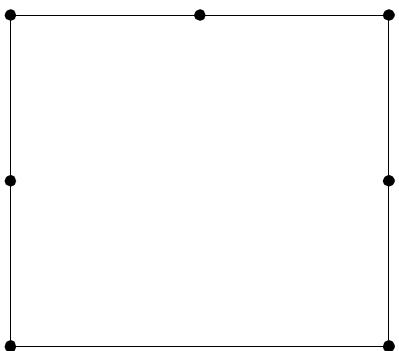


Figure 9-4

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 13.).

The shell normal (param,snorm,x) should be turned on when using the CQUAD4, CTRIA3, CQUADR, or CTRIAR element. See [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 CQUADR and CTRIAR elements can be used in conjunction with each other, but not with other elements. CQUADR and CTRIAR should not be used for nonlinear analysis.

An increase in accuracy can be expected when using the QUADR element vs the QUAD4 element, particularly in areas of the model that are more coarsely meshed. However the QUADR element has other restrictions which make the QUAD4 element the generally recommended element.

Existing QUAD4 element models can easily be converted to QUADR elements by setting System Cell ([QRMETH \(370\)](#)) in the NASTRAN Statement. QRMETH=5 will convert all QUAD4/TRIA3 elements in the model to QUADR/TRIAR. To show the difference in accuracy between the QUADR and QUAD4 elements, four mesh element sizes for a simple “T-Section” test model were run using MSC Nastran. von Mises stress results, taken at a central position in the “T-Section”, were compared and they showed the QUADR was more consistent and accurate as the size was reduced. See [Figure 9-5](#).



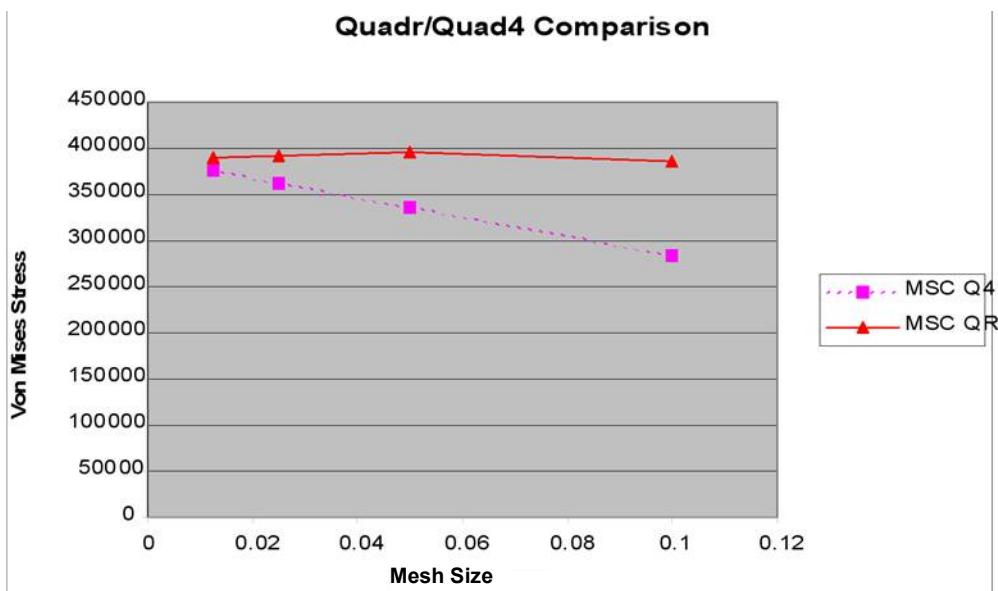


Figure 9-5

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—the default value of 100 on 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. This formulation of using a common unique normal, 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 9-6](#).) The default for β can be changed by setting PARAM,SNORM, β 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.

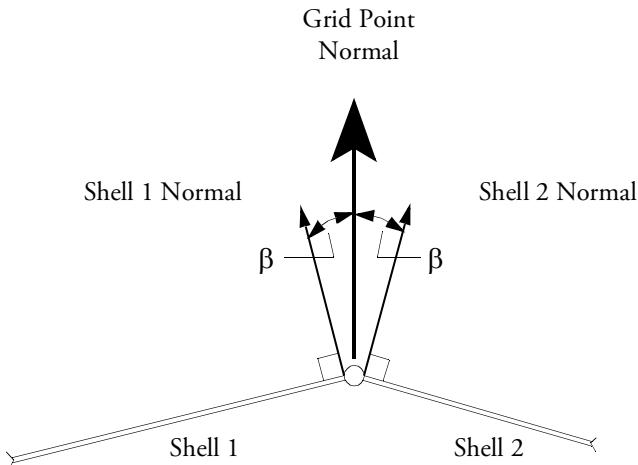


Figure 9-6 Unique Grid Point Normal

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 and V1, V2, and V3 are unscaled components of the vector in the coordinate system.

A second parameter, [Unique Grid Point Normal for Adjacent Shell Elements](#), 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 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,100." 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. The weld family of elements offer excellent ways of modeling structural connections.

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 9-7](#) 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.

In a study of mesh densities, elements and output options, three different mesh densities were used in the example as shown in [Figure 9-7](#). 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_0 (see Timoshenko and Goodier, *Theory of Elasticity*, Reference 3.) is given by the following equation:

$$\sigma_0 = \frac{p_i a^2 \left(1 + \frac{b^2}{r^2}\right)}{(b^2 - a^2)}$$

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 $p_i/r/a$ versus r/a . The results are summarized in [Table 9-1](#).



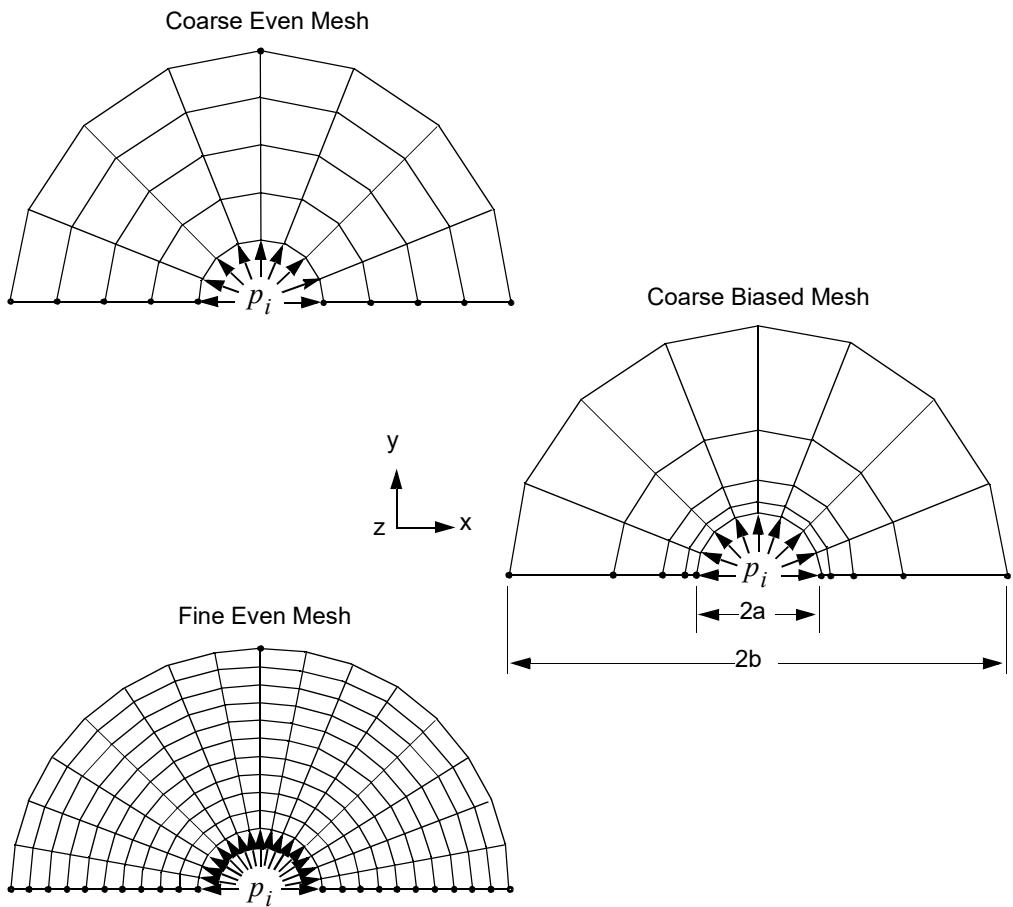


Figure 9-7 Circular Disk with Different Meshes

Table 9-1 Stresses Close to $r = a$ for a Circular Disk

Case No.	Element Type	DOF (L-Set)	Description	Theoretical Stress
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



Table 9-1 Stresses Close to $r = a$ for a Circular Disk

Case No.	Element Type	DOF (L-Set)	Description	Stress Theoretical
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. Looking at [Figure 9-7](#), 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 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.

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](#), may be used for accessing the quality of the mesh density for the conventional h-version elements.

Getting a good starting mesh density can be very useful, but there are mesh refinements which can be performed “automatically” inside MSC Nastran. One is the use of [Local Adaptive Mesh Refinement](#) for h-elements.

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.

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:

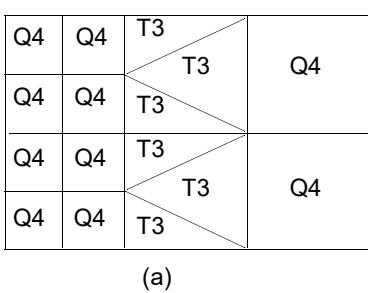


- Never place a mesh transition in an area of interest or in an area where there is a large variation in stress.
- Mesh transitions should be located away from the areas of interest in a region.

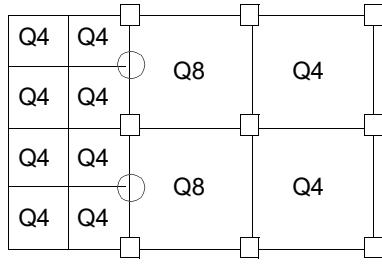
Due to incompatibilities between finite element types, any transition between different element types (even a transition from CQUAD4 to CTRIA3 elements) can result in local stress anomalies. Normally, these stress anomalies are localized and dissipate quickly as you move away from the transition. However, 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.



(a)



(b)

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.

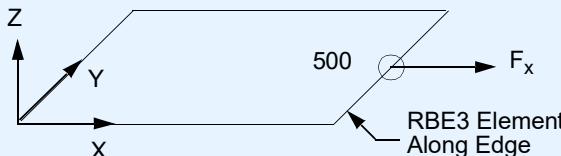
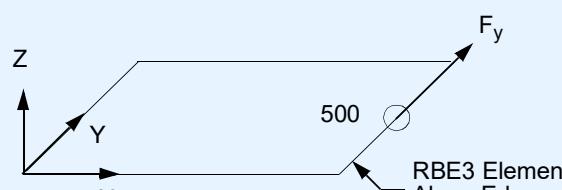
As an alternative mesh transition method, a spline element can be used. In MSC Nastran, the RSPLINE element is an elastic interpolation element that connects N end points. Displacements for intermediate grid points are interpolated from the equations of an elastic beam passing through these grid points. 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

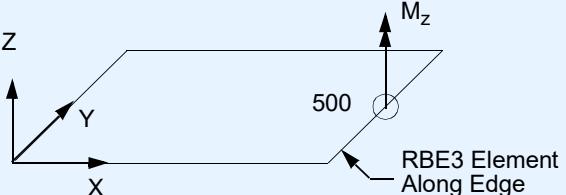
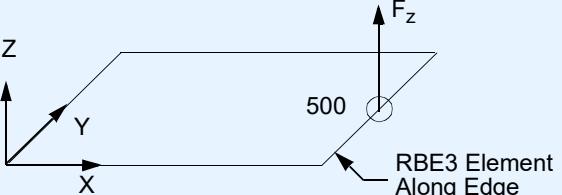
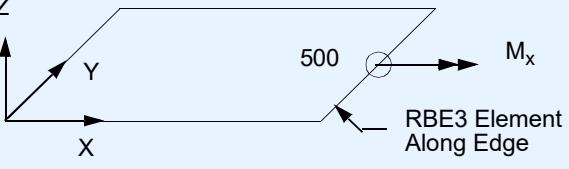


of interest are labeled. Five different loadings-as shown in [Table 9-2](#) are applied to this structure along the free edge (right end). For convenience, a RBE3 element is created along the free edge by defining a dependent point (grid point 500) at the center. This grid point is connected to all the 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](#) 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.

Loading Description	
1. Membrane Loads	
2. In-Plane Shear Loads (parabolic distribution)	



Loading Description	
3. In-Plane Bending Loads	 <p>A diagram showing a horizontal beam segment of length 500. A coordinate system (X, Y, Z) is established at the left end. A clockwise moment M_z is applied at the right end. An 'RBE3 Element Along Edge' is indicated at the right end.</p>
4. Vertical Loads	 <p>A diagram showing a horizontal beam segment of length 500. A coordinate system (X, Y, Z) is established at the left end. A vertical force F_z is applied at the right end. An 'RBE3 Element Along Edge' is indicated at the right end.</p>
5. Twisting Loads	 <p>A diagram showing a horizontal beam segment of length 500. A coordinate system (X, Y, Z) is established at the left end. A clockwise moment M_x is applied at the right end. An 'RBE3 Element Along Edge' is indicated at the right end.</p>



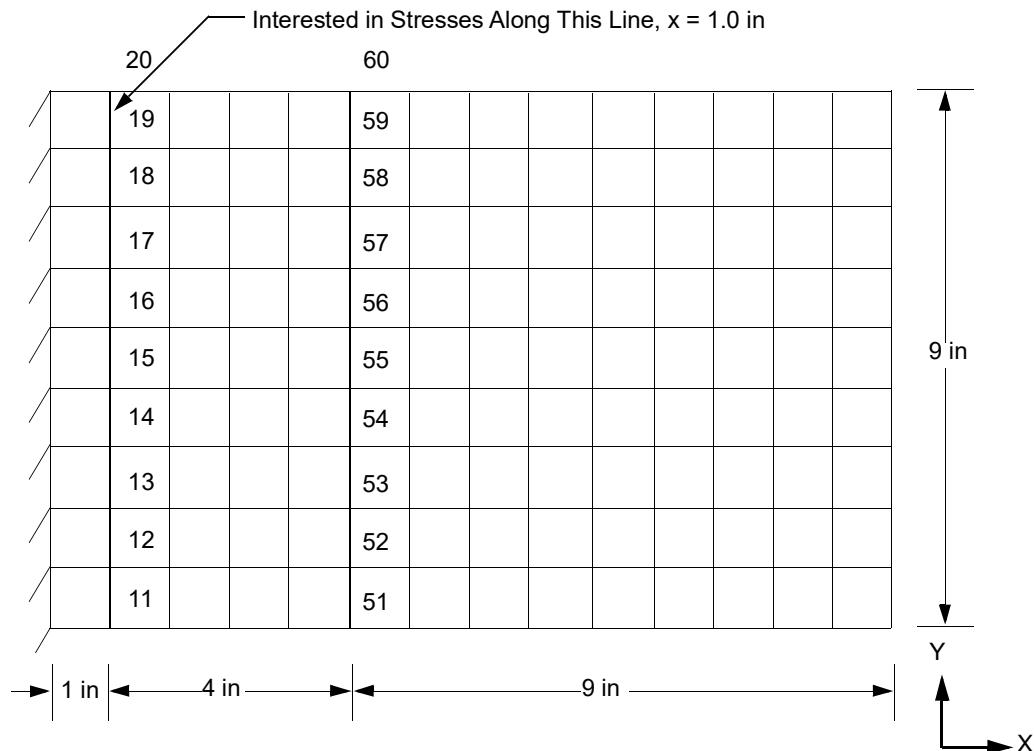


Figure 9-9 Baseline Model



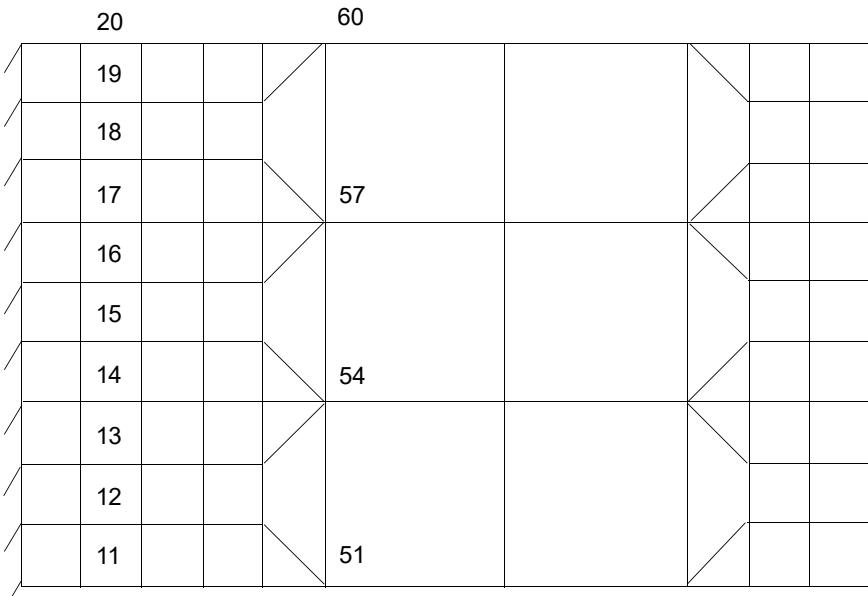


Figure 9-10 Transition with CTRIA3

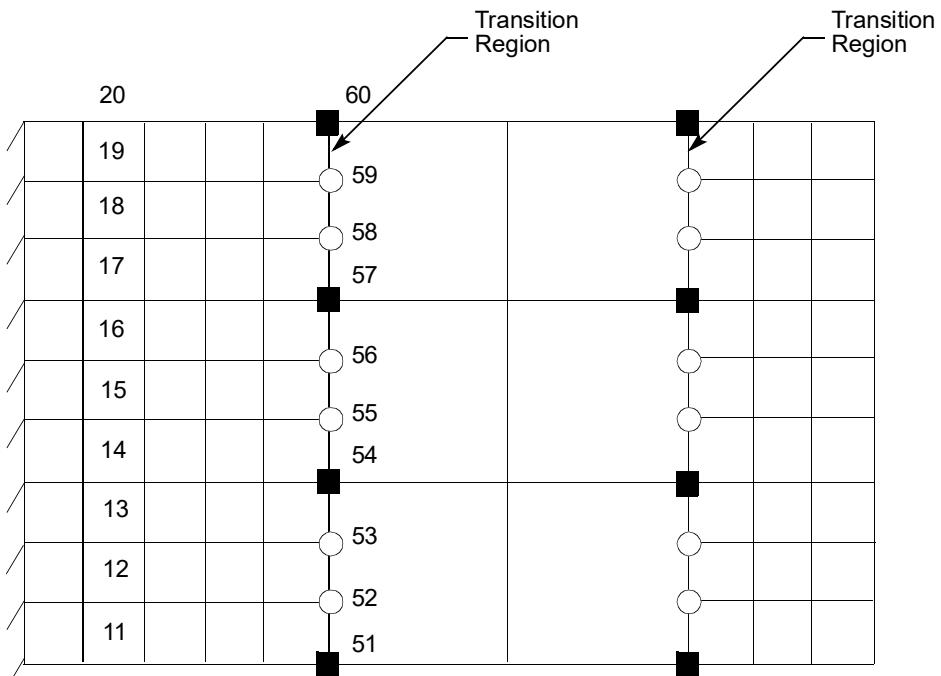


Figure 9-11 Transition with RSPLINE



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:

RSPLINE

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](#)) and the CQUAD4 corner stress option are used for comparison. The results are shown in [Figure 9-11](#) through [Figure 9-29](#). 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 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.

Mesh transition can also be made with CINTC elements for connection meshes on dissimilar edges. Contact may also be used.



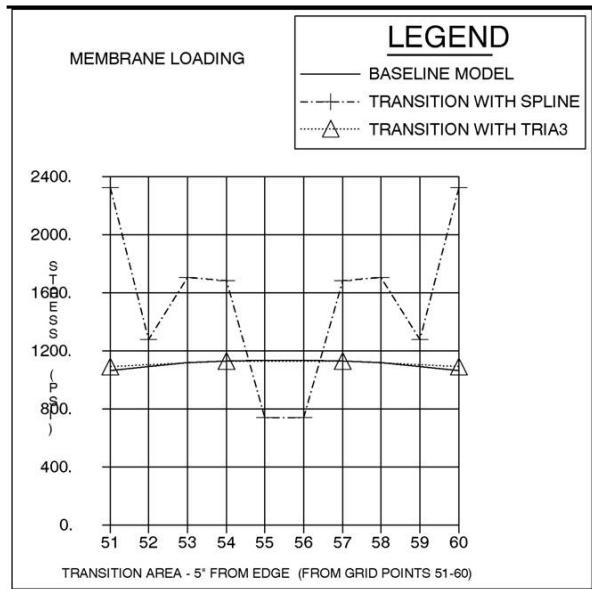
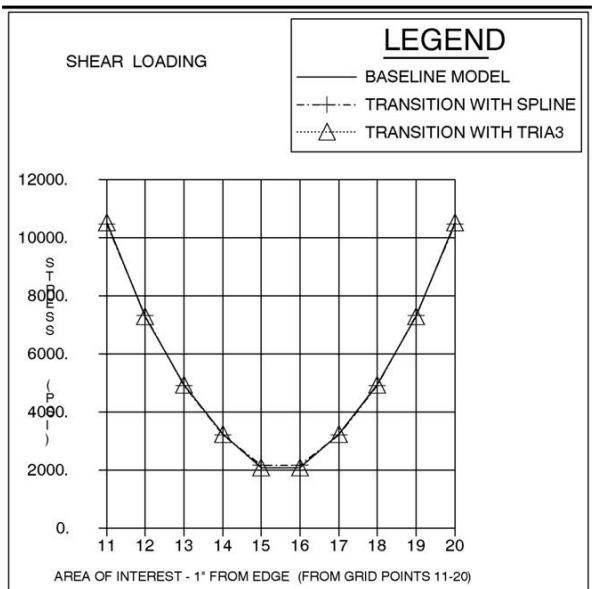


Figure 9-12 Resultant Stresses for Membrane Loads



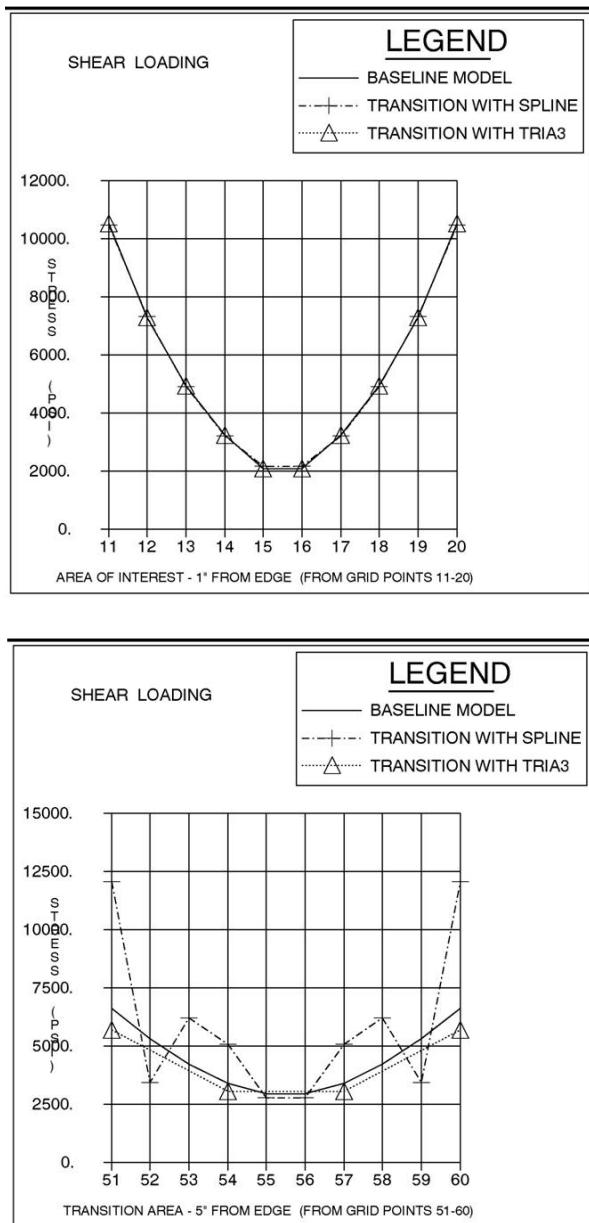


Figure 9-13 Resultant Stresses for In-Plane Shear Loads



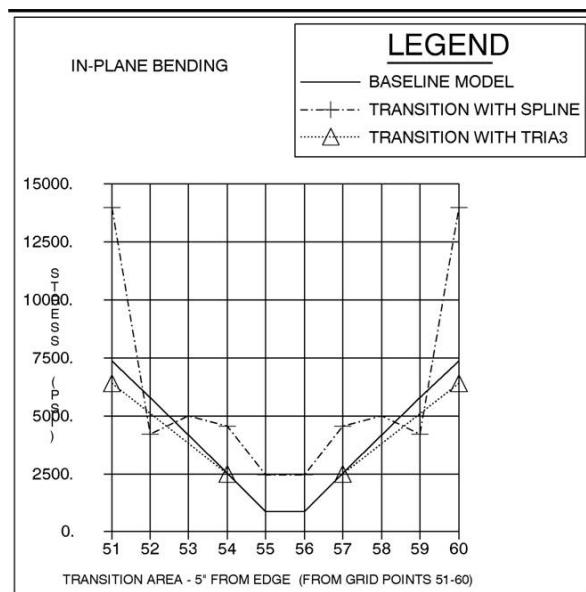
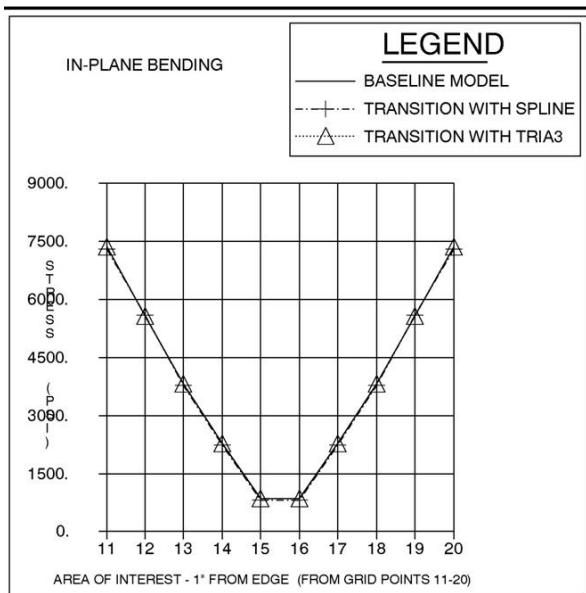


Figure 9-14 Resultant Stresses for In-Plane Bending Loads



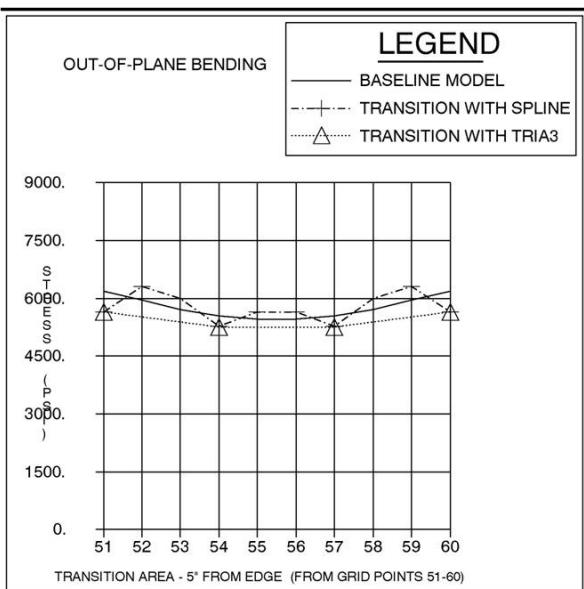
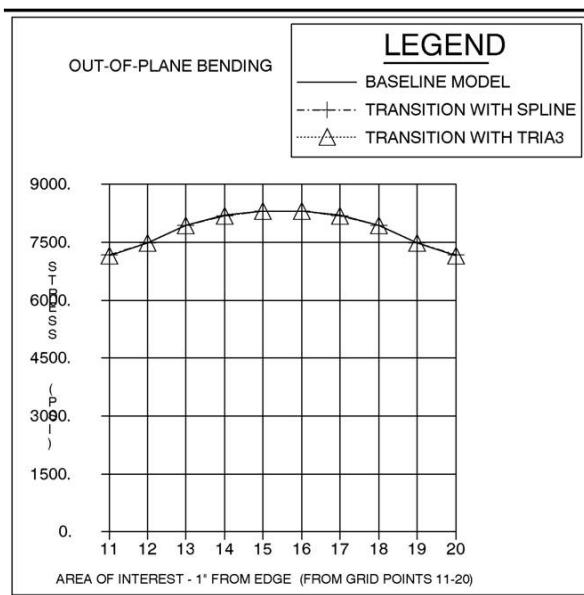


Figure 9-15 Resultant Stresses for Vertical Loads



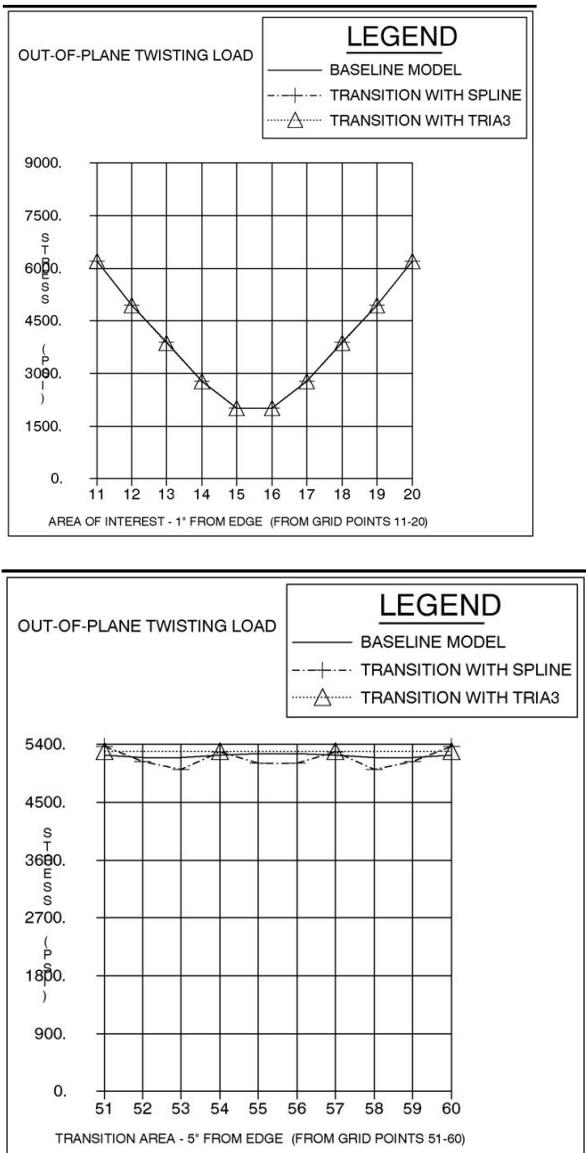


Figure 9-16 Resultant Stresses for Twisting Loads

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 Constraint Elements](#)). 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 secondary 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.

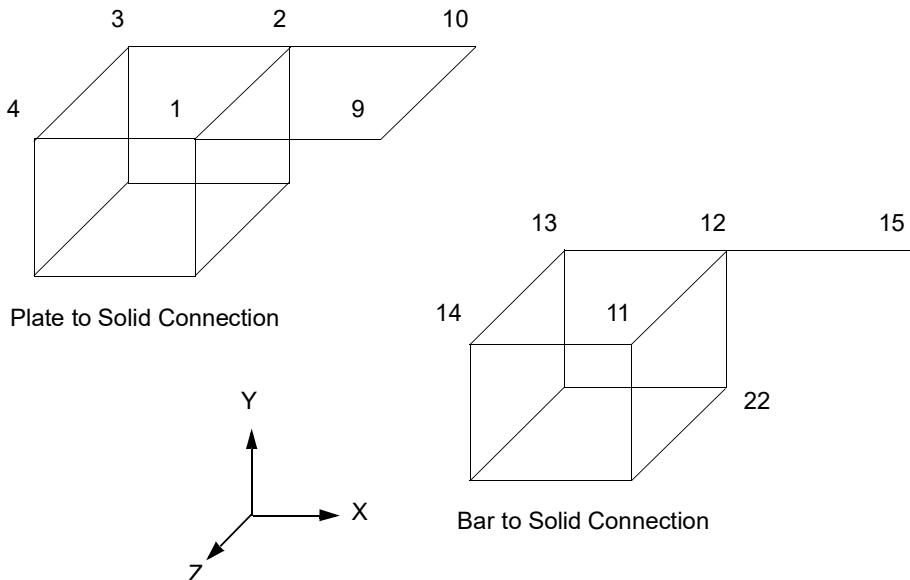


Figure 9-17 Typical Transition Between Dissimilar Elements

For the plate to solid connection, two RBE3 elements suffice:

RBE3

1	2	3	4	5	6	7	8	9	10
\$RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	



	1	2	3	4	5	6	7	8	9	10
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. 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)

The RSSCON for modeling shell-to-solid transition 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.



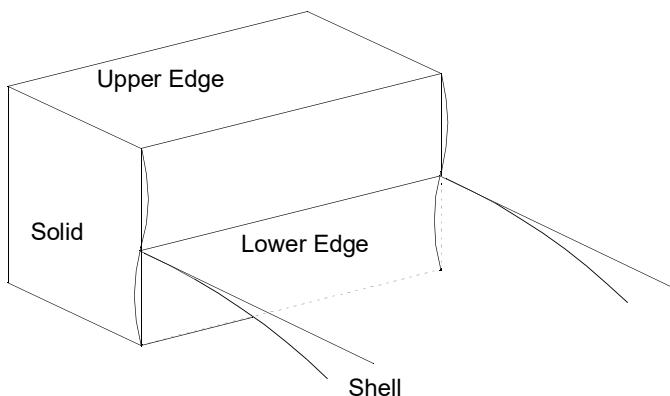


Figure 9-18 Clamped Connection of a Shell to a Solid

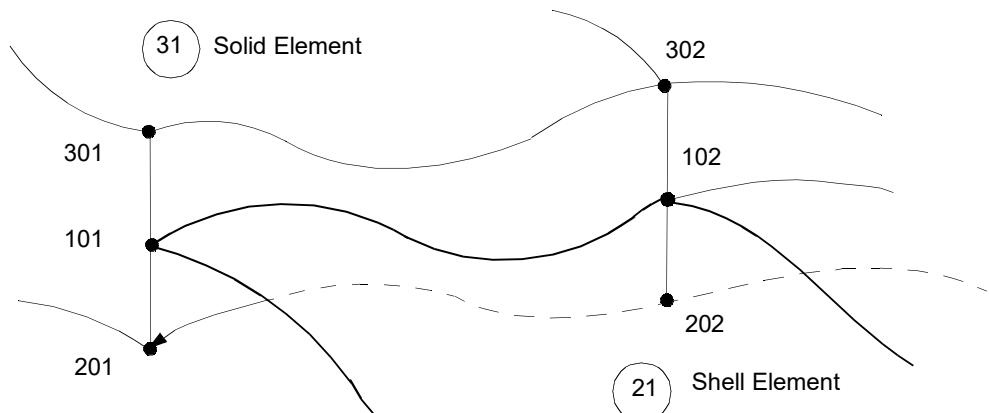
Through the RSSCON Bulk Data entry, the shell elements CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR can be connected to the solid elements CHEXA, CPENTA, and CTETRA. Elements with midside nodes are also supported.

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 are described in the Bulk Data entry [RSSCON](#) 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.





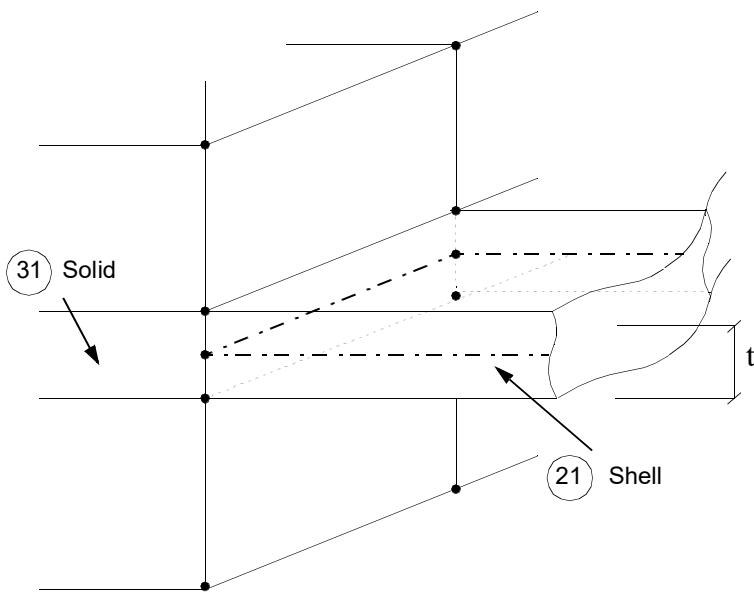
RSSCON, 100, ELEM, 21, 31
or
RSSCON, 200, GRID, 101, 201, 301, 102, 202, 302

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](#).





RSSCON, 100, ELEM, 21, 31

Figure 9-20 Best Modeling Practice for RSSCON

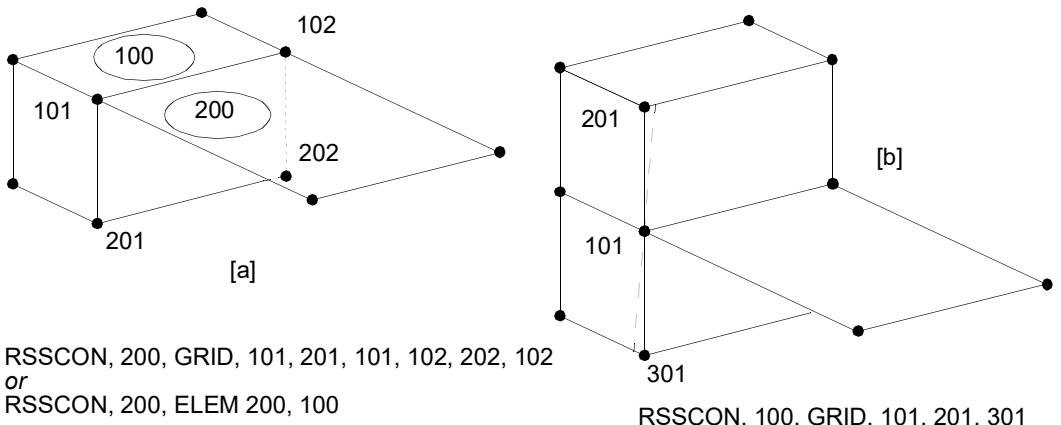


Figure 9-21 RSSCON in Meshes Where Solid and Shell Elements Share Grid Points



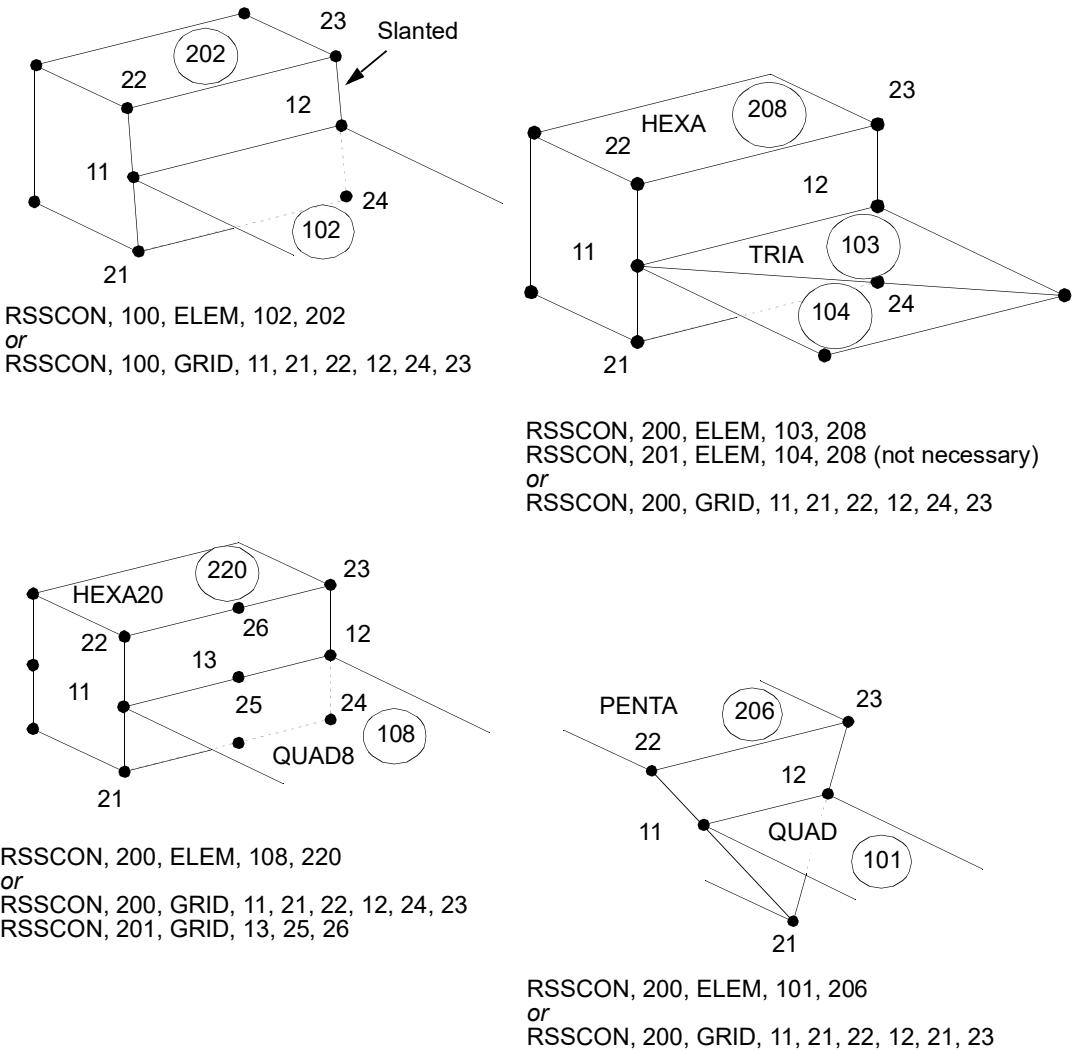


Figure 9-22 Modeling Options with RSSCON

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 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.

By default, the moved shell grid points are not printed.

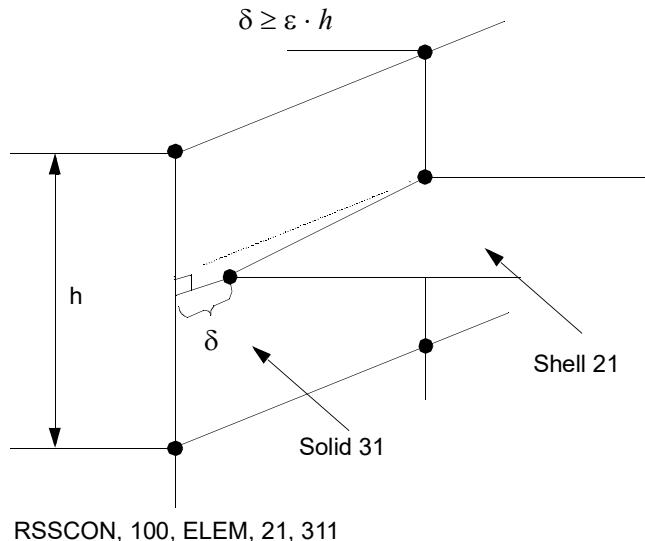


Figure 9-23 RSSCON Modeling Practice Parameters

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, ε 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 CQUAD4s. (See [Install_dir/mscxxxx/doc/linstat/quad41.dat](#))
2. The second model is a plate model consisting of all CHEXAAs. (See [Install_dir/mscxxxx/doc/linstat/hexa1.dat](#))
3. The third model is a plate-solid model using MPCs to transition between the plate and solid model. (See [Install_dir/mscxxxx/doc/linstat/mpc1.dat](#))
4. The fourth model is a plate-solid model using RSSCONs to transition between the plate and solid model. (See [Install_dir/mscxxxx/doc/linstat/rsscon1.dat](#))

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](#). You might want to review the contour plots in a postprocessor.



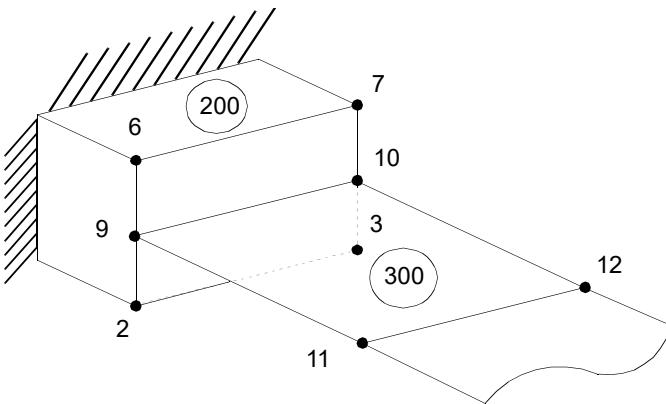


Figure 9-24 Stress Distribution for CQUAD4 Model

Table 9-2 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 CHEXA8s	0.1087	9720.
Plate-to-Solid Transition Using MPCs	0.1103	9720.
Plate-to-Solid Transition Using RSSCONs	0.1103	9720.

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 9-25](#). (See MSC_DOC_DIR/doc/linstat/plate3.dat) 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 9-25](#). 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 9-3](#), 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. 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.



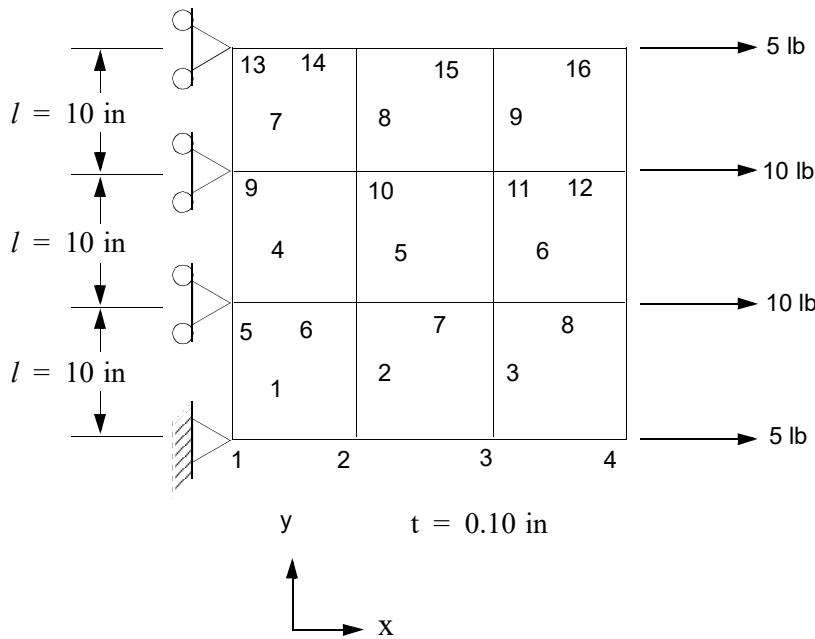


Figure 9-25 Uniaxial Load for a 3 x 3 Mesh

Table 9-3 Element Component Stresses

CQUAD4,7,1,9,10,14,13
 CQUAD4,8,1,10,11,15,14
 CQUAD4,9,1,11,12,16,15
 \$

CQUAD4,4,1,5,6,10,9
 CQUAD4,5,1,6,7,11,10
 CQUAD4,6,1,7,8,12,11
 \$

CQUAD4,1,1,1,2,6,5
 CQUAD4,2,1,2,3,7,6
 CQUAD4,3,1,3,4,8,7
 \$

$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$



CQUAD4,7,1,9,10,14,13 CQUAD4,8,1,10,11,15,14 CQUAD4,9,1,11,12,16,15 \$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$
CQUAD4,4,1,5,6,10,9 CQUAD4,5,1,7,11,10,6 CQUAD4,6,1,7,8,12,11 \$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$	$\sigma_x = 0$ $\sigma_y = 10$ $\sigma_v = 10$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$
CQUAD4,1,1,1,2,6,5 CQUAD4,2,1,2,3,7,6 CQUAD4,3,1,3,4,8,7 \$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$	$\sigma_x = 10$ $\sigma_y = 0$ $\sigma_v = 10$

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 9-25](#) 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 9-25](#), 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 9-25](#), 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 9-5](#). The relations used to compute the average grid point stresses in the volumes containing solid elements are described in [Table 9-5](#). The topological method is the default method.

Table 9-4 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	$\sigma_i = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ <p>where:</p> <p> σ_i = average stress at interior grid points N_e = number of directly connected elements σ_{ei} = element center stresses if only CQUAD4 and/or CTRIA3 elements are connected; element vertex stresses if CQUAD8, CTRIA6, and/or CQUAD4 (with corner output option) elements are connected </p>	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.



Table 9-4 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															
Edge	$\sigma_E = \frac{2}{N_e} \sum_{i=1}^{N_e} \sigma_{ei} - \frac{1}{N_i} \sum_{i=1}^{N_i} \sigma_i$ <p>where:</p> <table border="0"> <tr> <td>σ_E</td> <td>=</td> <td>average stress at edge grid points</td> </tr> <tr> <td>N_e</td> <td>=</td> <td>number of directly connected elements</td> </tr> <tr> <td>N_i</td> <td>=</td> <td>number of interior points connected to the edge points by line segment</td> </tr> <tr> <td>σ_i</td> <td>=</td> <td>average stress at interior grid points connected to the edge points by line segments</td> </tr> <tr> <td>σ_{ei}</td> <td>=</td> <td>element center stresses if only CQUAD4 and/or CTRIA3 elements are connected; element vertex stresses if CQUAD8, CTRIA6, and/or CQUAD4 (with corner output option) elements are connected</td> </tr> </table>	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	σ_{ei}	=	element center stresses if only CQUAD4 and/or CTRIA3 elements are connected; element vertex stresses if CQUAD8, CTRIA6, and/or CQUAD4 (with corner output option) elements are connected	<p>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.</p> <p>Remarks for interior grid points also apply.</p>
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															
σ_{ei}	=	element center stresses if only CQUAD4 and/or CTRIA3 elements are connected; element vertex stresses if CQUAD8, CTRIA6, and/or CQUAD4 (with corner output option) elements are connected															
Corner (CQUAD4 without corner output option)	<p>Corner points connected to CQUAD4 element with the grid point diagonally opposite the corner point, an interior point, or an edge point</p> $\sigma_c = 2\sigma_1 - \sigma_D$ <p>where:</p> <table border="0"> <tr> <td>σ_c</td> <td>=</td> <td>average stress at corner grid point</td> </tr> <tr> <td>σ_1</td> <td>=</td> <td>CQUAD4 center stress</td> </tr> <tr> <td>σ_D</td> <td>=</td> <td>average stress at grid point diagonally opposite corner point</td> </tr> </table> <p>(If the point diagonally opposite the corner point is not an interior or edge point, then $\sigma_c = \sigma_1$)</p>	σ_c	=	average stress at corner grid point	σ_1	=	CQUAD4 center stress	σ_D	=	average stress at grid point diagonally opposite corner point	Remarks for interior grid points also apply						
σ_c	=	average stress at corner grid point															
σ_1	=	CQUAD4 center stress															
σ_D	=	average stress at grid point diagonally opposite corner point															



Table 9-4 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)	<p>Corner points connected to CTRIA3 elements and the other two grid points, namely, E1 and E2 of the connected CTRIA3, are edge points.</p> $\sigma_c = 3\sigma_1 - (\sigma_{E1} + \sigma_{E2})$ <p>where:</p> <div style="background-color: #f0f0f0; padding: 10px;"> 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 $\sigma_c = \sigma_c$.) </div>	Remarks for interior grid points also apply
Grid points connected to CQUAD8, CTRIA6 and/or CQUAD4 (with corner output option)	$\sigma_c = \frac{1}{N_c} \sum_{i=1}^{N_c} \sigma_{ri}$ <p>where:</p> <div style="background-color: #f0f0f0; padding: 10px;"> 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 </div>	Remarks for interior grid points also apply.



Table 9-5 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	$\sigma_1 = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ <p>where:</p> <p>σ_1 = average stress at interior or edge grid points N_e = number of directly connected elements σ_{ei} = element vertex stresses</p>	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 9-26](#) is an example of such a situation. Grid points 1, 2, 3, 4, and 5 are examples of exception grid points.

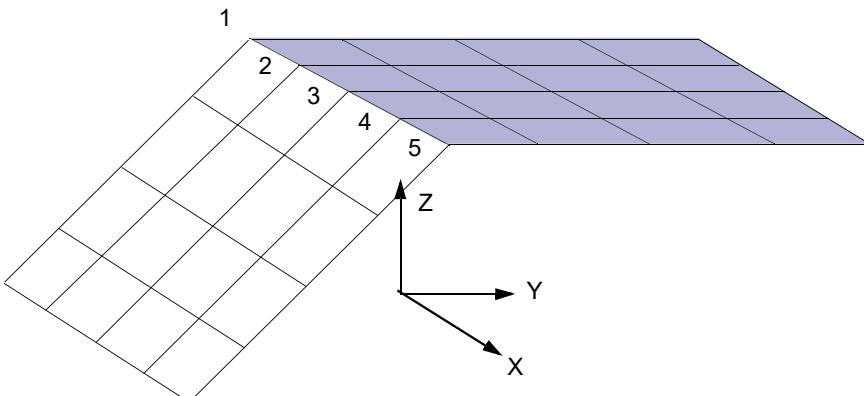


Figure 9-26 Adjacent Elements with Different Slopes Connected to the Same Line



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 9-26](#), 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 9-5](#)).

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 Interface

A sample of the grid point stress data recovery option is shown in this section.

Case Control Section

The following Case Control commands are used by the grid point stresses:

GPSTRESS (Case)	Requests grid point stresses output.
STRFIELD (Case)	Requests grid point stresses for graphical postprocessing and stress discontinuities.
STRESS (Case) or ELSTRESS	Requests element stress output.
OUTPUT (Case)(POST)	Delimiter.
SET (Case)	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, 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. In the model shown in [Figure 9-25](#), 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 9-25](#) is used for this illustration. In this case, you may want to have two separate surfaces. The first surface contains elements 1, 2, 4, 5, 7, and 8; the second surface contains all elements. Part of the input file is shown. (See `Install_dir/mscxxxx//doc/linstat/gpstress.dat`)

```
$  
$ FILENAME - gpstress.dat  
$  
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
```

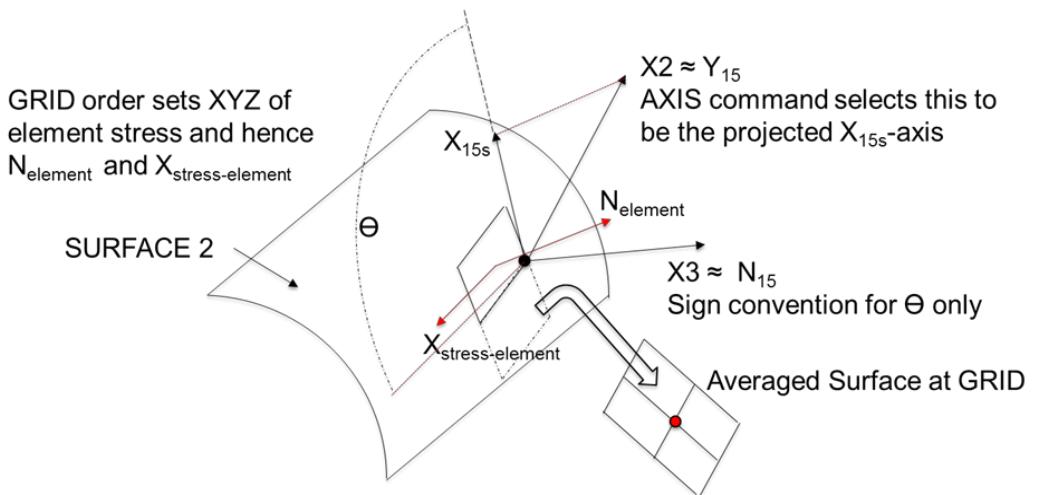
In the above example, surface 100 contains elements 1, 2, 4, 5, 7, and 8 that were referenced by set 1. Surface 200 contains all of the elements that were referenced by set 2. The element positive normal is in the direction of the z-axis of the basic coordinate system. By default, the grid point stresses 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.

S T R E S S E S A T G R I D P O I N T S				S U R F A C E 100								
GRID		ELEMENT		SURFACE X-AXIS X			NORMAL(Z-AXIS) Z		REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID			
				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	
2	0	MID		1.000E+01	-2.009E-14	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01	
		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	
15	0	MID		1.000E+01	-8.986E-15	-1.093E-07	.0000	1.000E+01	0.000E+00	5.000E+00	1.000E+01	
		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	

The above example is idealized in that each surface is flat. Consider the general case illustrated by Figure 9-27 below. When using a specified coordinate system (default=Basic), the designated AXIS (default X1=X) is projected onto the surface defined by the SURFACE entry. The NORMAL (default=R) is only used to define the normal to the surface and reverse it if requested, which then gives the in-plane Y direction based on the right-hand-rule. The stresses are then averaged at each grid after being transformed to this consistent coordinate system and the resulting averaged stress is still a 2D tensor in the plane of the surface. Of course some tricky things are done when the normal of the adjacent elements differ slightly and you can use the TOPO or GEOM methods to take this into account.



SURFACE 2 SET 10 SYSTEM CORD 15 AXIS X2 NORMAL X3

$$\sigma_{\Theta s} = \sigma_{xe} \cos^2 \Theta + \sigma_{ye} \sin^2 \Theta - 2 \tau_{xye} \cos \Theta \sin \Theta - \text{Mohr's circle formula}$$

Figure 9-27 Grid Point Stress illustration of Projection onto SURFACE



So referring to [Figure 9-27](#)

1. The curved surface represents all the elements in SURFACE 2 defined by SET 10 case control (OUTPUT(POST)).
2. The red vectors show a typical element coordinate system at the centroid with X being in the plane of the element and is the X stress direction of the element. N is the normal to the element.
3. These vectors (at the integration points) are extrapolated to the nodes and contributions from adjacent elements are used to create the local surface for a particular grid based on the actual normal vector determined by the TOPO or GEOM methods.
4. The stresses from each element are transformed into the projected coordinate system specified. In this case CORD 15.
5. The black vectors represent COORD 15, where AXIS is specified as X2 (or the Y-Axis) of COORD 15.
6. This axis is projected on to the surface and becomes the X-Axis direction of the consistent coordinate system to which the stresses are transformed.
7. In this example the NORMAL is defined as the X3 (or Z-axis) of COORD 15.
8. So $N_{15} \times X_{15}$ s (cross product) gives the Y-axis of the consistent coordinate system.
9. Stresses are then averaged properly based on this consistent coordinate system.

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 9-28](#). 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 9-28 are applied to each of the respective elements, the force distributions are then equivalent for all the elements.

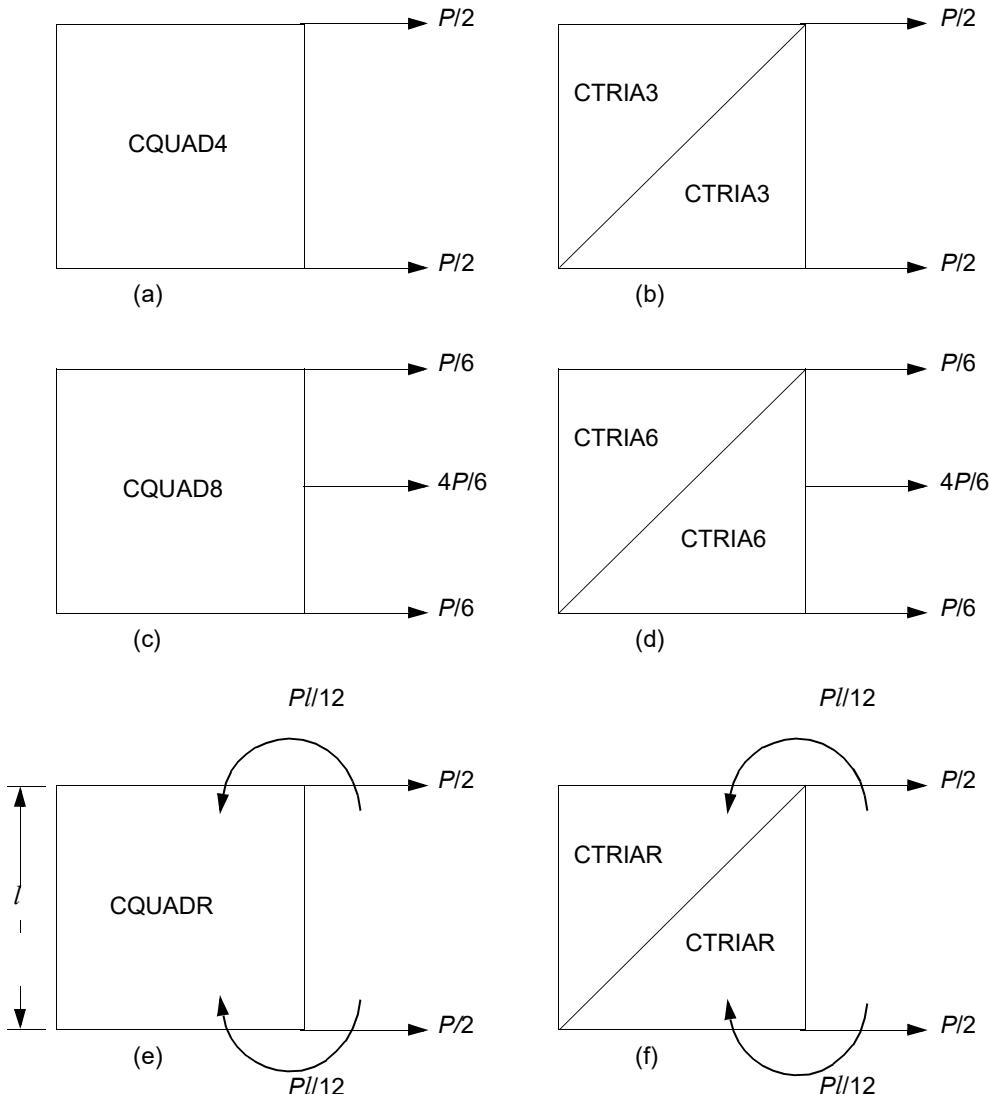


Figure 9-28 Consistent Loads Due to Uniform In-Plane Element Edge Load



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 9-29. 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 9-29. 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 9-29(c) and Figure 9-29(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.

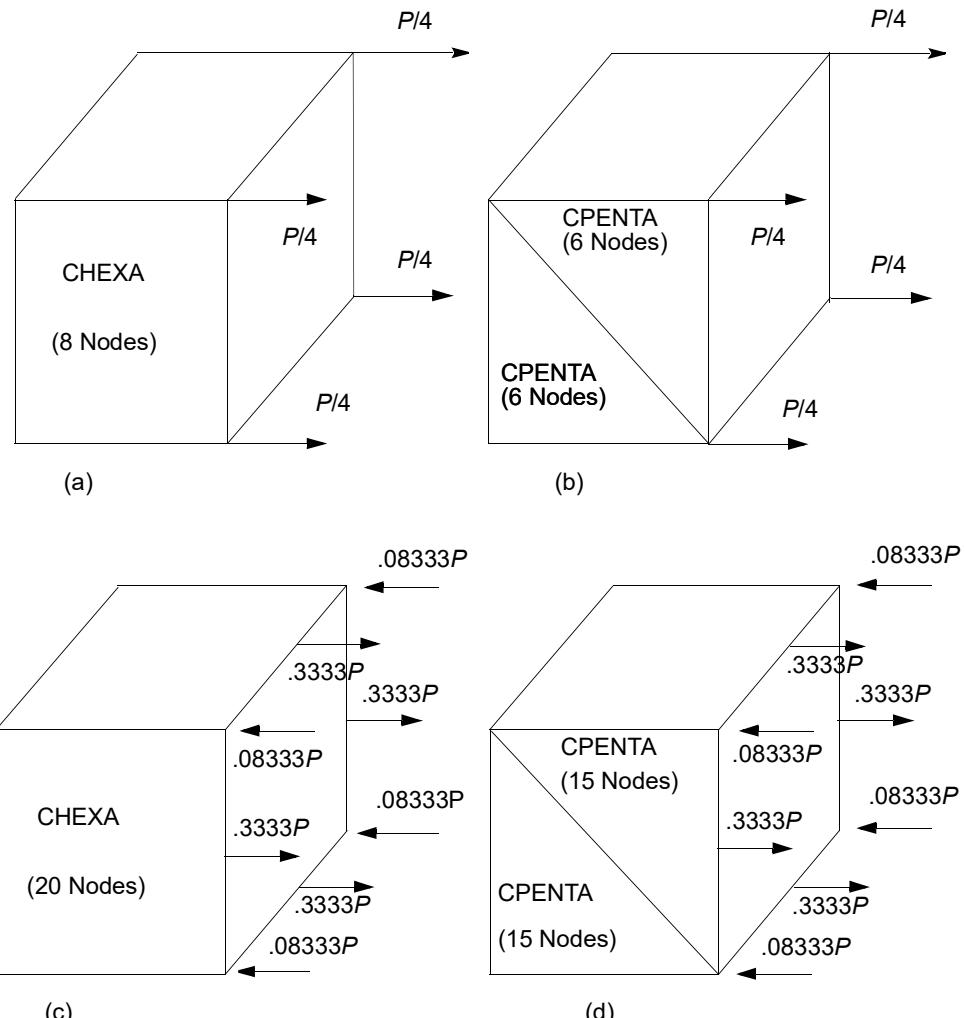


Figure 9-29 Consistent Loads Due to Uniform Loads on a Solid Element Face



As you can see, consistent loads are functions of the element types and applied loads.

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](#). 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 9-30\(a\)](#) and [Figure 9-30\(b\)](#) 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.

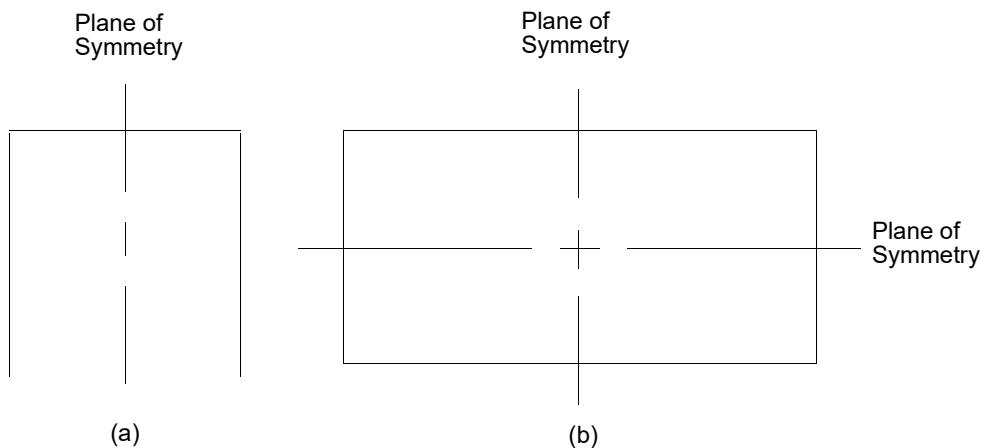


Figure 9-30 Structures with Planes of Symmetry

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 9-31\(a\)](#). This load can be broken into two separate loads as shown in [Figure 9-31\(b\)](#) and [Figure 9-31\(c\)](#).



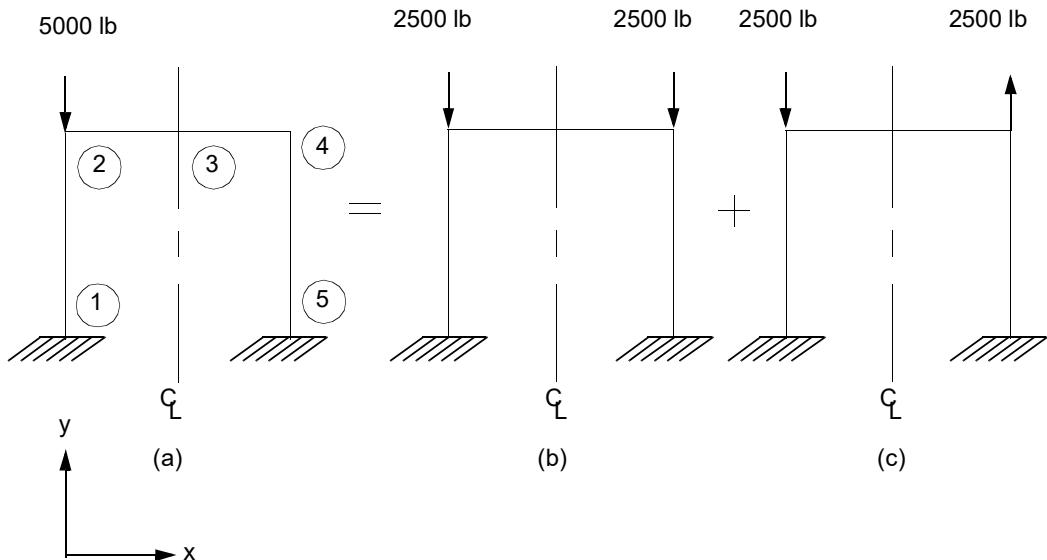


Figure 9-31 Decomposing a General Loading into Symmetric and Antisymmetric Loadings

Note that Figure 9-31(a) is a case of a general loading applied to a structure with one plane of symmetry. Figure 9-31(b) and Figure 9-31(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 9-31(b), needs to be modeled by applying the symmetric boundary condition at the plane of symmetry (see Figure 9-32).

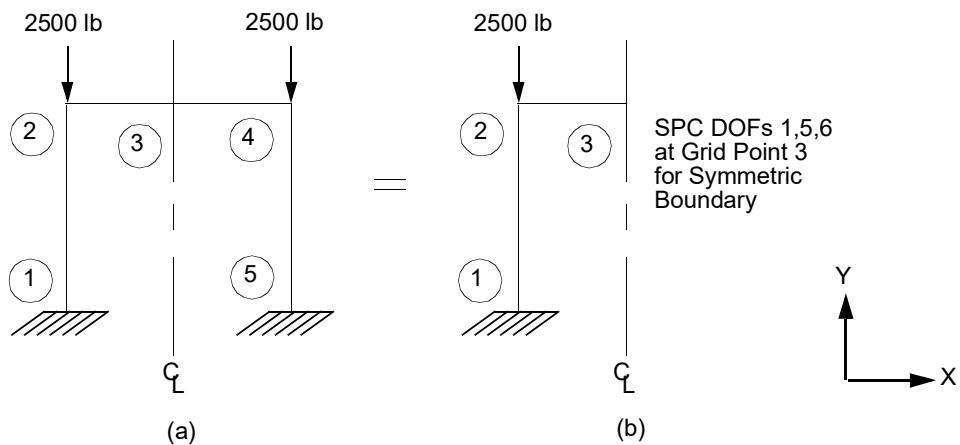


Figure 9-32 Symmetric Model

Similarly, only one-half of the structure shown in Figure 9-31(c) needs to be modeled by applying the antisymmetric boundary condition as shown in Figure 9-33.



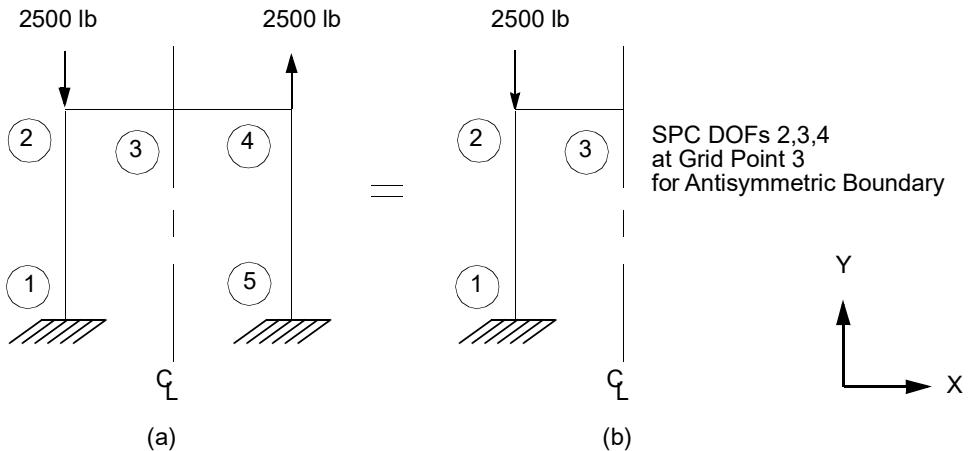


Figure 9-33 Antisymmetric Model

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 activated by Case Control commands. The above frame example is used to illustrate the required Case Control commands. Listing 9-1 contains the input file for this problem. (See MSC_DOC_DIR/doc/linstat/symbar.dat)

Listing 9-1 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
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 selects out the symmetric boundary condition and load as shown in [Figure 9-32\(b\)](#). The second subcase selects out the antisymmetric boundary condition and load as shown in [Figure 9-33\(b\)](#).

The third subcase is a “results combination” subcase. No boundary or load conditions need to be applied for this loading condition (see [Modeling Guidelines, 333](#)). 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 9-34](#) 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	T2	T3	D I S P L A C E M E N T	V E C T O R	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0	.0	.0	.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0	.0	.0	.0	.0	3.453039E-05
RIGHT SIDE OF MODEL - Y LOAD							SUBCOM 4		
POINT ID.	TYPE	T1	T2	T3	D I S P L A C E M E N T	V E C T O R	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0	.0	.0
2	G	1.381215E-04	-5.524853E-06	.0	.0	.0	.0	.0	-2.762431E-05
3	G	1.381215E-04	-1.666667E-04	.0	.0	.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	R1	R2	R3
			T2	T3	R1				
1	G	.0	.0	.0	.0	.0	.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0	.0	.0	.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0	.0	.0	.0	.0	3.453039E-05
4	G	-1.381215E-04	-5.524862E-06	.0	.0	.0	.0	.0	2.762431E-05
5	G	.0	.0	.0	.0	.0	.0	.0	.0

Figure 9-34 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](#)). 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.





10

Model Verification

- Introduction to Model Verification
- Preprocessor Checks
- Strain Energy Output
- Diagnostic Tools
- Stress Error Estimators
- Postprocessor Checks
- Monitor Points
- Summary



Introduction to Model Verification

In [Modeling Guidelines](#) (Ch. 9), 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.

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.

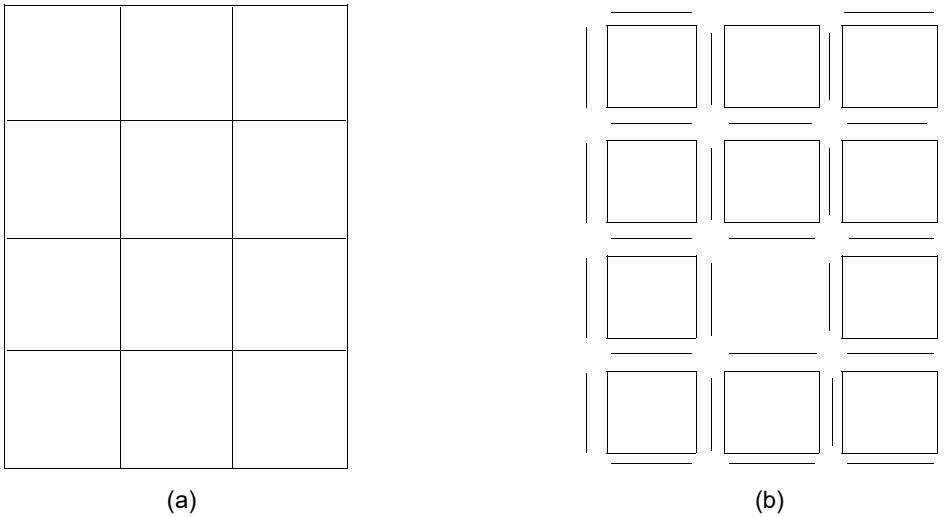


Figure 10-1 Shrink Plot

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.



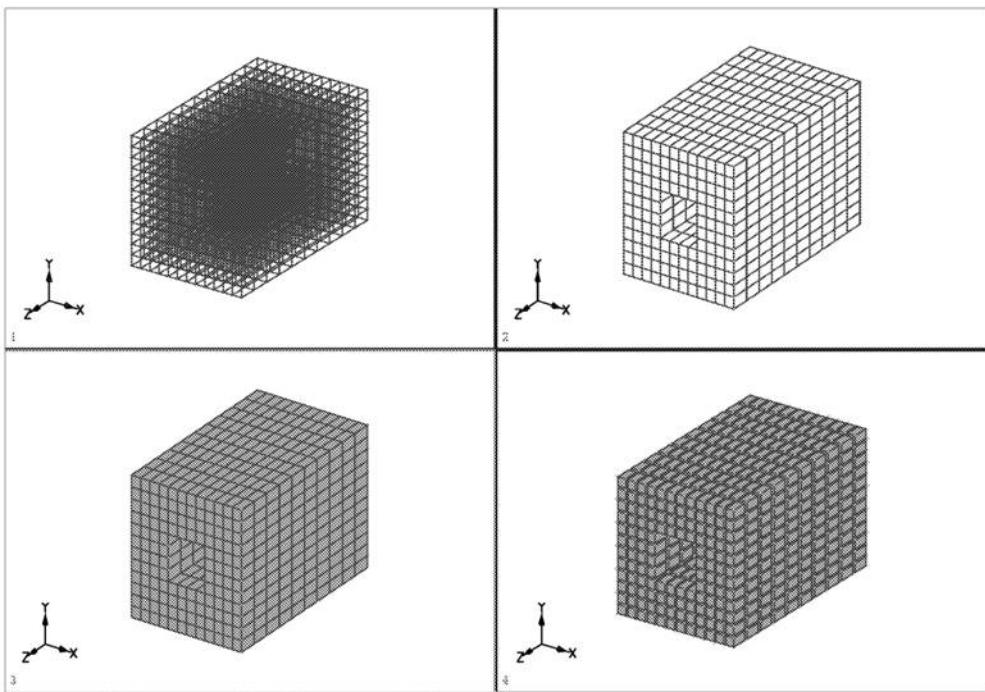


Figure 10-2 Wireframe, Hiddenline, and Shaded Plots

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.



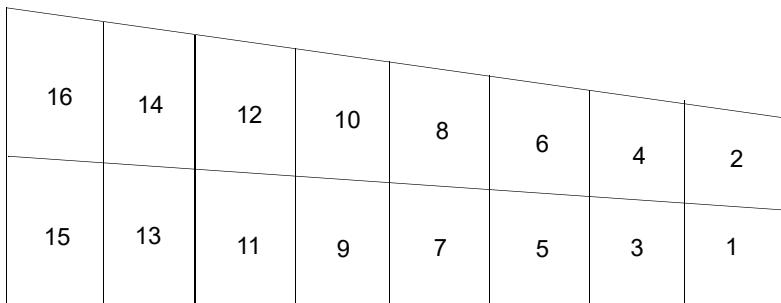


Figure 10-3 Wing Free Edge Check

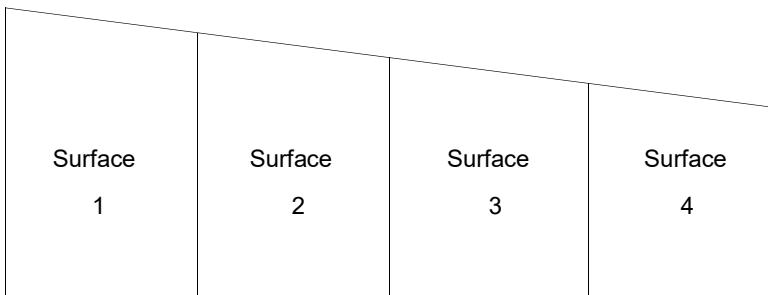


Figure 10-4 Wing Geometric Surfaces

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.



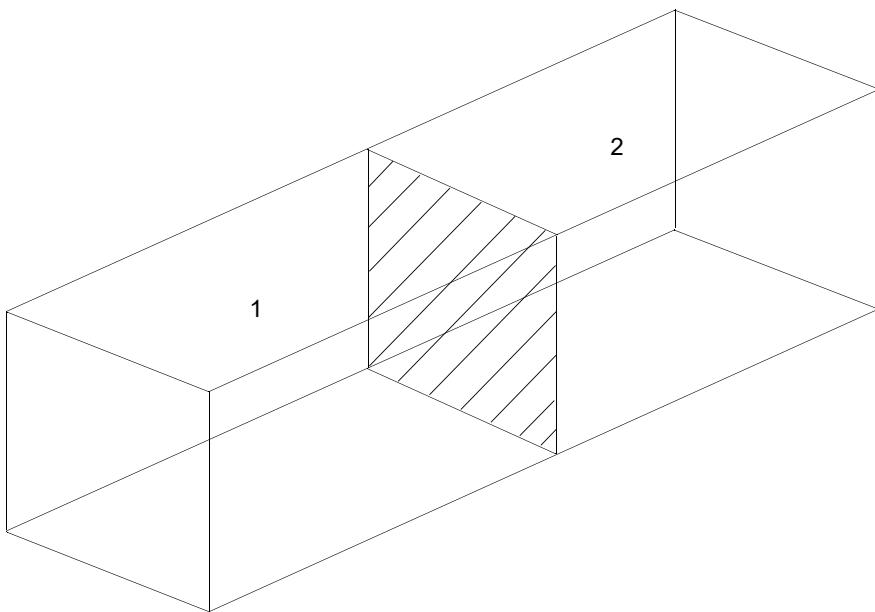


Figure 10-5 Possible Modeling Error as Indicated by Free Face

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.



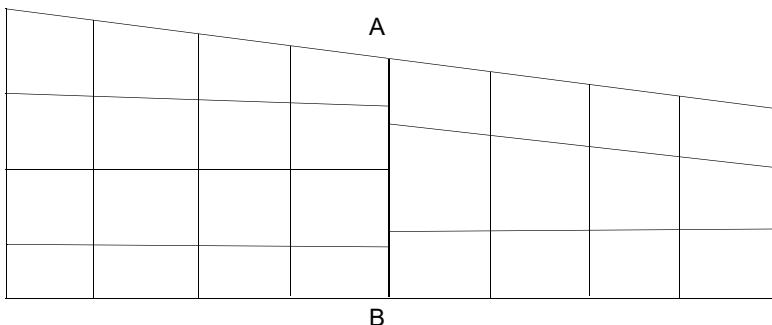


Figure 10-6 Model with Zipper Effect

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](#). 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 [MSC Nastran Elements](#), each CQUAD4 has a positive normal direction associated with it. This direction is defined by the way you connect the grids of the element using the right-hand rule. The symbols used to denote the directions of the normals are shown in [Figure 10-8](#).

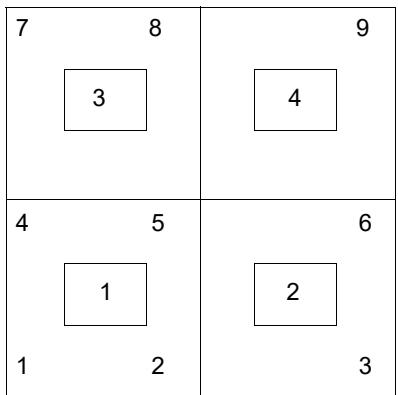


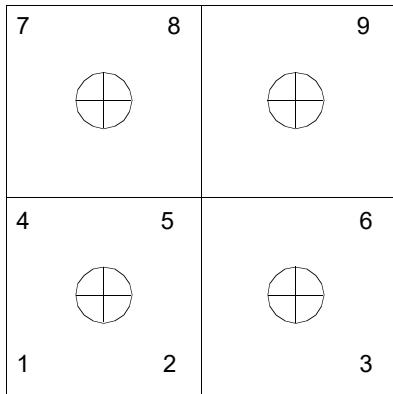
Figure 10-7 CQUAD4 Model

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\(a\)](#).

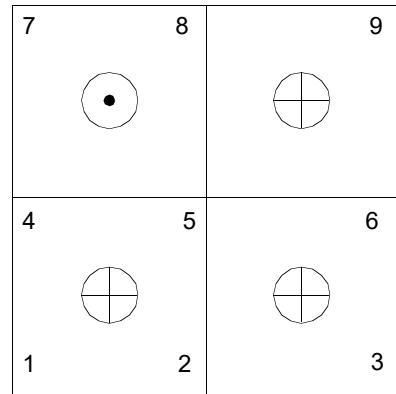


Listing 10-1 Consistent CQUAD4 Connectivity

```
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
```



(a)



(b)



- Positive Normal Pointing into the Paper



- Positive Normal Pointing out of the Paper

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](#)).

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.



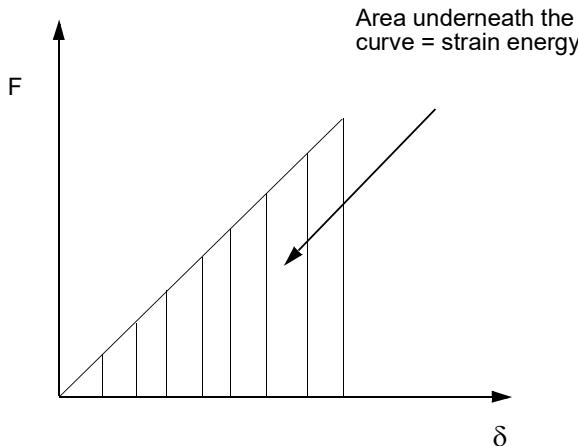


Figure 10-9 Load Deflection Curve

This energy is defined as

$$U = 0.5 \cdot F \cdot \delta$$

Since $F = k \cdot \delta$,

$$U = 0.5 \cdot k \delta^2 \quad (10-1)$$

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.

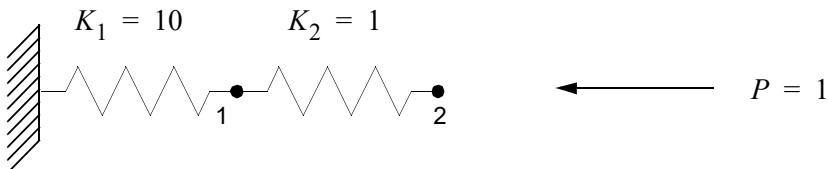


Figure 10-10 Spring Sample Problem

The deflections at grid points 1 and 2 can be calculated as follows:



$$\delta_1 = \frac{P}{K_1}; \delta_2 = \frac{P}{K_1} + \frac{P}{K_2} \quad (10-2)$$

From [Equation \(10-1\)](#) and [Equation \(10-2\)](#),

$$U_1 = 0.5(K_1)(\delta_1)^2 = 0.5(10)\left(\frac{1}{10}\right)^2 = 0.05$$

$$U_2 = 0.5(K_2)(\delta_2 - \delta_1)^2 = 0.5(1)\left[\left(\frac{1}{10}\right) + \left(\frac{1}{1}\right) - \left(\frac{1}{10}\right)\right]^2 = 0.5$$

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 ₂ (m ²)	I ₁₄ (m ⁴)	I ₂₄ (m ⁴)	J ₄ (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



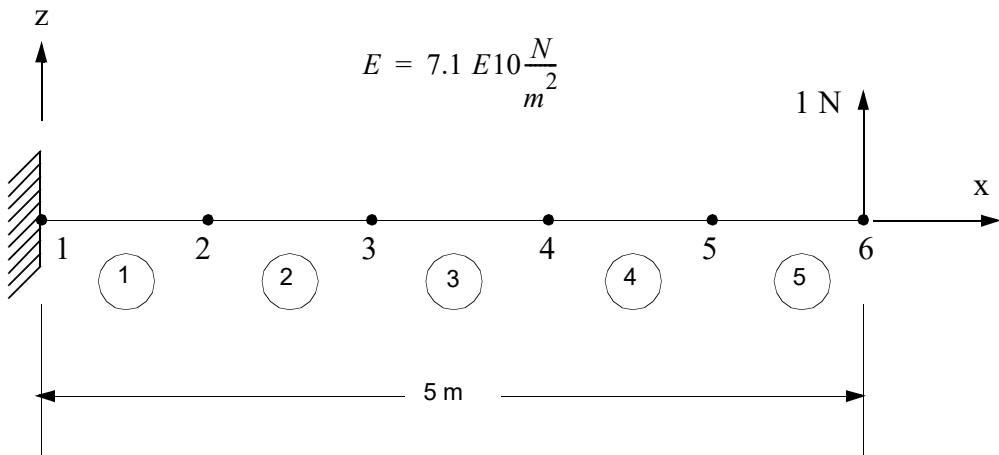


Figure 10-11 Cantilever Beam

When this sample problem is run, the largest deflection occurs at the tip of the cantilever beam. (See [MSC_DOC_DIR/doc/linstat/secant.dat](#)) The vertical deflection at grid point 6 is equal to

$1.126833 \cdot 10^{-3}$ 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 $0.06m \cdot 0.06m$ and 1 m 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. (See [MSC_DOC_DIR/doc/linstat/secant5.dat](#)) If you do this, the vertical displacement at grid point 6 is then reduced from $1.126833 \cdot 10^{-3}$ meters down to $1.122165 \cdot 10^{-3}$ meters, a reduction of only 0.414%. Now review what the element strain energy output suggests. The [ESE \(Case\)](#) Case Control command provides the output summarized in [Figure 10-12](#):

Multiple sets of output requests can be selected by using the Case Control command [SETP \(Case\)](#) in the *MSC Nastran Quick Reference Guide*. If an ESE command references a SETP identification number the all the SETs listed on the SETP entry are processed individually.

For large models, the element strain energy request can potentially generate a large amount of printout. If this is not desired, use the [ESE \(PLOT\) = ALL](#) Case Control command option to create a postprocessing file containing element strain energy data without generating printed output.

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.



Now rerun by replacing element number 1 with the new 0.06×06 element. (See `MSC_DOC_DIR/doc/linstat/SECANT1.dat`) 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.

ELEMENT STRAIN ENERGIES				
ELEMENT-TYPE = BAR	SUBCASE	*	TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM	= 5.634164E-04
	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.

When loads from temperature differences or element deformation are present, the default definition of element strain energy for linear elements differs from the definition for nonlinear elements. In linear elements the element load vector for temperature loads and element deformation, assumes that the temperatures are constant within a subcase. For nonlinear elements, it is assumed that the temperature varies linearly within a subcase. The user may request the definition for nonlinear elements be applied to linear elements by adding `PARAM,XFLAG,2` to the input file.

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.

Element Summary Output (ELSUM)

The `ELSUM (Case)` Case Control command controls the generation of a printed table of informational properties for the various element types present in the input data file. The information produced includes element measures (length, thickness, area, and volume) and mass property data (structural, non-structural, total, and weight [weightmass*total mass]). The information is grouped according to the finite element type by element type/ID or by property type/ID. Both groupings may be selected at the same time. The output produced can be full or subtotals. The SUMMARY only option limits the output to only the mass property



sub-totals. No individual element property output is generated. This reduces the amount of information, if only the totals for an element type are of interest.

With the grouping by element property type, mass property output can be summarized for all of the elements that reference property ids within each property type.

For each element present in the model, physical metrics appropriate to the element type are computed. These metrics may include length, thickness, area, and volume. Mass properties are computed for supported element types. For the CONM1 element, the mass is taken to be the average of the M11, M22 and M33 values on the CONM1 Bulk Data entry. In addition, references to material properties and element properties are gathered for each element. For bar, beam and shell elements, an attempt is made to determine whether the property reference is to a basic property entry, or has been derived from one of the available alternate input entries (PBARL, PBEAML, PBCOMP, PCOMP, PCOMPG). If it is determined that the property is derived, a character (L for PBARL or PBEAML, P for PBCOMP, PCOMP or PCOMPG) is appended to the property identification number on output. For the case of the element type grouping, each element type is summarized in turn. For each element type, information for each element is displayed. Sub-totals are generated for mass property information. For the case of the property type grouping, each property type referenced by an element type is summarized in turn. All of the property ids referenced within the type are summarized in ascending property id order. Within this summary, information for each element referencing the property id is displayed. Various subtotals are accumulated and displayed.

Only the following element types produce mass property information: CBAR, CBEAM, CBEND, CHEXA, CMASSi, CONM1, CONM2, CONROD, CPENTA, CQUAD4, CQUAD8, CQUADR, CRAC2D, CRAC3D, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, CTRIAX6, and CTUBE

The complete description of the Case Control command ELSUM is located in [ELSUM \(Case\)](#) in the *MSC Nastran Quick Reference Guide*. The EID and PID keywords are used to select the particular grouping desired for the element summary output. The EID keyword requests grouping by element type while the PID keyword requests grouping by element property type. The BOTH keyword requests both groupings. The PIDSUM keyword requests that only mass property totals be output for the PID grouping. The EIDSUM keyword requests that only mass property totals be output for the EID grouping.

In the following ELSUM Case Control command output [Figure 10-13](#) is an example of the EID grouping output and [Figure 10-14](#) is an example of the PID grouping output. If PIDSUM or EIDSUM were used, then only the subtotals and totals information would be present in the figures.



ELEMENT PROPERTY SUMMARY (BY ELEMENT TYPE / ID)									
ELEMENT TYPE = BAR									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3401	3401	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
3403	3403	1	3.00000E+00	1.00000E+00	3.00000E+00	1.50000E-01	1.50000E+00	1.65000E+00	1.65000E+00
3410	3403	1	2.00000E+00	1.00000E+00	2.00000E+00	1.00000E-01	1.00000E+00	1.10000E+00	1.10000E+00
17101	17103 L	1	1.00000E+00	1.01034E+00	1.01034E+00	5.05168E-02	0.00000E+00	5.05168E-02	5.05168E-02
17103	17103 L	1	3.00000E+00	1.01034E+00	3.03101E+00	1.51550E-01	0.00000E+00	1.51550E-01	1.51550E-01
17110	17103 L	1	2.00000E+00	1.01034E+00	2.02067E+00	1.01034E-01	0.00000E+00	1.01034E-01	1.01034E-01
SUBTOTAL MASS FOR ALL BAR					6.03101E-01	3.00000E+00		3.60310E+00	3.60310E+00
.									
ELEMENT TYPE = BEAM									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
200	200	1	1.00000E+00	1.00000E+02	1.00000E+02	5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00
17000	17000 L	1	1.00000E+00	9.99998E+01	9.99987E+01	4.99994E+00	0.00000E+00	4.99994E+00	4.99994E+00
SUBTOTAL MASS FOR ALL BEAM					9.99994E+00	0.00000E+00		9.99994E+00	9.99994E+00
.									
ELEMENT TYPE = PENTA									
ELEM ID	PROP ID	MATL ID		VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
6801	6801	2		4.00000E-02	2.00000E-03	0.00000E+00	2.00000E-03	2.00000E-03	
SUBTOTAL MASS FOR ALL PENTA					2.00000E-03	0.00000E+00		2.00000E-03	2.00000E-03
.									
ELEMENT TYPE = QUAD4									
ELEM ID	PROP ID	MATL ID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3301	3301	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
3313	3313	1	6.12009E-01	6.36864E+00	3.89766E+00	1.94883E-01	1.91715E+00	2.11203E+00	2.11203E+00
3321	3321 P	3321	1.00000E+00	1.00000E+02	1.00000E+02	1.00000E+02	0.00000E+00	1.00000E+02	1.00000E+02
17503	17503	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	0.00000E+00	5.00000E-02	5.00000E-02
SUBTOTAL MASS FOR ALL QUAD4					1.00295E+02	2.41715E+00		1.02712E+02	1.02712E+02
.									
ELEMENT TYPE = ROD									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
101	101	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
107	101	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
SUBTOTAL MASS FOR ALL ROD					1.00000E-01	1.00000E+00		1.10000E+00	1.10000E+00
.									
ELEMENT TYPE = TETRA									
ELEM ID	PROP ID	MATL ID		VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
3901	3901	1		5.77350E-01	2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02	
SUBTOTAL MASS FOR ALL TETRA					2.88675E-02	0.00000E+00		2.88675E-02	2.88675E-02
.									
TOTAL MASS FOR ALL SUPPORTED ELEMENT TYPES					1.38647E+02	1.53344E+01		1.53981E+02	1.53981E+02

Figure 10-13 Sample Element Summary by Element Type Output



ELEMENT PROPERTY SUMMARY (BY PROPERTY TYPE / ID)									
PROPERTY TYPE = PBAR, ID = 3401									
ELEM ID	ELEM TYPE	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3401	BAR	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
SUBTOTAL MASS FOR ALL BAR	ELEMENTS	FOR PBAR,	ID = 3401		5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
.
PROPERTY TYPE = PBEAM, ID = 200									
ELEM ID	ELEM TYPE	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
200	BEAM	1	1.00000E+00	1.00000E+02	1.00000E+02	5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00
SUBTOTAL MASS FOR ALL BEAM	ELEMENTS	FOR PBEAM,	ID = 200		5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00	
.
PROPERTY TYPE = PSOLID, ID = 3901									
ELEM ID	ELEM TYPE	MATL ID		VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
3901	TETRA	1		5.77350E-01	2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02	
SUBTOTAL MASS FOR ALL SOLID	ELEMENTS	FOR PSOLID,	ID = 3901		2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02	
.
PROPERTY TYPE = (NONE)									
ELEM ID	ELEM TYPE	MATL ID			STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
2901	CONM1	0		1.00000E+00	0.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	
2904	CONM1	0		2.20000E+00	0.00000E+00	2.20000E+00	2.20000E+00	2.20000E+00	
SUBTOTAL MASS FOR ALL CONM1	ELEMENTS				3.20000E+00	0.00000E+00	3.20000E+00	3.20000E+00	
.
PROPERTY TYPE = PSHELL, ID = 3301									
ELEM ID	ELEM TYPE	MATL ID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3301	QUAD4	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
SUBTOTAL MASS FOR ALL SHELL	ELEMENTS	FOR PSHELL,	ID = 3301		5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
.
===== TOTAL MASS FOR ALL SUPPORTED ELEMENT TYPES =====									
					1.38647E+02	1.53344E+01	1.53981E+02	1.53981E+02	
					=====	=====	=====	=====	=====

Figure 10-14 Sample Element Summary by Element Property Type Output.

Element Geometry Checks (GEOMCHECK)

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 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 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. 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, PYRAM 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.

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.

The geometry check is requested by the Executive Control Statement **GEOMCHECK** in the *MSC Nastran Quick Reference Guide* and the format is as follows:

FATAL
GEOMCHECK test_keyword [= tol_value], [MSGLIMIT=n], **[MSGTYPE = INFORM]**, [SUMMARY], [NONE]
WARN

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_EPIA	Real ≥ 0.0	150.0	Edge point included angle in degrees
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_EPIA	Real ≥ 0.0	150.0	Edge point included angle in degrees
HEX_DETJ	Real	0.0	$ J $ minimum value
HEX_WARP	Real ≥ 0.0	0.707	Face warp coefficient



Name	Value Type	Default	Comment
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_EPIA	Real ≥ 0.0	150.0	Edge point included angle in degrees
PEN_DETJ	Real	0.0	$ J $ minimum value
PEN_WARP	Real ≥ 0.0	0.707	Quadrilateral face warp coefficient
PYR_AR	Real ≥ 0.0	100.0	Longest edge to shortest edge aspect ratio
PYR_EPLR	Real ≥ 0.0	0.50	Edge point length ratio
PYR_EPIA	Real ≥ 0.0	150.0	Edge point included angle in degrees
PYR_DETJ	Real	0.0	$ J $ minimum value
PYR_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 provides a sample of the 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 LENGTH RATIO DETERMINANT 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 SKW 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 SKW 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 SKW 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									
ELEMENT TYPE	SKEW ANGLE	ASPECT/TAPER RATIO	MINIMUM INTER. ANGLE	MAXIMUM INTER. ANGLE	SURFACE/FACE WARP FACTOR	OFFSET RATIO	EDGE POINT LENGTH RATIO	JACOBIAN 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-	-.20	(TOLERANCE = -.02).

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.

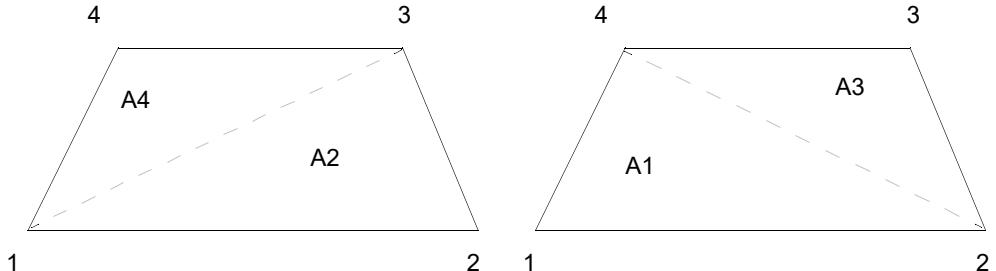
CQUAD4 and CQUADR

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 maximum of the four values is taken as the taper ratio. If the ratio exceeds the tolerance, an informational message is produced.



$$\text{Taper (i)} = \left| \frac{A_i}{A_{\text{quad element}}/2} - 1 \right|, i = 1, 2, 3, 4.$$

A_i is the i -th area (A_1, A_2, A_3 or A_4); and $A_{\text{quad element}}$ is the total area of the quadrilateral element.

The maximum of the four values is used as the taper ratio.

Taper message is issued if taper ratio > 0.5.

Figure 10-15 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.

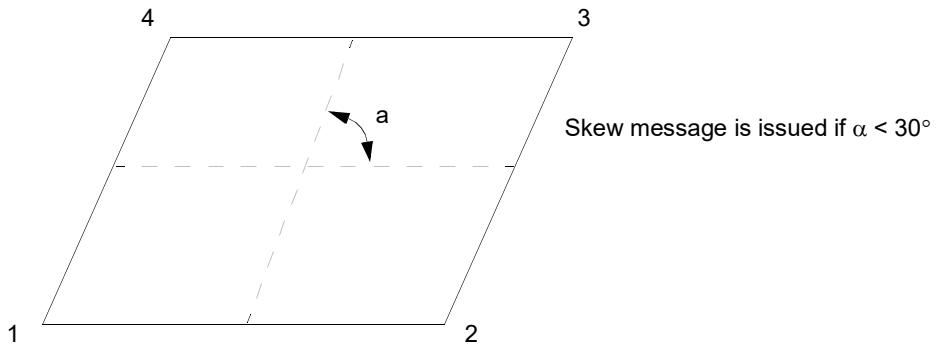


Figure 10-16 CQUAD4 Skew



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

Two optional tests are performed on the triangular shaped planar elements. They are the Interior Angle test and Skew test as discussed previously under the CQUAD4 and CQUADR test description.

CHEXA, CPENTA, CPYRAM 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 and Edge Point Included Angle 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 the tolerance (default 150.0 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, CPYRAM 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.



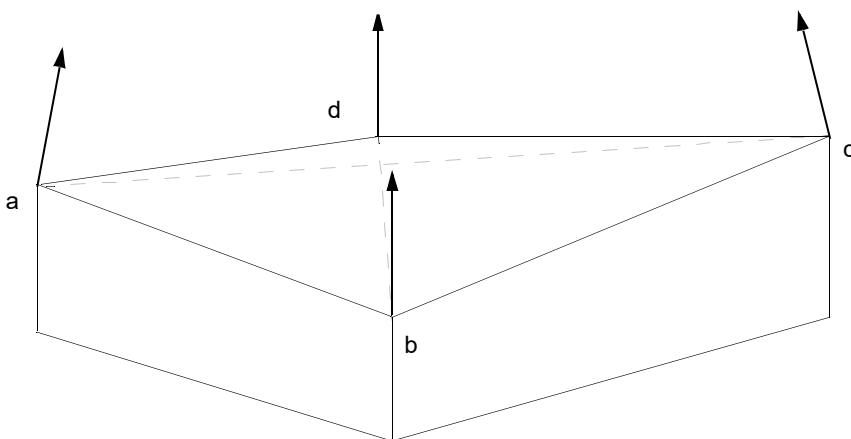


Figure 10-17 CHEXA - Warping

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.

Model Checking Material Properties (MODEL_CHECK)

The main purpose of this feature is to provide the users with a means to evaluate the mass and thermal load characteristics of a structure. This feature is intended primarily for linear static analysis runs that are being used to validate aspects of the finite element model prior to embarking on more expensive production static and dynamics analyses. It allows the user to temporarily modify the material density and thermal expansion coefficient fields of material property bulk data entries. The MODEL_CHECK Executive Control statement and several NASTRAN Statement keywords (DEF_i's) are used. The affected material property fields will be temporarily modified during the run. All results will be based on the temporary values of the material properties. The material property data stored on the Material Property Table data block is not modified and retains the original values supplied on the bulk data entries.

Editing large models can be slow, tedious and error prone. This can be replaced with MODEL_CHECK. If MODEL_CHECK is selected the appropriate entry of the material property data (mass density and/or thermal expansion coefficient(s)) is set to the user-specified value. All subsequent computations use the updated values. There is no need to edit Bulk Data input with multiple sets of material or thermal property entries.

This feature is activated using the MODEL_CHECK Executive Control statement. The MODEL_CHECK statement features keywords that allow a value to be temporarily assigned to the material density and thermal expansion coefficients. The general format of the statement used to change material properties to a specified value is:

```
MODEL_CHECK [ MAT_DENSITY=value, ] [ MAT_TECO=value, ] [ MAT_TEIJ=value ]
```



This feature is similar to PARAM,CHECKOUT,YES, but does not require modifying the Bulk Data section of the input data.

- MODEL_CHECK is ignored in RESTARTs
- Material property updates will take effect for all MAT1, MAT2, MAT3, MAT8 and MAT9 Bulk Data entries present in the input.
- Material properties stored on the Material Property Table (MPT) data block are those present on the bulk data material property entries.
- Post-processing of results could reference in-consistent data since the results are based upon different material property data than is present in the bulk data.

Part of a sample problem (see MSC_DOC_DIR/doc/linstat/varmat10a.dat) is presented to demonstrate the material property update feature. The example consists of several disjoint models and is not intended to be representative of any production model. The MODEL_CHECK statement is used to temporarily set all material densities and thermal expansion coefficients to 0.0. This results in model mass contributions from only the CONM2 element and the non-structural mass as well as results for the temperature load case (subcase 3) being null.

```
$*****
$***** Version: 2004+
$TEST DECK NAME: varmat01a.dat (Baseline Model w/MODEL_CHECK)
$          Density is not temperature dependent.
$          Thermal expansion coefficient is not temp. dependent
$          MODEL_CHECK command present.
$PURPOSE:
$  Demonstrate usage of the MODEL_CHECK Executive Control Statement
$  Note that non-structural mass cannot be modified using this command.
$DESCRIPTION:
$  The model consists of
$  1) CONM2 eid 3001 contributing mass.
$  2) ROD eid 101 referencing MAT1 materials
$  3) QUAD4 eid 3301 w/ PSHELL referencing MAT1 materials
$  4) QUAD4 eid 3313 w/ PSHELL referencing MAT1 and MAT2 materials
$  5) QUAD4 eid 3321 w/ PCOMP referencing MAT8 materials
$  6) HEXA eid 6701 w/ PSOLID referencing MAT9 materials
$  7) TRIAX6 id 5301             referencing MAT3 materials
$A MODEL_CHECK executive command is used to modify material property
$data temporarily during the run. The command demonstrates
$ 1) setting the material density to 0.0 via the OFF option
$ 2) setting the material thermal expansion coefficients to 0.0 via
$     the OFF option
$ 3) continuation of MODEL_CHECK command
$EXPECTED RESULTS:
$ 1) There should be an informational message indicating which of the
$     material property data is being modified.
$ 2) The grid point weight generator and element summary outputs should
$     indicate mass for only the conm2 element and the non-structural
```



```
$      masses of the rods and plates
$ 3) results output for subcase 3 (temperature loading) should be 0.0
$      since MODEL_CHECK is setting the alphas=0.0
$
ID MSC, varmat
TIME 30 $
SOL 101
$
=====
$ temporarily set the density and thermal expansion coefficients for all
$ materials to 0.0
$
model_check  mat_density=off,
              mat_tecoeff=off,mat_teij=off
$
=====
$
CEND
TITLE= VARIABLE MATERIAL TEST CASE
SUBTITLE = MSC V2005
ECHO = SORT
SPC = 12
TEMP(MATE) = 1
SET 10002= 1 THRU 13999, 14399 THRU 16900
DISP=ALL
ELFORCE=ALL
OLOAD = ALL
$      SPCF=ALL
ESE=10002
SEFINAL=1
GPFO=10002
$
ELSUM(EID,PRINT) = ALL
$
SUBCASE 1
LABEL = STATIC LOAD
LOAD = 1
P2G=MATLOAD
K2GG=MATK
SUBCASE 3
LABEL = TURN ON TEMP LOADS
TEMP(LOAD) = 2
$
BEGIN BULK
$does not vary, so it's not included here
ENDDATA
```

The printed output for the example problem consists of all of the output typically generated by a static analysis for the requests present in the case control section of the input and is not reproduced here. There is a small section of additional printed output generated in the Executive Control Echo region when the MODEL_CHECK material property options are used. The output summarizes the requested material property updates and is shown on the next several lines.



```

$ MODEL_CHECK MAT_DENSITY=OFF,
      MAT_TCOEFF=OFF, MAT_TEIJ=OFF
*****
***** THE PRESENCE OF THE MODEL CHECK EXECUTIVE STATEMENT CAUSES THE FOLLOWING MATERIAL ****
***** PROPERTY VALUES TO BE MODIFIED FOR EVERY MAT1, MAT2, MAT3, MAT8 AND MAT9 MATERIAL ****
***** PROPERTY ENTRY PRESENT IN THE INPUT BULK DATA SECTION: ****
***** MATERIAL DENSITY REVISED VALUE = 0.0000E+00 ****
***** THERMAL EXP. DIRECT COEFF. REVISED VALUE = 0.00000E+00 ****
***** THERMAL EXP. SHEAR COEFF. REVISED VALUE = 0.00000E+00 ****
*****

```

There is no other printed output generated that is peculiar to the material property update feature. All other output is typical printed output that is generated by the case control requests. For this example, the Grid Point Weight Generator output was requested via param,grdpnt,0 in the bulk data so that the mass of the structure could be verified as being produced by only the CONM2 element and the element non-structural mass.

Weight Checks (PARAM,GRDPNT,WEIGHTCHECK)

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 Sections. See [Grid Point Weight Generator](#) for a detailed explanation.

The data entry format is

PARAM, GRDPNT, i

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 [WEIGHTCHECK \(Case\)=YES](#) Case Control command will also activate the GPWG output, but only the “G” set can be selected in statics.

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 is shown in [Figure 10-18](#). (See [MSC_DOC_DIR/doc/linstat/diag.dat](#))

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 10-18 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



(MAT1, 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, x`

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/m3)
$ MAT1    1      7.1E10      .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      7.1E10      .33      2.65E4
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, $(0.1019 \times 2.65E4 = 2700)$. 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 output is contained in [Four Concentrated Mass Model](#) (App. 20).

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 3×3 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](#) in the *MSC Nastran Quick Reference Guide* 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. The case control command AUTOSPC replaces PARAM,EPZERO and PARAM,AUTOSPC.

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 $|\varepsilon|$ 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](#) 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.

Solver MAXRATIO Information Output

The SPARSESMV Executive Control statement provides the analyst better control of the generation of matrix diagonal term ratio statistics produced by the sparse symmetric matrix decomposition process in the DCMP module, then the MAXRATIO DMAP parameter. In addition, a output data option is available in the form of a simple bar chart that provides a comprehensive view of the ratio data.

The method involves the computation of a ratio defined as the original matrix diagonal term divided by the decomposed matrix diagonal term. These ratios are placed in a table together with the external identifier associated with the row/column of the term. This table is then processed according to the options requested by the user. The matrix diagonal term ratio output options are controlled by keywords specified on the SPARSESMV Executive Control statement. See [SPARSESMV](#) in the *MSC Nastran Quick Reference Guide* for a complete description of this statement.

The matrix diagonal term ratios can be presented in two different views. The first view is the table view, in which each ratio is listed together with the external identifier of the row/column of the matrix, as well as the original input matrix diagonal term.

The second view of the ratios is statistical in nature. It is similar to a bar chart. A series of bar segments are generated. There are two options for specifying the segment widths of the bars. The default option uses powers of 10 as the widths (e.g., 10.0 to 100.0, and 100.0 to 1000.0). The second option allows the user to specify how many segments are desired. The program will compute the segment width using the maximum and minimum ratios. For each bar in the chart, the total number of terms in the range is tabulated together with a visual indication of the percentage number of terms in that particular bar.

Note that when negative matrix diagonal term ratios are detected, they will always be output if the TABLE option is specified.

These views of the ratios do not replace any existing diagnostics generated by the DCMP module if a problem is detected. Under these conditions, output from the table view may duplicate previous output generated by DCMP module error processing.

The defaults produce both table and bar outputs. The table is limited to 25 ratios that exceed 1.0E+05. The bar chart uses powers of ten for segment widths. The basic use is by adding

```
SPARSESMV DCMP (MDTRATIO)
```

in the Executive Control Section of the input data file.

The use of this feature is limited to sparse symmetric matrix operations in the DCMP module.

If there are scalar-type points present in the problem, the degrees of freedom associated with these points will be grouped into the results for the translational degrees of freedom output.



A simple example is presented that demonstrates the use of some of the features available for output of the matrix diagonal term ratios. The SPARSEESOLVER Executive Control statement is used to specify the desired features. The example is for demonstration purposes only, and is not representative any particular modeling situation. The model data consists of a simple plate structure subject to an end load.

The output generated by the example is shown as follows. Notice that there are two separate sections of output: one for translational degrees of freedom, and one for rotational degrees of freedom. Within each section, both a bar chart and table of matrix diagonal term ratios are output.

TRANSLATIONAL DOF DIAGONAL TERM RATIO STATISTICS CHART FOLLOWS FOR THE DECOMPOSITION OF MATRIX KLL						
DIAGONAL TERM RATIO RANGE		#TERMS	% TOT	MAXIMUM RATIO =	MINIMUM RATIO =	
1.0000E+00	TO	1.0000E+01	62	79.49	*****	>
1.0000E+01	TO	1.0000E+02	12	15.38	*****	
1.0000E+02	TO	1.0000E+03	4	5.13	*****	

0	MATRIX/FACTOR DIAGONAL TERMS RATIO SUMMARY TABLE FOR TRANSLATIONAL DOF SORTED ON DIAGONAL RATIO	GRID POINT ID	DEGREE OF FREEDOM	MATRIX/FACTOR DIAGONAL RATIO	MATRIX DIAGONAL	
				(TOP 1 RATIOS>MAXRAT=	6.90963E+02)	
		58	T3	6.90963E+02	5.65535E+04	

ROTATIONAL DOF DIAGONAL TERM RATIO STATISTICS CHART FOLLOWS FOR THE DECOMPOSITION OF MATRIX KLL						
DIAGONAL TERM RATIO RANGE		#TERMS	% TOT	MAXIMUM RATIO =	MINIMUM RATIO =	
1.0000E+00	TO	1.0000E+01	38	63.33	*****	>
1.0000E+01	TO	1.0000E+02	18	30.00	*****	
1.0000E+02	TO	1.0000E+03	4	6.67	*****	

0	MATRIX/FACTOR DIAGONAL TERMS RATIO SUMMARY TABLE FOR ROTATIONAL DOF SORTED ON DIAGONAL RATIO	GRID POINT ID	DEGREE OF FREEDOM	MATRIX/FACTOR DIAGONAL RATIO	MATRIX DIAGONAL	
				(TOP 1 RATIOS>MAXRAT=	3.35974E+02)	
		58	R2	3.35974E+02	2.14135E+04	

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 is shown in [Figure 10-19](#). (See [MSC_DOC_DIR/doc/linstat/diag.dat](#)) Note that loads from scalar points are not included in the OLOAD RESULTANT output.



		OLOAD	RESULTANT					
0	SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	0.000000E+00	----	----	0.000000E+00	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	0.000000E+00	----	----
		MZ	----	----	----	----	0.000000E+00	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
		FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
0	2	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	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-8.268750E+02	0.000000E+00

Figure 10-19 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 $-5 \times (100 \text{ N}) = -500 \text{ Nm}$ 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 \(Case\)](#) 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 10-20](#), 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	
LOAD VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
6	G	.0	.0	1.000000E+02	.0	.0	.0		
GRAVITY LOADS								SUBCASE 2	
LOAD VECTOR									
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 10-20 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 [SPCFORCES \(Case\)](#) output is requested. It consists of seven lines of output per load case. The SPCFORCE RESULTANT output for the problem diag.dat is shown below:

SPCFORCE RESULTANT									
=0	SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3	
0	1	FX	0.000000E+00	-----	-----	0.000000E+00	0.000000E+00	0.000000E+00	
		FY	-----	0.000000E+00	-----	0.000000E+00	-----	0.000000E+00	
		FZ	-----	-----	-1.000000E+02	0.000000E+00	0.000000E+00	-----	
		MX	-----	-----	-----	0.000000E+00	-----	-----	
		MY	-----	-----	-----	-----	5.000000E+02	-----	
		MZ	-----	-----	-----	-----	-----	0.000000E+00	
0	2	TOTALS	0.000000E+00	0.000000E+00	-1.000000E+02	0.000000E+00	5.000000E+02	0.000000E+00	
		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 10-21 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 10-19](#) and [Figure 10-21](#), 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 10-22 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 10-22). 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).



Equilibrium Checks

A useful tool to ensure the validity and integrity of your finite element analysis solution in static analysis is the equilibrium balance check. The [EQUILIBRIUM \(Case\)](#) Case Control command automates the procedure to produce a summation of the vector resultants (which should be zero if the solution is in equilibrium). This is similar to the steps performed in previous chapters with the individual LOAD, SPCF and MPCF resultants.

Checking the equilibrium conditions at various steps is a simple and effective tool that can be used to detect modeling difficulties and evaluate the quality of the solution. For example, a column of the stiffness matrix of a structure is a set of forces that develop when the degree of freedom represented by that column is given a unit displacement. If the resultant of those forces about any reference point is calculated, it should be zero if no unintentional constraints are imposed. It is also useful to compute and compare the resultants of the applied loads and the constraint forces (single point as well as multi-point). If they balance to zero, confidence in the solution is increased.

The command can be used to specify a grid point to be used as the reference point for the summation. Each subcase can specify a different point, if desired. If the point specified does not exist, the input vector resultants are used without modification. The origin of the basic coordinate system may also be specified as the reference point.

The VECPLOT module performs the tasks associated with generation of the equilibrium results. This option transforms the input force vectors to the basic system and computes the resultants with respect to the grid point specified by the GRDPNT DMAP parameter. If that point does not exist, the origin of the basic coordinate system is used instead. The output is a summary listing of the force resultants and an Equilibrium Check OFP output.

The EQUILIBRIUM command generates a total summation that includes the effects of all available force vectors. These force vectors are the applied loads, single point constraint forces and multi-point/rigid element force vectors. These vectors are produced by the presence of the associated Case Control commands, LOAD (for applied loads), SPCF (for single point constraint forces) and MPCF (for multi-point/RBE constraint forces). If an EQUILIBRIUM command is present in the Case Control Section for a subcase, then the program ensures computation of the SPCF and MPCF for the subcase.

The summary output is as follows:

E Q U I L I B R I U M S U M M A T I O N S I N B A S I C C O O R D I N A T E S Y S T E M								
SUBCASE ID	TYPE	SET ID	T1	T2	T3	R1	R2	R3
1001	LOAD	100	1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	3.000000E+03	-2.000000E+03
	SPCF	200	-1.000000E+03	-3.179752E-13	-4.128382E-12	-6.889597E-12	-3.000000E+03	2.000000E+03
TOTALS WRT POINT		101	0.000000E+00	-3.179752E-13	-4.128382E-12	-6.889597E-12	0.000000E+00	0.000000E+00
3001	LOAD	300	0.000000E+00	0.000000E+00	1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00
	SPCF	200	1.215391E-14	-3.839810E-15	-1.000000E+03	0.000000E+00	-3.646172E-14	3.198743E-14
TOTALS WRT POINT		302	1.215391E-14	-3.839810E-15	0.000000E+00	-1.151943E-14	-3.646172E-14	3.198743E-14

The OFP format is as follows:



E Q U I L I B R I U M C H E C K								
RESULTANT LOADS IN BASIC COORDINATE SYSTEM								
SUBCASE NO.	REFERENCE POINT	LOAD TYPE	T1	T2	T3	R1	R2	R3
1001	101	APP-LOAD F-OF-SPC	1.000000E+03 -1.000000E+03	0.000000E+00 -3.179752E-13	0.000000E+00 -4.128382E-12	0.000000E+00 -6.889597E-12	3.000000E+03 -3.000000E+03	-2.000000E+03 2.000000E+03
		TOTALS	0.000000E+00	-3.179752E-13	-4.128382E-12	-6.889597E-12	0.000000E+00	0.000000E+00
2001	ORIGIN	APP-LOAD F-OF-SPC	0.000000E+00 -2.343780E-29	1.000000E+03 -1.000000E+03	0.000000E+00 -1.013140E-11	-3.000000E+03 3.000000E+03	0.000000E+00 4.052559E-11	4.000000E+03 -4.000000E+03
		TOTALS	-2.343780E-29	0.000000E+00	-1.013140E-11	0.000000E+00	4.052559E-11	0.000000E+00

Output of Maximum, Minimum Grid Point-Based Responses

MAXMIN Survey Output Request (old form)

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 Case Control command, [MAXMIN \(Case\)](#) (old form). See the [MSC Nastran Quick Reference Guide](#) for a complete description of this command.

With PARAM,PRTMAXIM,YES and PARAM,PRTRESLT,YES you can obtain simple surveys of grid point related results output such as displacements, applied loads, and forces of single point constraint, where 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. But with the MAXMIN command there are a lot more controls of the output.

The MAXMIN 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.

The example problem maxmin.dat (see MSC_DOC_DIR/doc/linstat/maxmin.dat) demonstrates some features available with the MAXMIN Case Control command.

The Case Control Section in [Listing 10-3](#) shows the MAXMIN command at line 7 requests print and punch of max/min survey results for all available output quantities generated and 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 in the basic coordinate system frame of reference. 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 forces of single point constraint. A vector magnitude survey is requested 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.

Listing 10-3 Case Control for MAXMIN (Old Form)

```

0          C A S E   C O N T R O L   E C H O
COMMAND
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

```

The MAXMIN survey output results in [Listing 10-4](#) 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.

Listing 10-4 MAXMIN (old form) Displacement Output

MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1							RESULTS FOR SUBCASE 1000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2							RESULTS FOR SUBCASE 1000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1							RESULTS FOR SUBCASE 2000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2							RESULTS FOR SUBCASE 2000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1							RESULTS FOR SUBCOM 3000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
MAXMIN Options: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2							RESULTS FOR SUBCOM 3000		
POINT ID.	TYPE	CID	*** T1 ***	T2	T3	R1	R2	R3	
59	G	BASIC	1.749636E-02	-3.285893E-03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	BASIC	0.000000E+00	-1.046443E-05	-6.331082E-08	0.000000E+00	0.000000E+00	0.000000E+00	
2	G	100	-6.331082E-08	1.046443E-05	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	
51	G	BASIC	0.000000E+00	1.350188E-04	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	

MAXMIN Output of Maximums and Minimums (new form)

The MAXMIN (new form) was designed for “sort2” data recovery such as in Modal Transient solution (SOL 112). However it can be used in statics sort1 to produce max/min output in a different format than old form. The selection method is also different.

The [MAXMIN\(DEF\) \(Case\)](#) Case Control command which must be above the subcase level defines the operations to monitor. There can be multiple MAXMIN(DEF) entries. The [MAXMIN \(Case\)](#) (new form) Case Control commands below the subcase level select the data recovery quantities desired for each subcase.

The output heading is:



M A X / M I N S U M M A R Y (xxxxxx)

where xxxxx describes the class being monitored, such as displacements for grids or element names for stresses, forces or strains.

Using the same problem as maxmin old form the Case Control part of maxmina.dat (see [MSC_DOC_DIR/doc/linstat/maxmina.dat](#)) has changed the DISPlacement output request from the old form to the new form for the t1 and t2 components. (see [Listing 10-5](#) lines 7 and 9). The output (see [Listing 10-6](#)) shows the same displacement data results for subcase 1000 as [Listing 10-4](#) but now the t1/t2 components are in the same listing.

Listing 10-5 Case Control for MAXMIN (new form)

```
C A S E   C O N T R O L   E C H O
COMMAND
COUNT
1   TITLE = MAXMIN CASE CONTROL TEST
2   SUBTITLE = TEST VARIOUS OPTIONS
3   LABEL = COORDINATE SYSTEMS WITH SETS
4   SPC=100
5   SET 100 = 2,51,59
6   DISP=100
7   MAXMIN(DEF) DISP T1 T2
8   SUBCASE 1000
9   MAXMIN GRID = 100 $
10  LOAD=1000
11  SUBCASE 2000
12  LOAD=1000
13  BEGIN BULK
```

Listing 10-6 MAXMIN Displacement Output (new form)

COORDINATE SYSTEMS WITH SETS						SUBCASE 1000
M A X / M I N S U M M A R Y (DISPLACEMENTS)						
POINT	CID	OPERATION	COMPONENT	SUBCASE	VALUE	
59	0	MAX	1	T1	1000 1.749636E-02	
2	0		2		1000 0.000000E+00	
51	0		3		1000 0.000000E+00	
2	0	MIN	1	T1	1000 0.000000E+00	
51	0		2		1000 0.000000E+00	
59	0		3		1000 1.749636E-02	
59	0	ABS	1	T1	1000 1.749636E-02	
2	0		2		1000 0.000000E+00	
51	0		3		1000 0.000000E+00	
51	0	MAX	1	T2	1000 1.350188E-04	
2	0		2		1000 -1.046443E-05	
59	0		3		1000 -3.285893E-03	
59	0	MIN	1	T2	1000 -3.285893E-03	
2	0		2		1000 -1.046443E-05	
51	0		3		1000 1.350188E-04	
59	0	ABS	1	T2	1000 3.285893E-03	
51	0		2		1000 1.350188E-04	
2	0		3		1000 1.046443E-05	

As an example of element output the Case Control part of mmq4es.dat (see [MSC_DOC_DIR/doc/linstat/mmq4es.dat](#)) is shown. In [Listing 10-7](#) lines 3 and 9 show the request for quad4 element stress output in subcase 1001. The component numbers 9 and 17 were used instead of the NDDL names because these components names changed from SMAX to TMAX, but the numbers are the same.

Listing 10-7 MAXMIN Case Control (new form)



```
CASE      CONTROL      ECHO
COMMAND
1   $MAXMIN(DEF) STRESS QUAD4 SMAX1 SMAX2 MAXA MINA FULL
2   $ VERSION INDEPENDENT REQUEST
3   MAXMIN(DEF) STRESS QUAD4    9     17 MAXA MINA FULL
4   ECHO=NONE
5   SPC=1
6   SUBCASE 1001
7   TITLE=ONLY MAXMIN SHEAR INVARIANT, STRCUR
8   LOAD=101
9   MAXMIN(ELEM,SHEAR) = ALL
10  BEGIN BULK
```

The [Listing 10-8](#) shows the max/min summary requested plus the normal quad4 stress output. Note the element reference system may not be consistent, so it is suggested that the invariant be used to determine the ordering for two dimensional elements. In the results presented the SHEAR invariant was selected.



Listing 10-8 MAXMIN QUAD4 Stress Output and Normal Stress Output (new form)

MAX / MIN SUMMARY (QUAD4)						SUBCASE 1001
ELEMENT	CID	OPERATION	COMPONENT	SUBCASE	VALUE	
37	-1	MAX	1	TMAX1	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	TMAX1	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03
37	-1	MAX	1	TMAX2	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	TMAX2	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03

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)						SUBCASE 1001
ELEMENT	FIBER ID.	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)	MAX SHEAR
		NORMAL-X	NORMAL-Y	SHEAR-XY		
13	-5.00000E-02	-3.104902E+02	-1.519392E+03	-1.588575E+03	-34.5841 7.847449E+02 -2.614627E+03	1.699686E+03
	5.00000E-02	-3.104902E+02	-1.519392E+03	-1.588575E+03	-34.5841 7.847449E+02 -2.614627E+03	1.699686E+03
14	-5.00000E-02	2.803682E+02	4.815094E+02	-4.061530E+03	-45.7092 4.443714E+03 -3.681835E+03	4.062775E+03
	5.00000E-02	2.803682E+02	4.815094E+02	-4.061530E+03	-45.7092 4.443714E+03 -3.681835E+03	4.062775E+03
25	-5.00000E-02	1.260785E+03	2.108877E+04	-1.582418E+03	-85.4656 2.121426E+04 1.135291E+03	1.003949E+04
	5.00000E-02	1.260785E+03	2.108877E+04	-1.582418E+03	-85.4656 2.121426E+04 1.135291E+03	1.003949E+04
26	-5.00000E-02	2.543391E+03	1.722717E+04	-4.047183E+03	-75.5672 1.826878E+04 1.501783E+03	8.383499E+03
	5.00000E-02	2.543391E+03	1.722717E+04	-4.047183E+03	-75.5672 1.826878E+04 1.501783E+03	8.383499E+03
31	-5.00000E-02	1.224244E+03	2.709546E+04	-9.584824E+02	-87.8812 2.713093E+04 1.188782E+03	1.297107E+04
	5.00000E-02	1.224244E+03	2.709546E+04	-9.584824E+02	-87.8812 2.713093E+04 1.188782E+03	1.297107E+04
32	-5.00000E-02	2.949327E+03	2.166685E+04	-2.462046E+03	-82.6305 2.198529E+04 2.630894E+03	9.677195E+03
	5.00000E-02	2.949327E+03	2.166685E+04	-2.462046E+03	-82.6305 2.198529E+04 2.630894E+03	9.677195E+03
37	-5.00000E-02	1.340506E+03	2.942988E+04	-3.323971E+02	-89.3221 2.943381E+04 1.336573E+03	1.404862E+04
	5.00000E-02	1.340506E+03	2.942988E+04	-3.323971E+02	-89.3221 2.943381E+04 1.336573E+03	1.404862E+04
201	-5.00000E-02	-3.406158E+02	-5.742985E+03	-8.588726E+02	-8.8193 -2.073585E+02 -5.876242E+03	2.834442E+03
	5.00000E-02	-3.406158E+02	-5.742985E+03	-8.588726E+02	-8.8193 -2.073585E+02 -5.876242E+03	2.834442E+03
301	-5.00000E-02	9.549748E+02	-2.552619E+03	-1.202636E+03	-17.2199 1.327710E+03 -2.925354E+03	2.126532E+03
	5.00000E-02	9.549748E+02	-2.552619E+03	-1.202636E+03	-17.2199 1.327710E+03 -2.925354E+03	2.126532E+03
401	-5.00000E-02	3.927623E+03	-5.305154E+02	-1.307955E+03	-15.2016 4.283025E+03 -8.859182E+02	2.584472E+03
	5.00000E-02	3.927623E+03	-5.305154E+02	-1.307955E+03	-15.2016 4.283025E+03 -8.859182E+02	2.584472E+03

The FULL selection on the MAXMIN(DEF) entry retained the normal element stresses for the selected set.

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](#) 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.

Unintentional Grounding (GROUNDCHECK)

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 requested with the [GROUNDCHECK \(Case\)](#) Case Control command.

$$\text{GROUNDCHECK} \left[\begin{array}{l} \left[\begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], \text{PUNCH}, \text{SET} = \left\{ \begin{array}{l} \left[\begin{array}{l} \text{G}, \text{N}, \text{N + AUTOSPC}, \text{F}, \text{A} \end{array} \right] \\ \text{ALL} \end{array} \right\} \\ \text{GRID} = \text{gid}, \text{THRESH} = \text{e}, \text{DATAREC} = \left[\begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], (\text{RTHRESH} = \text{r}) \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

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 [Sample Output Listings](#) (App. 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	1	G R O U N D C H E C K F O R C E S (F - S E T)						
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, MPC forces, rigid elements, general elements, DMIG entries and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads) 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	DMI Forces
SPC Forces	Boundary Loads from Upstream Superelements
Element Elastic Forces	Forces on elements with geometric or material nonlinear properties during normal modes analysis in SOL 106; also called linear perturbation analysis.
GENEL Forces	
DMIG Referenced by K2GG Case Control command	
Thermal Loads	
MPC and Rigid Element Forces	

The model in [Figure 10-23](#) 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 partial input file is shown in [Listing 10-9](#). For the incorrect PBAR see MSC_DOC_DIR/doc/linstat/q4bargpf.dat and for the correct PBAR, see MSC_DOC_DIR/doc/linstat/q4bargf.dat.

Example



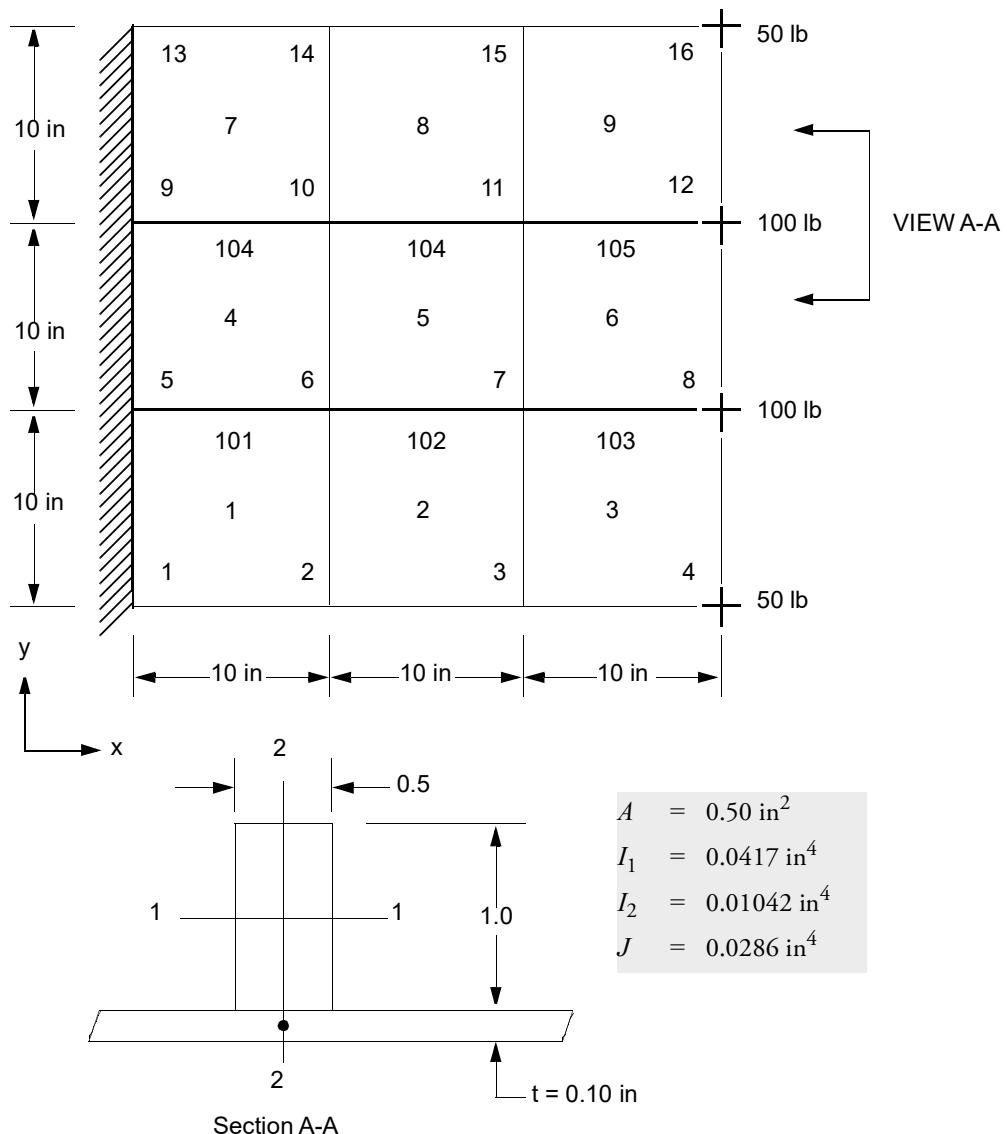


Figure 10-23 Clamped Plate with Stiffeners

Listing 10-9 Input File for the Demonstration of Grid Point Force Balance



```
$  
$ Incorrect PBAR PROPERTIES with A and I1 missing from q4bargpf.dat  
$  
PBAR    100      1      .50      .0417     .01042     .0286          +P100  
+P100    .5      .25      -.5      .25      -.5      -.25      .5      -.25  
PBAR    200      1           .01042     .0286          +P200  
$  
$ CORRECT PBAR PROPERTIES from q4bargpfc.dat  
$  
PBAR    200      1      .50      .0417     .01042     .0286          +P200  
+P200    .5      .25      -.5      .25      -.5      -.25      .5      -.25  
$
```

If everything is done correctly, expect the stress contour plot to look something like [Figure 10-24\(a\)](#). However, say that you made a modeling mistake somewhere, and the stress contour plot is coming out as shown in [Figure 10-24\(b\)](#). 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 10-25](#)), 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 as they are all zero.

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 10-26](#).



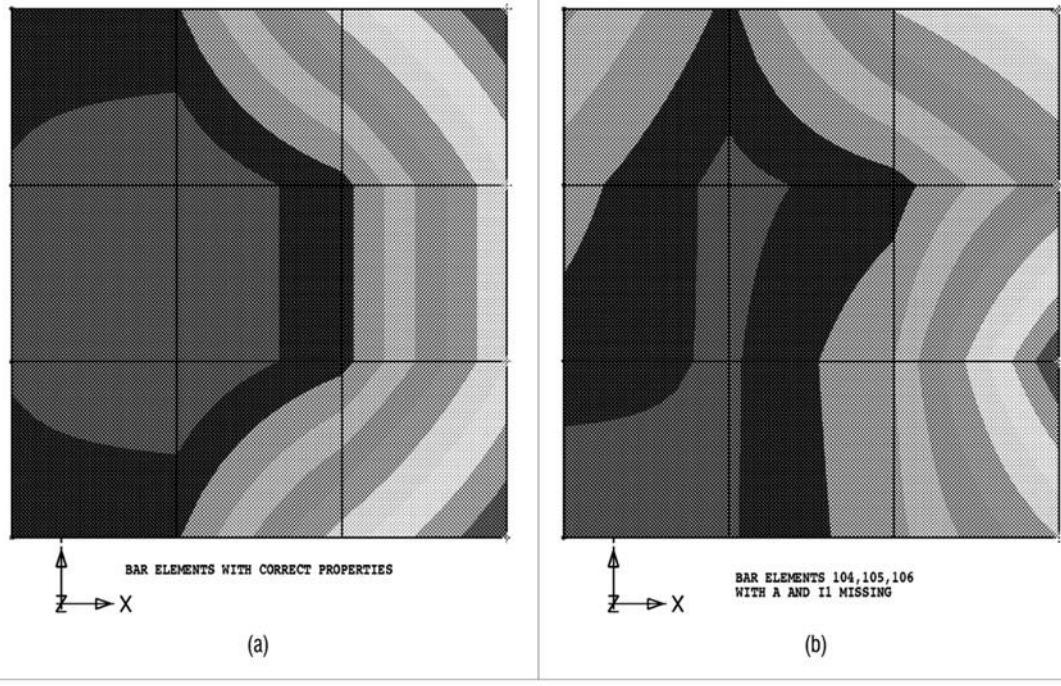


Figure 10-24 Stress Contour Plots of a Cantilever Plate

			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	1	F-OF-SPC	2.997335E+03	9.717916E+02	1.590328E+01	1.142525E+02	1.093198E+01	5.376202E-01	
1	1	QUAD4	-2.997335E+03	-9.717916E+02	-1.590328E+01	-1.142525E+02	-1.093198E+01	-5.376202E-01	
1		*TOTALS*	0.0	0.0	0.0	0.0	0.0	0.0	
9	9	F-OF-SPC	2.160733E+03	-7.640769E+02	4.267091E+01	5.077877E+02	-2.380696E+02	-1.310570E+02	
9	9	BAR	0.0	1.723920E+01	0.0	-7.145220E+02	0.0	1.310371E+02	
9	4	QUAD4	-1.822819E+03	6.794191E+02	-5.519084E+01	1.680608E+02	1.512287E+02	9.956017E-03	
9	7	QUAD4	-3.379135E+02	6.741856E+01	1.251993E+01	3.867351E+01	8.684086E+01	9.956017E-03	
9		*TOTALS*	-3.410605E-13	9.947598E-14	0.0	1.278977E-13	2.273737E-13	7.077672E-15	
10	104	BAR	0.0	-1.723920E+01	0.0	7.145220E+02	0.0	4.135498E+01	
10	105	BAR	0.0	7.436023E+00	0.0	-9.469761E+02	0.0	2.091970E+01	
10	4	QUAD4	1.218599E+03	-1.024201E+02	6.044593E+01	-1.481439E+02	4.891066E+01	-1.733289E+01	
10	5	QUAD4	-1.525199E+03	3.808516E+02	-6.905392E+01	3.982679E+02	-4.746482E+01	-1.783502E+01	
10	7	QUAD4	6.805377E+02	-2.630925E+02	-1.466520E+01	-7.246335E+01	-5.549834E+01	-1.685227E+01	
10	8	QUAD4	-3.739372E+02	-5.535785E+00	2.327319E+01	5.479342E+01	5.405249E+01	-1.735440E+01	
10		*TOTALS*	-1.193712E-12	8.171241E-13	-5.684342E-14	2.842171E-13	-4.092726E-12	2.096101E-13	
11	105	BAR	0.0	-7.436023E+00	0.0	9.469761E+02	0.0	4.634054E+01	
11	106	BAR	0.0	3.803732E+00	0.0	-7.731555E+02	0.0	1.268509E+01	
11	5	QUAD4	5.258212E+02	-5.352766E+02	9.303661E+01	-4.455990E+02	-8.730401E+01	-1.476272E+01	
11	6	QUAD4	-8.517637E+02	6.218045E+02	-8.716542E+01	4.214756E+02	3.996059E+01	-1.509897E+01	
11	8	QUAD4	4.494631E+02	-7.975973E+01	-3.114117E+01	-1.096983E+02	-2.945120E-01	-1.441384E+01	
11	9	QUAD4	-1.235206E+02	-3.135966E+00	2.526999E+01	-3.999883E+01	4.763793E+01	-1.475009E+01	
11		*TOTALS*	2.131628E-12	1.051603E-12	1.818989E-12	1.449507E-12	-3.865352E-12	-9.592327E-14	
16	9	APP-LOAD	0.0	0.0	-5.000000E+01	0.0	0.0	0.0	
16	9	QUAD4	2.131628E-14	-5.119516E-13	5.000000E+01	-2.247091E-13	1.364242E-12	1.110223E-16	
16		*TOTALS*	2.131628E-14	-5.119516E-13	7.958079E-13	-2.247091E-13	1.364242E-12	1.110223E-16	



Figure 10-25 Grid Point Force Balance Output for a Model with Incorrect CBAR Properties

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	1.709574E+03	6.298175E+02	1.949318E+01	8.553487E+01	-2.336222E+01	2.775507E-01		
1	1	QUAD4	-1.709574E+03	-6.298175E+02	-1.949318E+01	-8.553487E+01	2.336222E+01	-2.775507E-01		
1		*TOTALS*	0.0	0.0	0.0	0.0	0.0	0.0		
9		F-OF-SPC	-1.709574E+03	-4.253813E+02	1.305068E+02	1.778664E+02	-4.476638E+03	-5.384993E+01		
9	104	BAR	4.888152E+03	7.378169E+00	-1.451928E+02	-2.720984E+02	4.428308E+03	5.364330E+01		
9	4	QUAD4	-1.887228E+03	3.867914E+02	-1.594250E+00	1.111256E+01	2.245081E+01	1.033142E-01		
9	7	QUAD4	-1.291353E+03	3.121163E+01	1.628026E+01	8.311946E+01	2.587908E+01	1.033142E-01		
9		*TOTALS*	4.547474E-13	5.684342E-14	-1.421085E-13	-5.684342E-14	-7.389644E-13	7.771561E-16		
10	104	BAR	-4.888152E+03	-7.378169E+00	1.451928E+02	2.720984E+02	-2.976380E+03	2.013839E+01		
10	105	BAR	2.832370E+03	2.373930E+00	-1.454942E+02	-2.720984E+02	2.889182E+03	5.043980E+00		
10	4	QUAD4	1.887228E+03	2.083111E+02	1.594250E+00	1.788126E+01	-6.508303E+00	-6.080230E+00		
10	5	QUAD4	-1.127134E+03	-1.181978E+02	-1.597559E+00	3.297124E+01	2.504559E+01	-6.230502E+00		
10	7	QUAD4	1.952959E+03	-4.784751E+02	-1.915442E+01	-9.613683E+01	-2.857046E+00	-6.360682E+00		
10	8	QUAD4	-6.572677E+02	3.933660E+02	1.945910E+01	7.338474E+01	7.151707E+01	-6.510954E+00		
10		*TOTALS*	4.092726E-12	2.842171E-13	3.268497E-13	8.526513E-14	6.991741E-12	6.838974E-14		
11	105	BAR	-2.832370E+03	-2.373930E+00	1.454942E+02	3.001988E+02	-1.434240E+03	1.869532E+01		
11	106	BAR	8.832609E+02	2.753771E-01	-1.431433E+02	-1.767170E+02	1.377428E+03	-2.577389E+00		
11	5	QUAD4	1.127134E+03	-3.581044E+03	1.597559E+00	4.051904E+01	-9.069969E+00	-3.711060E+00		
11	6	QUAD4	-3.689837E+02	1.436027E+02	-1.571746E+00	5.289693E+01	2.615439E+01	-3.852338E+00		
11	8	QUAD4	1.234270E+03	-2.735811E+02	-2.937496E+01	-1.653973E+02	1.576359E+01	-4.206628E+00		
11	9	QUAD4	-4.331070E+01	1.678873E+02	2.699822E+01	-5.150050E+01	2.396459E+01	-4.347906E+00		
11		*TOTALS*	-3.964828E-12	-9.094947E-13	-5.826450E-13	-1.563194E-13	8.469669E-12	5.062617E-14		
12		APP-LOAD	0.0	0.0	-1.000000E+02	0.0	0.0	0.0		
12	106	BAR	-8.832609E+02	-2.753771E-01	1.431433E+02	1.767170E+02	5.400519E+01	5.331160E+00		
12	6	QUAD4	3.689837E+02	4.705017E+02	1.571746E+00	5.735566E+01	-1.043693E+01	-2.392765E+00		
12	9	QUAD4	5.142771E+02	-4.702263E+02	-4.471503E+01	-2.340727E+02	-4.356825E+01	-2.938395E+00		
12		*TOTALS*	1.591616E-12	1.705303E-13	-1.307399E-12	-4.547474E-13	-1.193712E-12	1.789680E-13		
16		APP-LOAD	0.0	0.0	-5.000000E+01	0.0	0.0	0.0		
16	9	QUAD4	1.437073E-12	5.099446E-13	5.000000E+01	6.936673E-13	1.364242E-12	-5.551115E-16		
16		*TOTALS*	1.437073E-12	5.099446E-13	1.136868E-13	6.936673E-13	1.364242E-12	-5.551115E-16		

Figure 10-26 Grid Point Force Balance Output for a Model with Correct CBAR Properties

Both Figure 10-25 and Figure 10-26 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.

Stress Error Estimators

Grid Point Stresses, 365 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 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.



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.

The element and grid point stress discontinuity output can only be obtained if the grid point stress output is requested via the [STRFIELD \(Case\)](#) 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 \(Case\)](#) 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. Element stresses (STRESS) 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](#).

Then the [GPSDCON \(Case\)](#) Case Control command requests grid point stress discontinuity output and/or [ELSDCON \(Case\)](#) requests element stress discontinuity output.

The following remarks apply to the use of the GPSDCON and/or ELSDCON commands.

1. They are only supported in Solution Sequence101.
2. They 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.
3. These commands can produce a substantial amount of data.
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.

Grid Point Stress Discontinuities

To illustrate the discontinuities requests, 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).



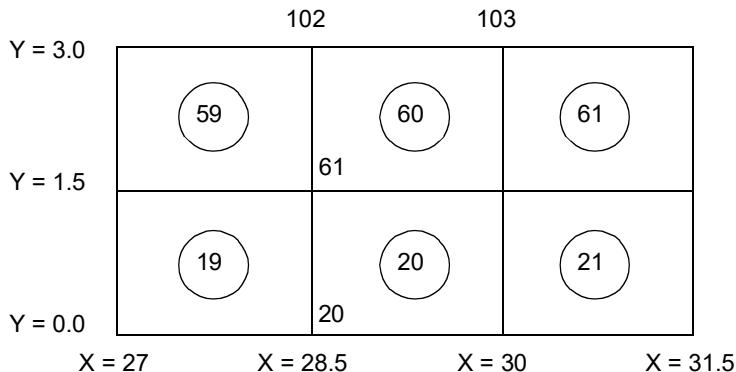


Figure 10-27 Partial Model for the Stress Discontinuity Calculations

The example problem in [Listing 10-10](#) (See MSC_DOC_DIR/doc/linstat/q4sdcon.dat) contains the full list of Case Control commands required to obtain Grid Point/Element Stress Discontinuity output. Not shown is example problem q4sdconc.dat (see MSC_DOC_DIR/doc/linstat/q4sdconc.dat) which requests corner stresses for the CQUAD4 elements.



Listing 10-10 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  
$  
STRESS = 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
```

The data in [Table 10-3](#) and [Table 10-5](#) is extracted from the MSC Nastran output for Grid points 20 and 61 for both examples. See [Mesh Stress Discontinuities at Grid Points](#) of the MSC Nastran Reference Guide for a discussion of equations for the error estimate.



Table 10-3 The Values of δ_g and ERROR ESTIMATE for CQUAD4 Element Grid 20
(X = 28.5, Y = 0.0)

Table 10-4

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL--Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	9.304E+02	1.927E-08	4.419E+01	5.378E+02
CQUAD4 (CORNER)	8.839E+01	1.989E+01	0.0	5.231E+01

Table 10-5 The Values of δ_g and ERROR ESTIMATE for CQUAD4 Element Grid 61
(X = 28.5, Y = 1.5)

Table 10-6

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL-Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	6.572E+02	1.330E-08	3.125E+01	6.436E+02
CQUAD4 (CORNER)	3.125E+01	1.048E+01	3.125E+01	2.622E+01

The following remarks should be noted in regard to [Table 10-3](#) and [Table 10-5](#).

1. By default, the stress components for the CQUAD4 elements are only output at the element centroids.
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 the tables
3. 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.

Smaller values of δ_g and the ERROR ESTIMATE are obtained from the corner stress request. In addition to values of δ_g for each stress component and the ERROR ESTIMATE 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_e that are computed for each of the N_g connected vertex grid points. This computation is done for each stress component for all of the elements of interest. (The results (δ_e and ERROR ESTIMATE) from MSC Nastran for the same examples, using CQUAD4 and CQUAD4 with the corner option for elements 20 and 60, are summarized in [Table 10-7](#) and [Table 10-9](#).

Table 10-7 The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types

Table 10-8

Element Type	Location at EID = 20			
	NORMAL-X δ_e	NORMAL-Y (δ_e)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	1.285E+03	1.998E-08	6.250E+01	7.428E+02
CQUAD4 (CORNER)	9.882E+01	2.813E+01	4.419E+01	6.457E+01

Table 10-9 The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types

Table 10-10

Element Type	Location at EID = 60			
	NORMAL-X (δ_e)	NORMAL-Y (δ_e)	SHEAR-XY (δ_e)	ERROR ESTIMATE
CQUAD4	1.282E+03	1.900E-08	6.250E+01	1.057E+03
CQUAD4 (CORNER)	4.419E+01	9.375E+00	4.419E+01	3.649E+01

In addition to values of (δ_e) for each stress component and the ERROR ESTIMATE, MSC Nastran output also contains values of 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 10-3](#) through [Table 10-9](#) 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.

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 10-28](#) illustrates the importance of consistent direction. The stresses in a CQUAD4 element are in terms of the element coordinate system (see [MSC Nastran Elements](#) and the *MSC Nastran Quick Reference Guide*).

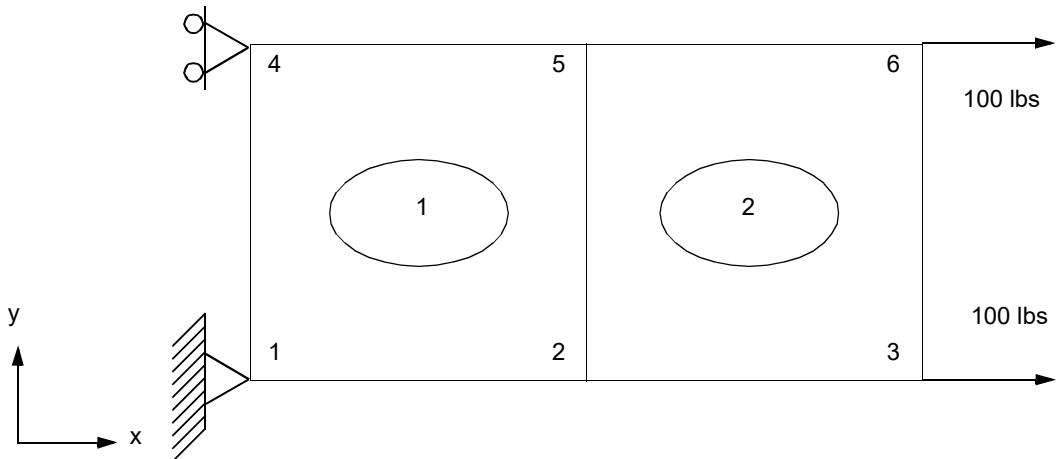


Figure 10-28 CQUAD4 Model



The complete input file used for this problem is shown in [Listing 10-11](#). (See `MSC_DOC_DIR/doc/linstat/q4consc.dat`).

Listing 10-11 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  

$  

$ THIS SECTION CONTAINS ELEMENT CONNECTIVITIES  

$  

CQUAD4  1          10     1       2       5       4  

CQUAD4  2          10     2       3       6       5       5  

$  

$CQUAD4  2          10     3       100.   1.  

$  

$ 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  

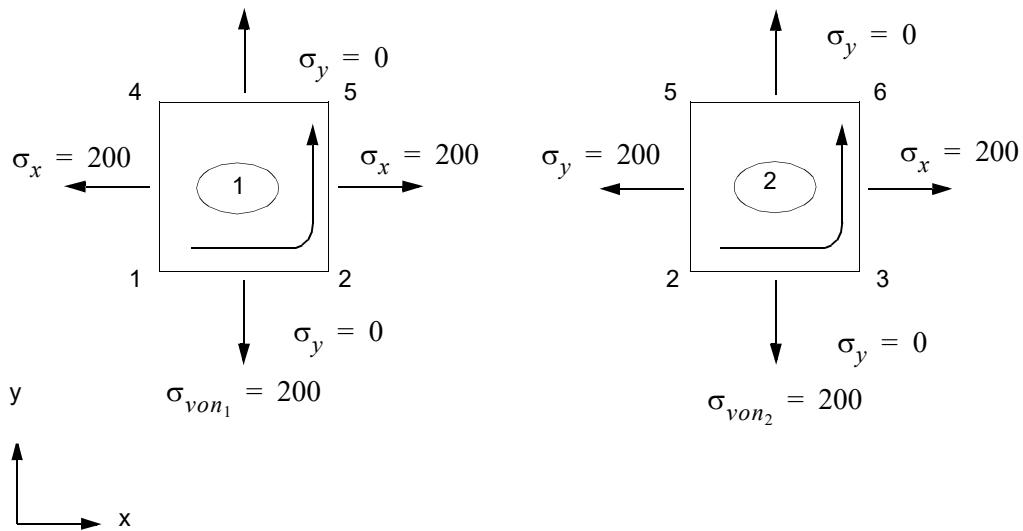
$  

ENDDATA

```



In this example, the connectivities for both CQUAD4 elements are described in a consistent manner as shown in [Figure 10-29](#). 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 10-29](#) 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 10-30](#), then the corresponding component stresses (e.g. σ_x) do not necessarily point in the same directions for both elements. As shown in [Figure 10-30](#), the σ_x component stresses point in the directions of the x- and y-axis for elements 1 and 2, respectively.



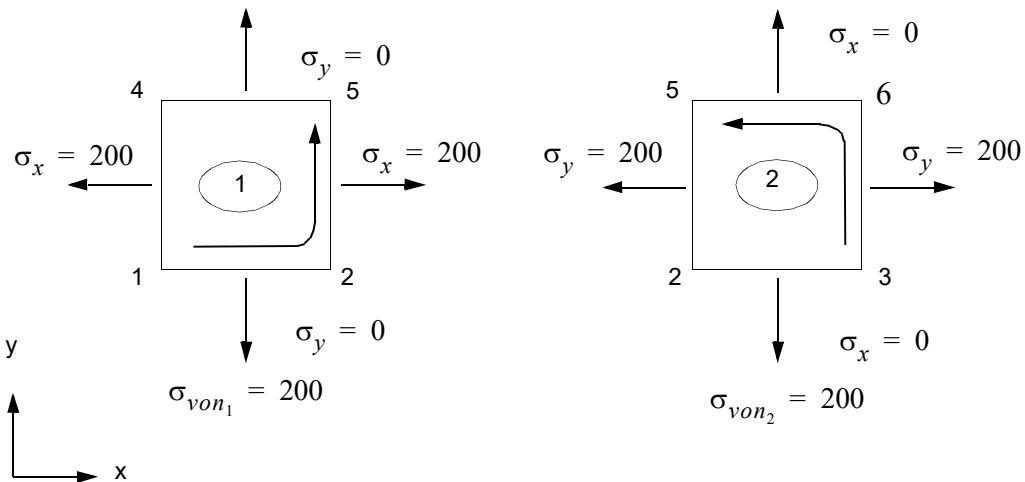


Figure 10-30 Inconsistent Connectivity

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 10-30 is identical to Listing 10-11 with the exception of the shaded CQUAD4 entry. In other words, replace the following CQUAD4 entry:

CQUAD4

1	2	3	4	5	6	7	8	9	10
CQUAD4	2	10	2	3	6	5			

with the following:

1	2	3	4	5	6	7	8	9	10
CQUAD4	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](#). 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](#).

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](#).

Monitor Points

Introduction

Monitor Points is a generic term used to refer to four different user input quantities that the user can select to provide a snapshot of the results of a structural analysis that is particularly meaningful to the engineering end-user. While Monitor Points were developed with aeroelastic considerations in mind, they have applicability in any type of analysis. This guide will focus on Sol 101, but it is also available in 103,108,109,111,112,144, 146 and 200. Monitor Points refer to the MONITOR Case Control command and to the Bulk Data entries shown below. They can be used to obtain an understanding of critical load paths across the subcases. The user can recover results at a particular location and output coordinate system of his choosing.

The coordinate system the results are most conveniently expressed in, may not be the same as those used in computing the monitor point results, so the output coordinate system on the MONPNT1, MONPNT3 and MONDPS1 entries facilitates communication among groups and makes the results consistent.



Also the ability to transfer moments allows the specification of a monitor point location that is apart from the finite element model and thereby facilitates communications between the loads group and the stress group in the aircraft development process.

User Interface

The MSC Nastran user interface starts with the **MONITOR (Case)** Case Control command, which provides control over the printing of the monitor points results. Toggles can be used to control the print of the individual monitor types. This command must be placed above the subcase level or in the first subcase and applies to all subcases.

The **MONPNT1** Bulk Data entry provides integrated loads at a user defined point in a user defined coordinate system. The user also identifies the nodes whose loads are to be integrated. The user can then extract the applied loading for the specified set of structural nodes. This enables the batch calculation of VMT (shear, moment and torque) data.

Note that the loads are specified by the COMP field that, in turn, identifies a AECOMP entry that selects SET1 entry for structural grids.

The **MONPNT2** Bulk Data entry provides element results from the TABLEs Stress, Strain, or Force in a tabulated fashion. This can be used to pinpoint a particular response for output, as opposed to finding a particular item in a large OFP listing. For the results to be accurate, the term selected must be a linear function of the displacements.

The user must identify the element TYPE and NDDL item. The type and nndl item is obtained from the nndl description for each table. See QRG Remark 5 of this entry. There are separate types for composite and element corner results.

The **MONPNT3** Bulk Data entry provides a summation of grid point forces at user specified integration points and in a user defined coordinate system. The summation of the internal loads is useful in calculating resultant forces at a cut in the structure. This can be used to provide the net load acting at a fuselage or wing station by making a “cut” in the structure and then identifying all the grids and elements on one side of the cut

The entry NAME is identified with a piece of structure by listing the elements and nodes associated with it. The grid point force data associated with these entities is then integrated to the location specified on the entry. The XFLAG can be used to exclude certain grid point force types from consideration.

The **MONDSP1** Bulk Data entry allows for the sampling of a displacement vector to create a blended displacement response at a user specified point and coordinate system. The displacement monitor point is essentially an RBE3 element (limited to a single dependent point) and the dependent grid is now an arbitrary point. The averaged displacement can be seen as providing a qualitative assessment of the elastic deflection of a vehicle. An example is to monitor the nominal pitch and plunge at a station along the wing.

This entry is similar to the MONPNT1 entry but now it is displacements that are being averaged. As with the MONPNT1, the COMP field, points to a AECOMP entry which specifies a SET1 entry or entries that specify the grids to be monitored.

The **MONSUM** Bulk Data entry defines a new monitor result that is the weighted sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type.



This entry can be used for both updating and summing of monitor point results. It allows for the modification of existing component results from MONDSP1, MONPNT1 and MONPNT3. A scalar multiple can provide, for example, a change in sign or a change in units. Enabling the weighted summation of two or more MONDSP1, MONPNT1 or MONPNT3's, that are of the same type, can provide the ability to present running results along a wing or fuselage.

The **MONSUM1** Bulk data entry specifies the location of the summed quantity. This enables the summation of monitor points from disparate points to a single location.

The first continuation defines a CP,X,Y,Z, CD combination that specifies the point where the summation is said to occur and performs the summation using the equation provided below. By contrast, the MONSUM does not specify the location and therefore cannot be used in subsequent MONSUMT processing.

The MONSUM1 does not support the combination of MONDSP1's since no physically meaningful interpretation or application can be envisioned for combining MONDSP1's in this manner.

The underlying equation that is executed for the MONSUM1 is:

$$MONSUM1_j = \sum_i^n COEF_{ij} MR_i$$

Where MR_i is the result from the individual component.

Relative to the MONSUM, the MONSUM1 allows the user to specify where the summed output is requested. MSC Nastran uses this information plus the locations of the referenced monitor points that are being summed to perform a coordinate transformation as part of the summation.

The **MONSUMT** Bulk data entry provides the ability to transfer moments and thereby allows the specification of a monitor point location that is apart from the finite element model. This facilitates communication between the loads group and the stress group in the development/simulation process.

With the MONSUMT entry moment transfers do occur. The first continuation defines a CP,X,Y,Z, CD combination that specifies the point where the loads are to be monitored and performs the summation/transfer using the equation provided below. By contrast, the MONSUM1 location is user specified and does not involve moment transfer.

The MONSUMT like MONSUM1 does not support the combination of MONDSP1's since no physically meaningful interpretation or application can be envisioned for combining MONDSP1's in this manner.

The MONSUMT can reference the results of another MONSUMT or MONSUM1 as long as there is not a circular reference.

The underlying equation that is executed for the MONSUMT is:

$$MONSUMT_j = \sum_i^n T_{ji} MR_i$$

Where T_{ji} is a partial rigid body vector for the location of the monitor points being summed and MR_i is the result from the individual component.



Example 1

The output in [Listing 10-12](#) shows part of the output from all the types of monitor point requests in SOL 101 using the example problem /Install/doc/linstat/monitor101.dat.



Listing 10-12

```

S T R U C T U R A L   I N T E R N A L   M O N I T O R   P O I N T   L O A D S (MOPNPT2)

MONITOR POINT NAME = MOPNPT2

      VALUE      EID      ETYPE      CLASS      RESPONSE      LABEL      SUBCASE NO.
-----+-----+-----+-----+-----+-----+
  1.504374E+00  1100081  QUAD4    STRESS    SY1      MPT21  THIS IS A STRESS-MOPNPT2 MONITOR POINT      1

MONITOR POINT NAME = MOPNPT2

      VALUE      EID      ETYPE      CLASS      RESPONSE      LABEL      SUBCASE NO.
-----+-----+-----+-----+-----+
  3.008748E+00  1100081  QUAD4    STRESS    SY1      MPT21  THIS IS A STRESS-MOPNPT2 MONITOR POINT      3

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

MONITOR POINT NAME = DISP2      COMPONENT = PLATE3      CLASS = GENERAL      SUBCASE NO.      1
LABEL = THIS IS A DISPLACEMENT MONITOR POINT
CP = 0          X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS  ELASTIC REST.
-----
  CX  0.000000E+00
  CY  0.000000E+00
  CZ  1.327989E-05
  CMX -7.265083E-07
  CMY -8.295716E-08
  CMZ 0.000000E+00

MONITOR POINT NAME = DISP2      COMPONENT = PLATE3      CLASS = GENERAL      SUBCASE NO.      3
LABEL = THIS IS A DISPLACEMENT MONITOR POINT
CP = 0          X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS  ELASTIC REST.
-----
  CX  0.000000E+00
  CY  0.000000E+00
  CZ  2.655978E-05
  CMX -1.453017E-06
  CMY -1.659143E-07
  CMZ 0.000000E+00

S T R U C T U R A L   M O N I T O R   P O I N T   I N T E G R A T E D   L O A D S (MOPNPT1)

MONITOR POINT NAME = MPT11      COMPONENT = PLATE1      CLASS = GENERAL      SUBCASE NO.      1
LABEL = THIS IS A LOAD MONITOR POINT
CP = 52          X = 1.00000E+00      Y = 2.00000E+00      Z = 3.00000E+00      CD = 52

  AXIS  RIGID APPLIED
-----
  CX  6.531973E+00
  CY  4.618802E+00
  CZ  0.000000E+00
  CMX -1.880346E+01
  CMY 2.659210E+01
  CMZ -4.812340E+01

MONITOR POINT NAME = SCALE      COMPONENT = **SUM**      CLASS = GENERAL      SUBCASE NO.      1
LABEL = MULTIPLY MPT11 BY 2.0
CP = 0          X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS  RIGID APPLIED
-----
  CX  1.306395E+01
  CY  9.237604E+00
  CZ  0.000000E+00
  CMX -3.760691E+01
  CMY 5.318421E+01
  CMZ -9.624680E+01

MONITOR POINT NAME = MPT11      COMPONENT = PLATE1      CLASS = GENERAL      SUBCASE NO.      3
LABEL = THIS IS A LOAD MONITOR POINT
CP = 52          X = 1.00000E+00      Y = 2.00000E+00      Z = 3.00000E+00      CD = 52

  AXIS  RIGID APPLIED
-----
  CX  1.306395E+01
  CY  9.237604E+00
  CZ  0.000000E+00
  CMX -3.760691E+01
  CMY 5.318421E+01
  CMZ -9.624680E+01

```



```

MONITOR POINT NAME = SCALE           COMPONENT = **SUM**          CLASS = GENERAL          SUBCASE NO.   3
LABEL = MULTIPLY MPT11 BY 2.0      CP = 0             X = 0.00000E+00       Y = 0.00000E+00       Z = 0.00000E+00      CD = 0
AXIS   RIGID APPLIED
-----
CX    2.612789E+01
CY    1.847521E+01
CMX   -7.521383E+01

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS (MONPNT3)

MONITOR POINT NAME = MPT31          SUBCASE NO.   1
LABEL = THIS IS A FREE BODY MONITOR POINT
CP = 52               X = 1.00000E+00     Y = 2.00000E+00     Z = 3.00000E+00

AXIS   REST. APPLIED
-----
CX    2.433662E-01
CY    1.720859E-01
CZ    0.000000E+00
CMX   -2.460616E+00
CMY   3.479837E+00
CMZ   -1.559533E+00

MONITOR POINT NAME = MPT31          SUBCASE NO.   3
LABEL = THIS IS A FREE BODY MONITOR POINT
CP = 52               X = 1.00000E+00     Y = 2.00000E+00     Z = 3.00000E+00

AXIS   REST. APPLIED
-----
CX    4.867325E-01
CY    3.441719E-01
CZ    0.000000E+00
CMX   -4.921232E+00
CMY   6.959673E+00
CMZ   -3.119067E+00

```

MONPNT2 produces a line of data for each subcase. The rest contain as many rows as the number of components specified in the AXES field. Be sure to use quantities that are a linear function of the solutions. This is not the case for synthesized responses such as von Mises stress. The MONSUM just multiplied components 1,2 and 4 of the MONPNT1 values by 2.0.

Example 2

The MSC Nastran TPL has a subdirectory “monsum1” that contains a series of simple examples that apply the MONSUM1 and MONSUMT entries across a variety of solution sequences.

A very simple example is monsum1101. This is a cantilevered beam example where the beam is of length 10 units and has a load of magnitude 10.0 applied at the half span. the example includes a MONSUM a MONSUM1 and MONSUMT, all of which are applied to one MONPNT1 that is calculated at the half span and another that is at the tip.

The results are shown below.



Listing 10-13

```

STRUCTURAL MONITOR POINT INTEGRATED LOADS(MONPNT1)
MONITOR POINT NAME = DALE          COMPONENT = **SUM**          CLASS = GENERAL          SUBCASE NO.      1
LABEL = A REALLY REALLY SIMPLE MONSUM
CP,X,Y,Z AND CD ARE NOT DEFINED FOR MONITOR RESULTS CREATED FROM A MONSUM

AXIS RIGID APPLIED
-----
CX 0.000000E+00
CY 0.000000E+00
CZ 2.000000E+02
CMX 0.000000E+00
CMY 5.000000E+02
CMZ 0.000000E+00

MONITOR POINT NAME = DALE1         COMPONENT = DALE1          CLASS = GENERAL          SUBCASE NO.      1
LABEL = A REALLY SIMPLE MONSUM1- NO MOMENT TRANSFER
CP = 0           X = 1.000000E+01       Y = 0.000000E+00       Z = 0.000000E+00       CD = 0

AXIS RIGID APPLIED
-----
CX 0.000000E+00
CY 0.000000E+00
CZ 2.000000E+02
CMX 0.000000E+00
CMY 5.000000E+02
CMZ 0.000000E+00

MONITOR POINT NAME = DALET         COMPONENT = DALET          CLASS = GENERAL          SUBCASE NO.      1
LABEL = A REALLY SIMPLE MONSUMT
CP = 0           X = 1.000000E+01       Y = 0.000000E+00       Z = 0.000000E+00       CD = 0

AXIS RIGID APPLIED
-----
CX 0.000000E+00
CY 0.000000E+00
CZ 2.000000E+02
CMX 0.000000E+00
CMY 1.000000E+03
CMZ 0.000000E+00

```

The MONSUM (DALE) simply adds these two MONPNT1's together resulting in a Z force of 20.0 and a moment about the y axis of 50.0 (from the monpnt1 at the tip). For the MONSUM, there is no output location so there is a message that "CP, X, Y, Z and CD are not defined for monitor points results created from a MONSUM." The MONSUM1 (DALE1) also adds the results together, but now the user has specified an output location so that CP, X, Y, Z and CD locations as input by the user are included in the output. This specification has no effect on the results relative to the MONSUM. Finally the MONSUMT (DALET) is located at the tip so now the MONPNT1 at the half span contributes a moment so that the Z force is again 20.0 but the moment is 100.0 (=50. + 5.0 * 10.0)

Guidelines

MONSUM output for CP, X, Y, Z and CD are not meaningful for the typical case and will cause the following message:

"CP, X, Y, Z AND CD ARE NOT DEFINED FOR MONITOR RESULTS CREATED FROM A MONSUM"



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.



11

Inertia Relief

- Introduction to Inertia Relief
- Inertia Relief Using PARAM,INREL,-1 (SUPPORT)
- Inertia Relief Using PARAM,INREL,-2 (no Support)



Introduction to Inertia Relief

Inertia relief is an advanced option in MSC Nastran that allows you to simulate unconstrained structures in a static analysis. It enables applied static loading to an unconstrained structure and the calculation of deformed shape and internal loads within the accelerated structure. Inertia relief calculates the rigid body mass x acceleration loads imparted by the applied loads, and applies them in combination to the flexible body to produce a load-balanced static formulation in the linear acceleration reference frame. The “steady-state” relative structural displacements and internal loads are calculated using support entries (PARAM,INREL,-1) or automatically, the no support capability (PARAM,INREL,-2). Typical applications of inertia relief include modeling an aircraft in flight, an automobile on a test track, or a satellite in space.

Inertia relief is commonly used to measure inertial compliance or dynamic stiffness of an unconstrained component under attachment loads or under center of gravity unit loads. Inertia relief has the valuable property within SOL 101 of providing the static-dynamic convergence shape and elastic modes solution of a dynamically loaded free-free structure as $f \rightarrow 0$ Hz in SOL 111 (using PARAM,LFREQ and residual load vectors).

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 SUPPORT 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 SUPPORT 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.

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,

$$[K_{aa}] \cdot [u_a] = \{P_a\} \quad (11-1)$$

$$[u_r] = \{0\} \quad (11-2)$$

With the automatic inertia relief, the u_r variables are no longer a subset of the u_a variables. The constraint equation, [Equation \(11-2\)](#), is replaced by a more general constraint equation,

$$[M_{aa} \cdot D_{a6}]^T \cdot \{u_a\} = \{u_r\} = \{0\} \quad (11-3)$$

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.

PARAM,INREL,-2, provides the following enhancements to inertia relief analysis:

1. Auto-suport (SUPORT entries are not required) for the 6 rigid body motions of an unconstrained connected structure.
2. Unique displacement solution that is decoupled from rigid body motion and identical to SOL 111 elastic modes static solution as $f \rightarrow 0$ Hz.
3. Improved accuracy in internal loads and stress calculations in “difficult-to-suport” models.

There is no actual Suport of grids or scalar points in the Inrel = -2 implementation. Unlike INREL = -1, there are no enforced zeroes or “center of displacement.” The INREL = -2 solution typically balances around the CG of the structure, with the CG displacing slightly in the direction of the applied loads. This “balance” differs for each applied load P . INREL = -2 uses PARAM,GRDPNT and rigid body acceleration as a geometric reference, not required to be part of structure or independent a-set.

Guidelines for Inertia Relief

1. Inertia relief analysis requires that the model have inertia, that is, mass. This is easily accomplished for most models by use of the RHO field (mass density) on the MATi Bulk Data entries. Line structures, such as a shaft made from BAR elements, may not have moments of inertia along the torsion axis. Use either BEAM elements, which can have torsion mass moments of inertia, or add inertia terms with CONM2 entries.
2. INREL = -2 with auto-suport is designed for the common 6 rigid-body-mode unconstrained connected model. If the model is intended to have exactly 6 rigid body modes, then INREL = -2 is the most convenient and numerically stable option. SUPPORTi entries must NOT be used.



3. If the model is semi-constrained with < 6 rigid body modes, including 1-D or 2-D models, INREL = -1 is recommended, using < 6 appropriate support DOF.
4. Inertia relief is not recommended on models with extra mechanisms or unconnected parts with mass, with > 6 zero Hz modes. INREL = -1 will fail with supports > 6, and INREL = -2 with auto-support will fail or give unreliable results.
5. For models with local or massless singularities beyond the 6 rigid body modes, INREL = -2 with auto-support can produce a reasonable result. But care should be taken, and singular DOF with high factor diagonals in the .f06 should be checked.
6. PARAM,GRDPNT is the reference point for INREL = -2. If default (0,0,0) is far from the model CG, then GRDPNT should be defined at or near the CG.
7. When SUPPORTi entries are used in INREL = -1, a good rule of thumb is that they be on structure strong and balanced enough to sustain the weight of the structure in a laboratory test (they must have significant stiffness). If the structure does not have such DOFs, a new grid can be defined as a reference node, and connected to many DOFs with an RBE3 element. The reference node must be in the n-set which implies that the UM feature of the RBE3 element must be used.
8. Inrel = -1 SUPPORTi entries must constitute a statically determinate set ≤ 6 . Good selections are either: a) all DOF at CG, or b) DOF distanced out in 3-2-1 fashion to create a good balance of the structure. See discussion in INREL = -1 section.
9. INREL = -2 should be used if the inertia relief solution is intended to match the SOL 111 $f \rightarrow 0$ Hz elastic solution exactly.
10. INREL = -1 should be used if a zeroed "center of displacement" is desired.
11. Both inertia relief methods are most accurate on ungrounded "clean" models with rigid body modes on the order E-4 less than first flexible mode frequency. Case Control command, GROUNDCHECK, can be helpful in detecting grounding problems. For INREL = -1, Support Strain Energy in .f06 should also be checked. The INREL = -2 method remains more robust and accurate, as long as 6 rigid modes are intended and remain fairly distinct from the 7th mode. For models where separation between 6th and 7th mode is less than E-1, then INREL = -1 with good support selection should be considered as well.

Inertia Relief Using PARAM,INREL,-1 (SUPORT)

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.



In addition to the PARAM,INREL,-1 entry, a **SUPPORT** 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 SUPPORT 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.)

To invoke inertia relief, you must provide a **SUPPORT** 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 SUPPORT entry in static analysis is if you hold the SUPPORT degrees of freedom constrained, there is no possible rigid body motion. If all possible rigid body motion is not described on the SUPPORT 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 SUPPORT 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 SUPPORT degrees of freedom in MSC Nastran. The set of degrees of freedom described on the SUPPORT 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 SUPPORT degrees of freedom.

The SUPPORT 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 SUPPORT entry is misused.

Solution 101 requires that the model does not contain mechanisms. The reference degrees of freedom are specified using the the SUPPORT entry as follows:

SUPPORT

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.



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](#).

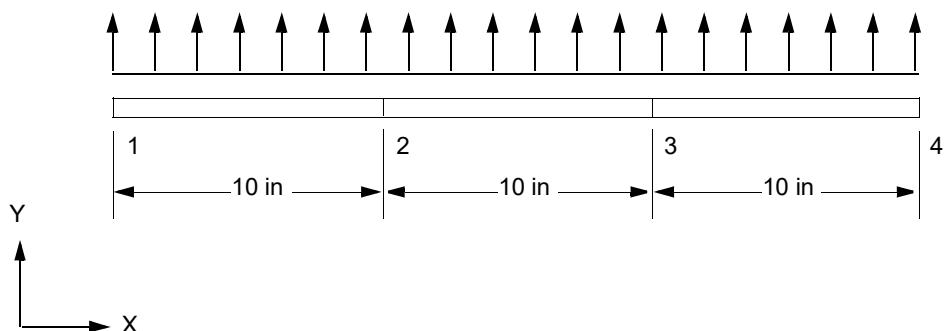


Figure 11-1 Inertia Relief Analysis of a CBEAM Model

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. Part of 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, (See **MSC_DOC_DIR/doc/linstat/inertia1.dat**), the SUPORT point is located at grid point 1. All six degrees of freedom 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  
$  
$ SUPORT ENTRY FOR INERTIA1.DAT  
$  
SUPORT 1      123456
```



```
PARAM    GRDPNT   1
PARAM    INREL    -1
$
$ SUPPORT 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          .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](#).



```

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
0          REFERENCE POINT =      1
0          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 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 RESULTANTS ABOUT      1 IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0
SUBCASE/ LOAD          OLOAD      RESULTANT
DAREA ID  TYPE    T1      T2      T3      R1      R2      R3
0       1  FX  0.000000E+00  -----  ----- 0.000000E+00 0.000000E+00
0       1  FY  ----- 3.000000E+04  ----- 0.000000E+00 4.500000E+05
0       1  FZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       1  MX  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       1  MY  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       1  MZ  ----- 0.000000E+00 0.000000E+00 7.639755E-11
0       1  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 0.000000E+00
0       2  FY 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       2  FZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       2  MX  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       2  MY  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       2  MZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       2  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 0.000000E+00 0.000000E+00 0.000000E+00
0       3  FY 0.000000E+00 -2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 -3.343500E+00
0       3  FZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       3  MX  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       3  MY  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       3  MZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       3  TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       4  FX 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       4  FY 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       4  FZ 0.000000E+00 -2.229000E-01 0.000000E+00 3.343500E+00 0.000000E+00
0       4  MX  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       4  MY  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       4  MZ  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       4  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 0.000000E+00 0.000000E+00 0.000000E+00
0       5  FY 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       5  FZ 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       5  MX 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00 0.000000E+00
0       5  MY 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       5  MZ 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0       5  TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00

```

Figure 11-2 Inertia Relief Output When Grid Point 1 is the SUPPORT Point



```

0      6    FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY  ----  0.000000E+00  ----  3.343500E+00  0.000000E+00 -7.058500E+01  ----
      FZ  ----  ----  3.343500E+00  0.000000E+00 0.000000E+00  ----  ----
      MX  ----  ----  ----  0.000000E+00 0.000000E+00  ----  ----
      MY  ----  ----  ----  ----  0.000000E+00 0.000000E+00  ----
      MZ  ----  ----  ----  ----  ---- 0.000000E+00 0.000000E+00
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 0.000000E+00 0.000000E+00 0.000000E+00
      FZ  ----  ----  0.000000E+00 0.000000E+00 0.000000E+00  ----  ----
      MX  ----  ----  ----  0.000000E+00 0.000000E+00  ----  ----
      MY  ----  ----  ----  ----  0.000000E+00 0.000000E+00  ----
      MZ  ----  ----  ----  ----  ---- 0.000000E+00 0.000000E+00
TOTALS 0.000000E+00 -3.343500E+00 0.000000E+00 0.000000E+00 0.000000E+00 -7.058500E+01

*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)
THE DECOMPOSITION OF KLL  YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF 7.702858E+00
*** 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     1.7762876E-15 0.0000000E+00
2     1.7762876E-15 7.7640316E-12
3     1.7762876E-15 0.0000000E+00
4     1.7762876E-15 0.0000000E+00
5     1.7762876E-15 0.0000000E+00
6     1.7762876E-15 4.0163286E-09

0
INTERMEDIATE MATRIX ... QRR

1     2.229000E-01  0.000000E+00  COLUMN 1  0.000000E+00  0.000000E+00  0.000000E+00 6
1     0.000000E+00  2.229000E-01  COLUMN 2  0.000000E+00  0.000000E+00  3.343500E+00 6
1     0.000000E+00  0.000000E+00  COLUMN 3  2.229000E-01  0.000000E+00 -3.343500E+00 0.000000E+00 6
1     0.000000E+00  0.000000E+00  COLUMN 4  0.000000E+00  1.858986E-01  0.000000E+00 0.000000E+00 6
1     0.000000E+00  0.000000E+00  COLUMN 5  -3.343500E+00  0.000000E+00  7.058500E+01 0.000000E+00 6
1     0.000000E+00  3.343500E+00  COLUMN 6  0.000000E+00  0.000000E+00  0.000000E+00 7.058500E+01 6
1 BAR WITH SUPORT ENTRY          OCTOBER 9, 2009 MSC NASTRAN 6/12/08 PAGE 10

0
INTERMEDIATE MATRIX ... QRL

1     0.000000E+00 -3.000000E+04  COLUMN 1  0.000000E+00  0.000000E+00  0.000000E+00 6
1
INTERMEDIATE MATRIX ... URA

1     0.000000E+00  1.345895E+05  COLUMN 1  0.000000E+00  0.000000E+00  2.473939E-12 6

```

Figure 11-2 Inertia Relief Output When Grid Point 1 is the SUPPORT Point (continued)



```

0 *** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO.           EPSILON          EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
    1           -2.3053298E-15   1.5617192E+02

0 RESULTANTS ABOUT      1 IN SUPERELEMENT BASIC SYSTEM COORDINATES.

0                               SPCFORCE RESULTANT
SUBCASE/ LOAD             T1            T2            T3            R1            R2            R3
DAREA ID  TYPE          0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
0       1   FX           0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
                  FY           0.000000E+00  -2.642321E-12 0.000000E+00 0.000000E+00 0.000000E+00
                  FZ           0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
                  MX           0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
                  MY           0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
                  MZ           0.000000E+00  ----- 0.000000E+00 0.000000E+00 0.000000E+00
TOTALS   0.000000E+00  -2.642321E-12 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00

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

F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T
POINT ID.  TYPE        T1            T2            T3            R1            R2            R3
    1   G           0.0           -2.642321E-12 0.0            0.0            0.0            0.0

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     0.0
                  2   1.000  0.0     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     0.0
                  3   1.000  0.0     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     0.0
                  4   1.000  0.0     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 inertial reaction forces and moments created by imposing unit acceleration(s) 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 [Figure 11-1](#)). (See MSC_DOC_DIR/doc/linstat/inertia2.dat) Some of intermediate output varies slightly, but the interesting area is in the resulting displacement and stress output as shown in [Figure 11-4](#).

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				
STRESSES (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

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:



DMIG

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.

A partial input file for this example is shown in [Listing 11-2](#). (See MSC_DOC_DIR/doc/linstat/uaccel.dat)

Listing 11-2 Imposing a 10 g Acceleration Using the PARAM,UACCEL

```
$
$ FILENAME - UACCEL.dat
$
$
$ SUPORT INFORMATION
$
SUPORT 2      123456
PARAM   GRDPNT 2
PARAM   INREL  -1
DMIG    UACCEL 0      9      1
DMIG    UACCEL 1          2          2      3864.0
$
```

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 $3864. \text{in/sec}^2$ 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 SUPPORT1 entries instead of the SUPPORT entry. Unlike the SUPPORT entry, which is automatically activated, the SUPPORT1 entry must be selected by the SUPPORT1 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
                                         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      -7.547017E-03 0.0          0.0          0.0          1.076069E-03
2       G       0.0       0.0          0.0          0.0          0.0          0.0
3       G       0.0      -3.340174E-02 0.0          0.0          0.0          -5.380345E-03
4       G     -9.475221E-02 0.0       0.0          0.0      -6.456414E-03

L O A D   V E C T O R
POINT ID.  TYPE      T1           T2           T3           R1           R2           R3
1       G       0.0      -1.435476E+02 0.0          0.0          0.0          0.0
2       G       0.0      -2.870952E+02 0.0          0.0          0.0          0.0
3       G       0.0      -2.870952E+02 0.0          0.0          0.0          0.0
4       G     -1.435476E+02 0.0          0.0          0.0          0.0          0.0

```

Figure 11-4 Partial Output of the UACCEL Example

Inertia Relief Using PARAM,INREL,-2 (no Suport)

Inertia relief using param,inrel-2, we will call the Automatic Inertia Relief Method, is the recommended inertia relief method. The SUPPORT entry is no longer needed. Only PARAM,INREL,-2 is needed in the input file. The method is designed for 3D models with exactly 6 rigid body modes. The reference frame is selected automatically, in a manner that poor solutions are unlikely because of the choice of reference frame variables.

The following features are not supported for the Automatic Inertia Relief Method:



- The DMIG,UACCEL option
- The Iterative Solver

As an example we will use the CBEAM model in [Figure 11-1](#) using the automatic inertia relief method. This is accomplished by removing the SUPPORT entry and replacing PARAM,INREL,-1 with PARAM,INREL,-2.(See [MSC_DOC_DIR/doc/linstat/inertia1a.dat](#)) [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:

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	-8.329169E-02	0.0	0.0	0.0	1.874063E-02		
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		

$$U_r = \{0.5 \cdot (-8.329169E-2) + 1.0 \cdot (4.164584E-2) + 1.0 \cdot (4.164584E-2) + 0.5 \cdot (-8.329169E-2)\} = 0 \quad (11-4)$$

BAR USING AUTOMATIC SUPPORT ENTRY									
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	-8.329169E-02	0.0	0.0	0.0	1.874063E-02		
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		

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	1.765873E-12	0.0	0.0	0.0	0.0		
2	G	0.0	1.642524E-12	0.0	0.0	0.0	0.0		
3	G	0.0	-2.466995E-13	0.0	0.0	0.0	0.0		
4	G	0.0	-1.067961E-12	0.0	0.0	0.0	0.0		

ELEMENT-ID	GRID	LENGTH	S T R E S S E S S		I N	B E A M	E L E M E N T S	(C B E A M)				
			STAT DIST/	SXC				S-XF	S-MAX	S-MIN	M.S.-T	M.S.-C
1	1	0.000	0.0	0.0	0.0	0.0	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		
2	2	0.000	0.0	0.0	0.0	0.0	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		
3	3	0.000	0.0	0.0	0.0	0.0	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		

Figure 11-5 Abridged Output Using the Automatic Inertia Relief Method



12

Matrix Operations

- Introduction to Matrix Operations
- Set Definition
- Static Condensation (Guyan Reduction)
- Direct Matrix Input



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

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



$$[K_{gg}]\{u_g\} = \{P_g\} \quad (12-1)$$

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 [Equation \(12-1\)](#).

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, 235](#) 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 \(Guyan Reduction\), 474](#). 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 SUPPORT 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 SUPPORT entry for the automatic calculation of inertia loads in static analysis is described in [Inertia Relief, 455](#).



In order for the SUPPORT entry to work, the finite element model must have a mass matrix. The SUPPORT entry must be specified for all six DOFs on a single grid point. (Note that the word “SUPPORT” is spelled with one “P”.)

Another important use of the SUPPORT entry is to provide a reference point in the computation of rigid body modes in dynamic analysis (see the [MSC Nastran Dynamic Analysis User's Guide](#)). [Table 12-1](#) 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

$g\text{-set} \text{ minus } m\text{-set} = n\text{-set}$
$n\text{-set} \text{ minus } s\text{-set} = f\text{-set}$
$f\text{-set} \text{ minus } o\text{-set} = a\text{-set}$
$a\text{-set} \text{ minus } r\text{-set} = l\text{-set}$

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 PS field on the GRID Bulk Data entries.
o	Degrees of freedom omitted by structural matrix partitioning (see Static Condensation (Guyan Reduction)).
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.

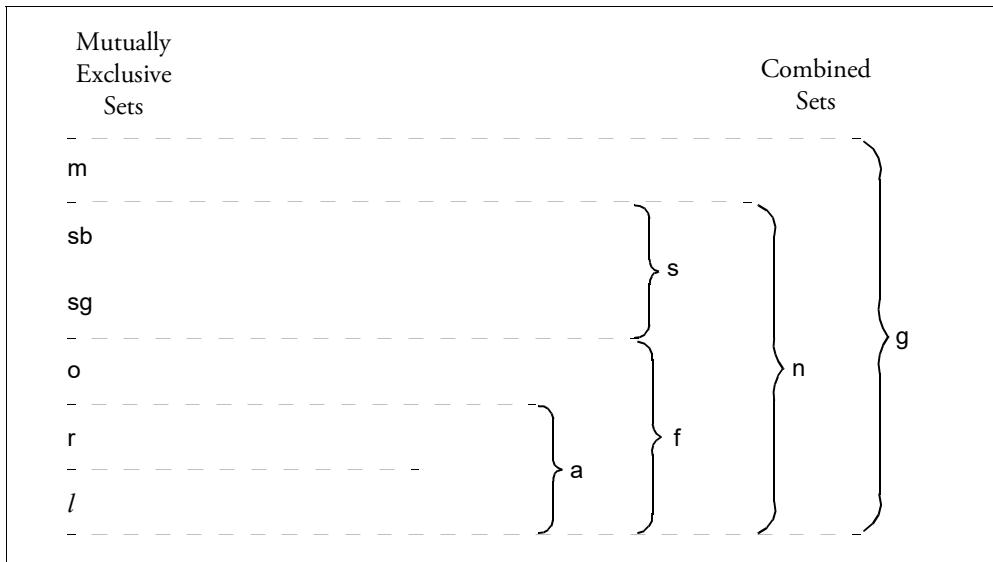


Figure 12-1 Combined Sets Formed from Mutually Exclusive Sets

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, RJOINT
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](#).

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

$$[K_{ll}]\{u_l\} = \{P_l\} \quad (12-2)$$

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. [Equation \(12-2\)](#) 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. The above discussed, the basic sets in MSC Nastran, for a complete definition of all sets see the [Degree-of-Freedom Set Definitions](#) in the [MSC Nastran Quick Reference Guide](#). [Constraint and Set Notation](#) in the [MSC Nastran Reference Guide](#) has a more complete description of how the sets are generated.

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



$$[K_{ff}]\{u_f\} = \{P_f\} \quad (12-3)$$

Partitioning the free degrees of freedom into two subsets of the f -set, you obtain

$$\{u_f\} = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} \quad (12-4)$$

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

$$[K_{ff}][u_f] = \begin{bmatrix} \overline{K}_{aa} & \overline{K}_{ao} \\ \overline{K}_{oa} & \overline{K}_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} \overline{P}_a \\ \overline{P}_o \end{Bmatrix} \quad (12-5)$$

Expanding the bottom equation, you arrive at

$$\{u_o\} = [G_{oa}]\{u_a\} + \{u_{oo}\} \quad (12-6)$$

where:

$$\begin{aligned} [G_{oa}] &= -[K_{oo}]^{-1}[K_{oa}] \\ \{u_{oo}\} &= [K_{oo}]^{-1}[P_o] \end{aligned}$$

Expanding the top part of [Equation \(12-5\)](#) and substituting u_o with [Equation \(12-6\)](#), you arrive at the equation

$$\{u_a\} = [K_{aa}]^{-1}\{P_a\} \quad (12-7)$$

where:

$$\begin{aligned} [K_{aa}] &= [\overline{K}_{aa} + K_{ao}G_{oa}] \\ \{P_a\} &= [\overline{P}_a + G_{oa}^T P_o] \end{aligned}$$

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 [Equation \(12-7\)](#), which is the analysis set. Once this is done, $\{u_o\}$ can then be solved by [Equation \(12-6\)](#), which is the omitted set. The conventional method involves solving [Equation \(12-3\)](#) 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 (ASET, OMIT)

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.

[ASET](#) can be used in conjunction with [ASET1](#). The formats for the ASET/ASET1 are as follows:

ASET

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 α -set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (σ -set).

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:

ASET1	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 α -set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (σ -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 α -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 (σ -set).

OMIT can be used in conjunction with **OMIT1**. The formats for the OMIT/OMIT1 are as follows:



OMIT

1	2	3	4	5	6	7	8	9	10
OMIT	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:

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).

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).
Gi	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 SPC*i* 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](#). (See MSC_DOC_DIR/doc/linstat/bulkm.dat).

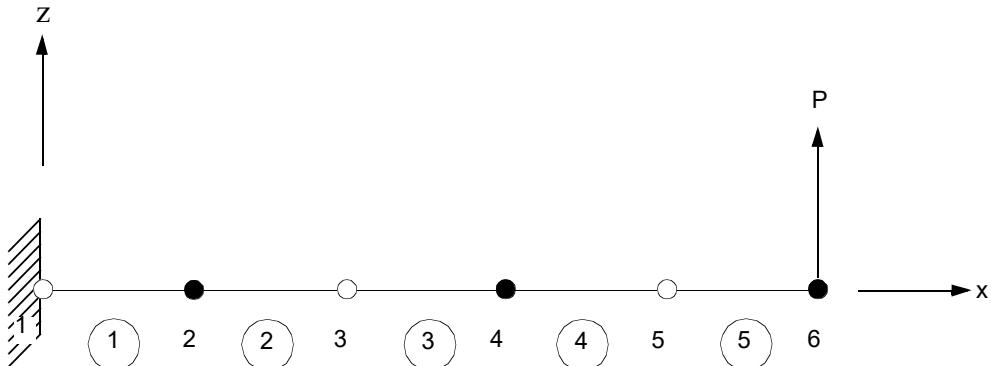


Figure 12-2 Cantilever Beam Using Guyan Reduction

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](#)). (See `MSC_DOC_DIR/doc/linstat/full.dat`) 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](#)). (See `MSC_DOC_DIR/doc/linstat/aset.dat`) 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](#)). (See MSC_DOC_DIR/doc/linstat/omit.dat) 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 *****
```

DISPLACEMENT VECTOR

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 *****
```

DISPLACEMENT VECTOR

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 *****
```

DISPLACEMENT VECTOR

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, 543](#) for further details regarding the use of the INCLUDE statement.

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. Also by providing the mass matrix as well, then dynamic analysis can also be preformed.
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. You can also as an approximation, reduce the matrices for the entire aircraft down to a manageable size using dynamic reduction (see [Static Condensation \(Guyan Reduction\)](#) and [Dynamic Reduction](#)). These reduced mass and stiffness matrices can then be read and added to your various component models. In this case, you may be analyzing a 2000-DOF system, instead of a 100,500-DOF system.
3. The same concept can be applied to a component attached to a test fixture. The stiffness and mass of the fixture at the attachment locations can be read in as a stiffness and mass matrices. Furthermore, there are times whereby the flexibility of the test fixture at the attachment points can be measured experimentally. In this instance, this experimental stiffness matrix can be input to your model. 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](#). 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 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,



$[K_{gg}^2]$ 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** Bulk Data entries. Each of these DMIG data entries contains a column containing the nonzero elements for the particular matrix that you are entering.

DMIG

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		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 being input:		
	1	=	Real, single precision (one field is used per element)
	2	=	Real, double precision (one field per element)
	3	=	Complex, single precision (two fields are used per element)
	4	=	Complex, double precision (two fields per element)
TOUT	Type of matrix to be created:		
	0	=	Set by precision system cell (default)
	1	=	Real, single precision
	2	=	Real, double precision
	3	=	Complex, single precision
	4	=	Complex, double precision



Field	Contents
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format; integer > 0 indicates amplitude, phase format.)
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	B1	
	G2	C2	A2	B2	-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, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank.

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.

Example:

DMIG	STIF	0	6	1					
DMIG	STIF	5	3		5	3	250.		
	5	5	-125.		6		-150.		

DMIG Case Control User Interface

In order to include these matrices, the Case Control Section must contain the appropriate commands; [K2GG \(Case\)](#) in the *MSC Nastran Quick Reference Guide*, M2GG, or P2G.

See the Example and Remarks in the [MSC Nastran Quick Reference Guide](#) for inputting and addition of multiple matrices on the same K2GG, M2GG, etc. selections. Discussions in this guide will only reference one matrix at a time.

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.

B2GG=ourdamp

The above Case Control command adds terms that are defined by the DMIG Bulk Data entries with the name “ourdamp” to the g-set damping 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.



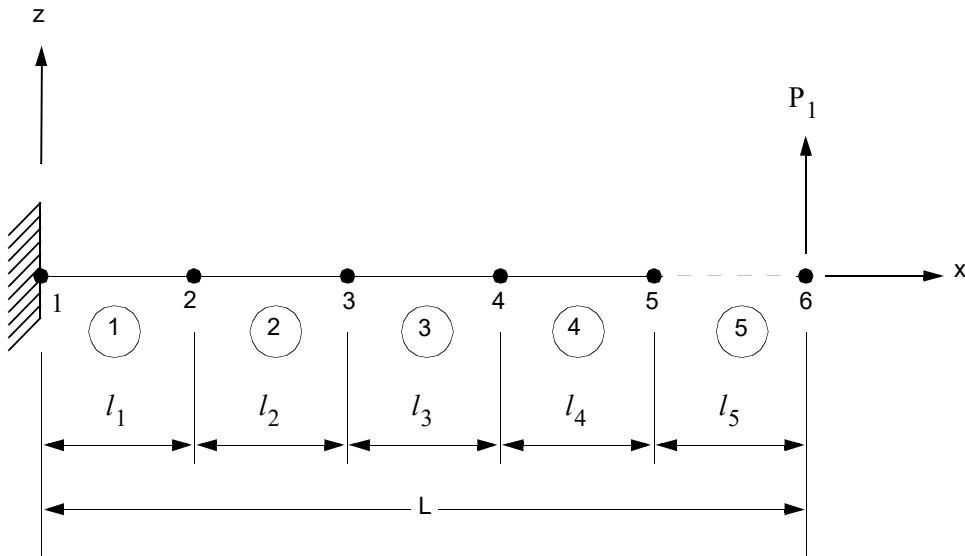


Figure 12-4 Planer Cantilever Beam

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. (See MSC_DOC_DIR/doc/linstat/dmigfull.dat) 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.      0.      1.  

GRID  1       0.      0.      0.      0.      246  

GRID  2       1.      0.      0.      0.      246  

GRID  3       2.      0.      0.      0.      246  

GRID  4       3.      0.      0.      0.      246  

GRID  5       4.      0.      0.      0.      246  

GRID  6       5.      0.      0.      0.      246  

GRID  10      0.      0.      10.     0.      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 $E = 7.1 \cdot 10^{10} N/m^2$, $I = 5.869 \cdot 10^{-7} m^4$, and $l_5 = 1m$, the stiffness matrix for CBAR element number 5 is as follows:

$$[K]_e = \begin{bmatrix} 5, 3 & 5, 5 & 6, 3 & 6, 5 \\ 5, 3 & 500038.3 & & \\ 5, 5 & -250019.4 & 166679.6 & \text{SYM} \\ 6, 3 & -500038.8 & 250019.4 & 500038.8 \\ 6, 5 & -250019.4 & 83339.8 & 250019.4 & 166679.6 \end{bmatrix} \quad (12-8)$$

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.

A partial input file using DMIG is contained in [Listing 12-6](#). (See MSC_DOC_DIR/doc/linstat/dmigstfs.dat) 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 as shown in partial listing ([Listing 12-7](#)). (See MSC_DOC_DIR/doc/linstat/dmigsifl.dat.) 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 DMIN with Small-Field Format

```

$  

$     FILENAME - DMINSTFS.DAT  

$  

$     SUBTITLE = USE DMIN TO BRING IN STIFFNESS MATRIX OF ELEMENT # 5 USING SMALL FIELD  

$     K2GG = UGSTIF  

$  

$     DMIN HEADER ENTRY  

$  

$     DMIN      UGSTIF   0       6       1  

$  

$     DMIN DATA COLUMN ENTRIES  

$  

$     DMIN      UGSTIF   5       3       5       3       500039.       +000001  

$     +0000015    5       -250019.       6       3       -500039.       +000002  

$     +0000026    5       -250019.             

$  

$     DMIN      UGSTIF   5       5       5       5       166680.       +000004  

$     +0000046    3       250019.       6       5       83340.       +000005  

$  

$     DMIN      UGSTIF   6       3       6       3       500039.       +000006  

$     +0000066    5       250019.             

$  

$     DMIN      UGSTIF   6       5       6       5       166680.  

$
```

Listing 12-7 Input File for Reading a Stiffness Matrix with Large-Field Format

```

$  

$     FILENAME - DMINSTFL.DAT  

$  

$     SUBTITLE = DMIN TO BRING IN STIFFNESS MATRIX FOR ELEMENT # 5 USING LARGE FIELD  

$     K2GG = UGSTIF  

$  

$     DMIN HEADER ENTRY  

$  

$     DMIN      UGSTIF   0       6       2  

$  

$     DMIN DATA COLUMN ENTRIES  

$  

$     DMIN*      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             

$  

$     DMIN*      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             

$  

$     DMIN*      UGSTIF   6       3           *A3  

$     *A3       6       3       5.000388 D+5   *B3  

$     *B3       6       5       2.500194 D+5             

$  

$     DMIN*      UGSTIF   6       5           *A4  

$     *A4       6       5       1.666796 D+5             

$
```



***** 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 partial input file is shown in [Listing 12-8](#). (See `MSC_DOC_DIR/doc/linstat/dmigload.dat`) 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  
$  
$  
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  
$  
DMIG      UGLOAD  0          9          1          2  
DMIG      UGLOAD  1          6          3        100.  
$
```



```
***** PARTIAL OUTPUT FROM DMIGFULL.F06 - FULL MODEL *****

TIP LOAD AT END
D I S P L A C E M E N T   V E C T O R   SUBCASE 1
POINT ID.  TYPE    T1      T2      T3      R1      R2      R3
  1   G     .0      .0      5.599566E-03  .0      -1.079916E-02  .0
  2   G     .0      .0      2.079839E-02  .0      -1.919851E-02  .0
  3   G     .0      .0      4.319665E-02  .0      -2.519804E-02  .0
  4   G     .0      .0      7.039454E-02  .0      -2.879777E-02  .0
  5   G     .0      .0      9.999225E-02  .0      -2.999767E-02  .0
  6   G     .0      .0      .0      .0      .0      .0
 10   G     .0      .0      .0      .0      .0      .0

GRAVITY LOAD
D I S P L A C E M E N T   V E C T O R   SUBCASE 2
POINT ID.  TYPE    T1      T2      T3      R1      R2      R3
  1   G     .0      .0      9.278418E-03  .0      -1.729160E-02  .0
  2   G     .0      .0      3.233388E-02  .0      -2.783526E-02  .0
  3   G     .0      .0      6.326194E-02  .0      -3.331796E-02  .0
  4   G     .0      .0      9.784514E-02  .0      -3.542669E-02  .0
  5   G     .0      .0      1.335530E-01  .0      -3.584844E-02  .0
  6   G     .0      .0      .0      .0      .0      .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
D I S P L A C E M E N T   V E C T O R   SUBCASE 10
POINT ID.  TYPE    T1      T2      T3      R1      R2      R3
  1   G     .0      .0      5.599566E-03  .0      -1.079916E-02  .0
  2   G     .0      .0      2.079839E-02  .0      -1.919851E-02  .0
  3   G     .0      .0      4.319665E-02  .0      -2.519804E-02  .0
  4   G     .0      .0      7.039454E-02  .0      -2.879777E-02  .0
  5   G     .0      .0      9.999225E-02  .0      -2.999767E-02  .0
  6   G     .0      .0      .0      .0      .0      .0
 10   G     .0      .0      .0      .0      .0      .0

GRAVITY LOAD
D I S P L A C E M E N T   V E C T O R   SUBCASE 20
POINT ID.  TYPE    T1      T2      T3      R1      R2      R3
  1   G     .0      .0      9.278418E-03  .0      -1.729160E-02  .0
  2   G     .0      .0      3.233388E-02  .0      -2.783526E-02  .0
  3   G     .0      .0      6.326194E-02  .0      -3.331796E-02  .0
  4   G     .0      .0      9.784514E-02  .0      -3.542669E-02  .0
  5   G     .0      .0      1.335530E-01  .0      -3.584844E-02  .0
  6   G     .0      .0      .0      .0      .0      .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 Superelements and Modules User's Guide](#) for a further description of this capability.



13

Linear Buckling

- Introduction to Linear Buckling
- Finite Element Approach
- Eigenvalue Extraction Method
- MSC Nastran Linear Buckling with Local Exclusion of Differential Stiffness
- Assumptions and Limitations of Linear Buckling Analysis
- Examples



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, 505](#).

This chapter is organized into the following sections:

- Finite element approach
- Eigenvalue extraction method
- Assumptions and limitations of linear buckling analysis
- Examples

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.

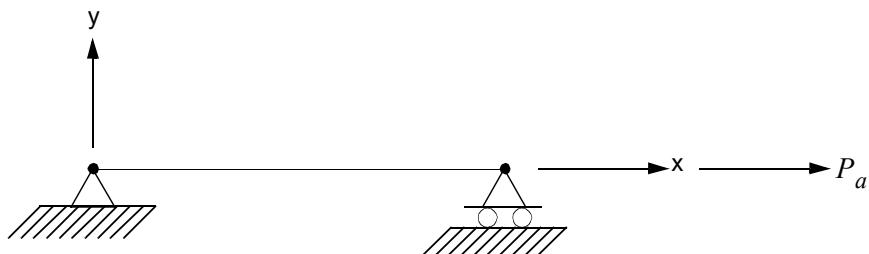


Figure 13-1 Single Planar Bar Element



From *The Nastran Theoretical Manual*, the differential stiffness for this planar bar element can be represented as

$$[k_d]_i = \begin{bmatrix} \frac{6F_{x_i}}{5l_i} & \frac{-F_{x_i}}{10} & \frac{-6F_{x_i}}{5l_i} & \frac{-F_{x_i}}{10} \\ \frac{-F_{x_i}}{10} & \frac{2l_i F_{x_i}}{15} & \frac{F_{x_i}}{10} & \frac{-l_i F_{x_i}}{30} \\ \frac{-6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} & \frac{6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} \\ \frac{-F_{x_i}}{10} & \frac{-l_i F_{x_i}}{30} & \frac{F_{x_i}}{10} & \frac{2l_i F_{x_i}}{15} \end{bmatrix} \quad (13-1)$$

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. [Equation \(13-1\)](#) can, therefore, be rewritten as follows:

$$[k_d]_i = P_a \begin{bmatrix} \frac{6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} & \frac{-6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{2l_i\alpha_i}{15} & \frac{\alpha_i}{10} & \frac{-l_i\alpha_i}{30} \\ \frac{-6\alpha_i}{5l_i} & \frac{\alpha_i}{10} & \frac{6\alpha_i}{5l_i} & \frac{\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{-l_i\alpha_i}{30} & \frac{\alpha_i}{10} & \frac{2l_i\alpha_i}{15} \end{bmatrix} = P_a [\bar{k}_d]_i \quad (13-2)$$

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, 505](#)) 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 [Equation \(13-1\)](#); however, the concept is the same.

The overall system stiffness matrix is represented by [Equation \(13-3\)](#).

$$[K] = [K_a] + [K_d] \quad (13-3)$$

The total potential energy is equal to

$$[U] = 0.5 \{u\}^T [K_a] \{u\} + 0.5 \{u\}^T [K_d] \{u\} \quad (13-4)$$

In order for the system to achieve static equilibrium, the total potential must have a stationary value; in other words, the relationship in [Equation \(13-5\)](#) must be satisfied

$$\frac{\partial [U]}{\partial u_i} = [K_a] \{u\} + [K_d] \{u\} = \{0\} \quad (13-5)$$

where u_i is the displacement of the i-th degree of freedom.

It is convenient to rewrite [Equation \(13-5\)](#) as

$$[[K_a] + P_a [\overline{K}_d]] \{u\} = \{0\} \quad (13-6)$$

where $[K_d] = P_a [\overline{K}_d]$ and P_a is the applied load. In order for [Equation \(13-6\)](#) to have a non-trivial solution, the following relationship must be true:

$$|[K_a] + P_a [\overline{K}_d]| = \{0\} \quad (13-7)$$

where $||$ stands for the determinant of the matrix. [Equation \(13-7\)](#) is only satisfied for certain values of P_a . These values of P_a 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,

$$P_{cr_i} = \lambda_i \cdot P_a \quad (13-8)$$

[Equation \(13-7\)](#) can, therefore, be rewritten as

$$|[K_a] + \lambda_i [K_d]| = [0] \quad (13-9)$$

[Equation \(13-9\)](#) is in the form of an eigenvalue problem. Once you obtain the eigenvalues λ_i , the buckling loads can then be obtained using [Equation \(13-8\)](#). 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 [Equation \(13-8\)](#), 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 [Equation \(13-9\)](#) 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 [Equation \(13-9\)](#). 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](#) 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 [Equation \(13-8\)](#) 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, 506](#) 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](#) contains six example problems that illustrate some of these guidelines.

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 Dynamic Analysis 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 \(Case\)](#) 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.

The [EIGRL](#) entry has the following format:



EIGRL

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, and 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.

The **EIGB** entry has the following format:

EIGB

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.

MSC Nastran Linear Buckling with Local Exclusion of Differential Stiffness

Introduction

In this release a new parameter, PARAM, EXCLUDE, n, has been added to allow the user to select sets of GRID points for which differential stiffness in SOL105 will be ignored for n positive or retained if n is negative.

Benefits

Allows users to remove zones from linear buckling analysis where buckling is known to be of no interest.

Parameter

EXCLUDE

Default = 0

PARAM, EXCLUDE, Integer. Add PARAM, EXCLUDE, n in the buckling subcase(s) to activate linear buckling with local exclusion of differential stiffness. If n is positive, it selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be ignored. If n is negative, the absolute value selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be retained.

Care must be exercised by the user to select GRID points that follow natural mesh lines as much as possible, otherwise an element may be connected to some GRID points that have differential stiffness and some GRID



points that do not. Sometimes this is impossible to ensure, and so the transition zone between retained-excluded differential stiffness should not be in, or close to, a region of interest. Selecting stiff (unlikely to buckle) zones for the transition zones is also advised.

Example

```
SOL 105
CEND
DISPLACEMENT (SORT1,REAL)=ALL
SPCFORCES (SORT1,REAL)=ALL
SUBCASE 1
  SPC = 1
  LOAD = 1
SUBCASE 2
  STATSUB = 1
  SPC = 1
  METHOD = 1
  PARAM,EXCLUDE,99
BEGIN BULK
EIGRL,1,,,3
$ exclude KDIFF for the thinner upper structure
SET1,99,49,THRU,84
...
ENDDATA
```

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, CBEAM3, CBEND, CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, 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. For 3-D buckling problems, the use of PARAM,K6ROT is recommended for CQUAD4 and CTRIA3 elements. A value of 100 is the default.
7. 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 or use full nonlinear analysis with SOL 400. The discussion of nonlinear analysis is beyond the scope of this user's guide. See Reference 16. for further details regarding nonlinear buckling analysis.



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](#).

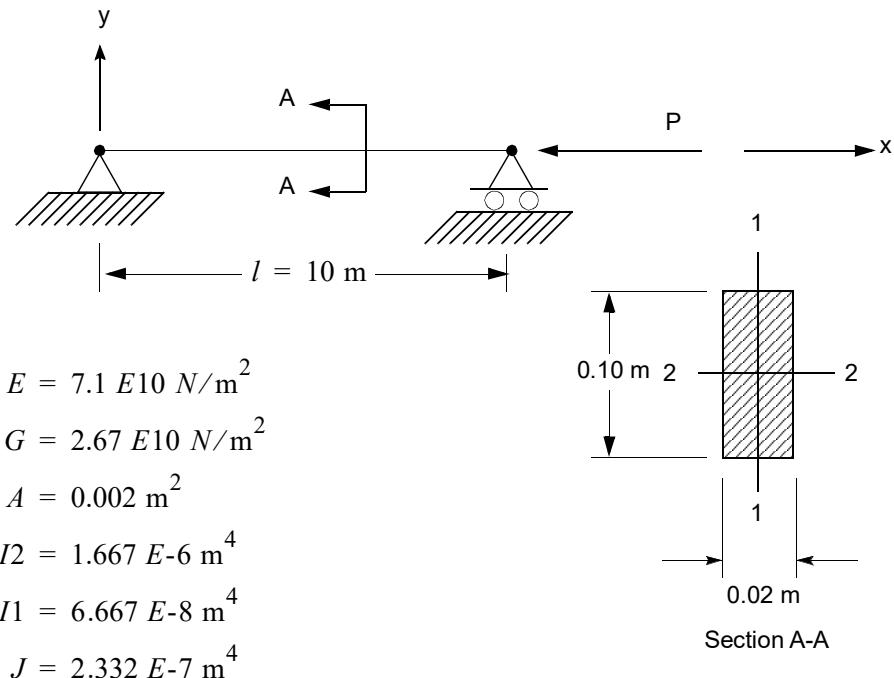


Figure 13-2 Euler Beam

The initial buckling occurs in the xz plane since I_1 is smaller than I_2 .

Results for Example 1



The 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  
$  
PARAMPOST0  
EIGRL 10  
FORCE 10      11          5  
        -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 [Listing 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

$$P_1 = (\lambda_1)(P_a) = (.46719)(-1000) = -467.2 \text{ N}$$

Listing 13-3 Eigenvalue Table for a Euler Beam

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		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

A partial output that contains the first, second, and fifth eigenvectors is shown in [Listing 13-4](#). The mode shapes for the first five modes are plotted in [Figure 13-3](#).

Listing 13-4 Eigenvectors for an Euler Beam



EIGENVALUE = 4.671909E-01

POINT ID.	TYPE	REAL EIGENVECTOR NO.			1		
		T1	T2	T3	R1	R2	R3
1	G	.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	

EIGENVALUE = 1.869135E+00

POINT ID.	TYPE	REAL EIGENVECTOR NO.			2		
		T1	T2	T3	R1	R2	R3
1	G	.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	

EIGENVALUE = 1.168152E+01

POINT ID.	TYPE	REAL EIGENVECTOR NO.			5		
		T1	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	2.671228E-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	



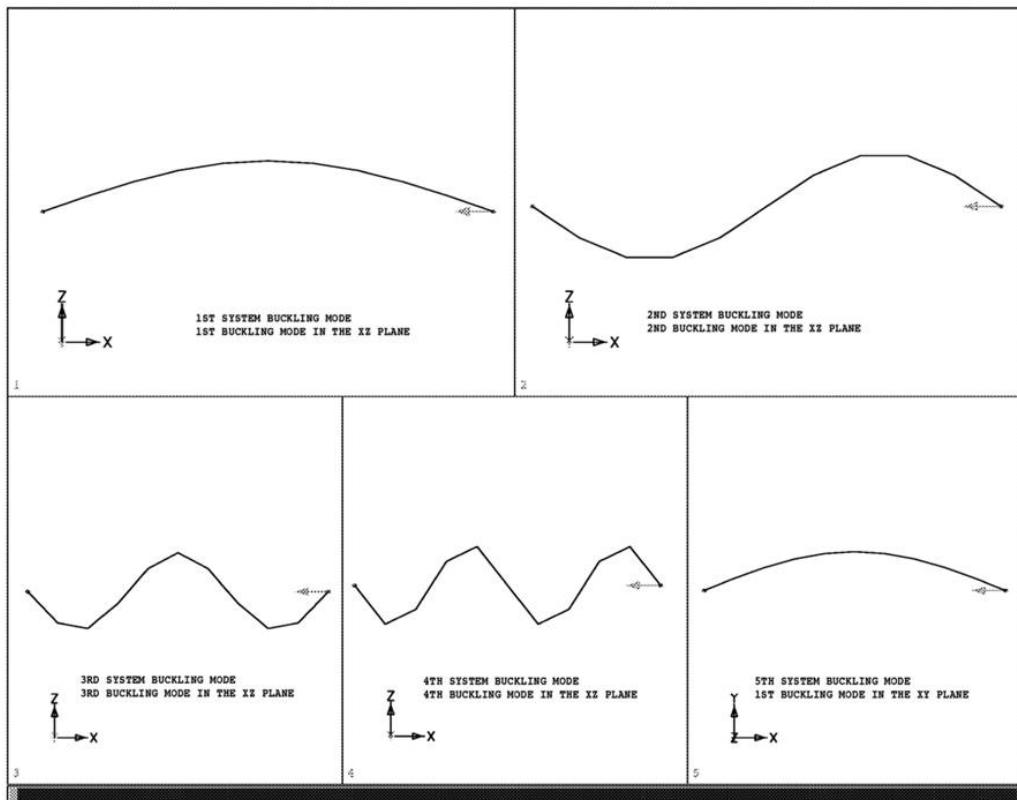


Figure 13-3 Mode Shapes for an Euler Beam

Note that the first four system buckling modes are in the xz plane and the first buckling mode in the xy plane is, therefore, equal to

$$P_5 = (\lambda_5)(P_a) = (11.6815)(-1000) = -11681.5 \text{ N}$$

Theoretical Results for Example 1

From Reference 8., 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 13-4](#) 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).

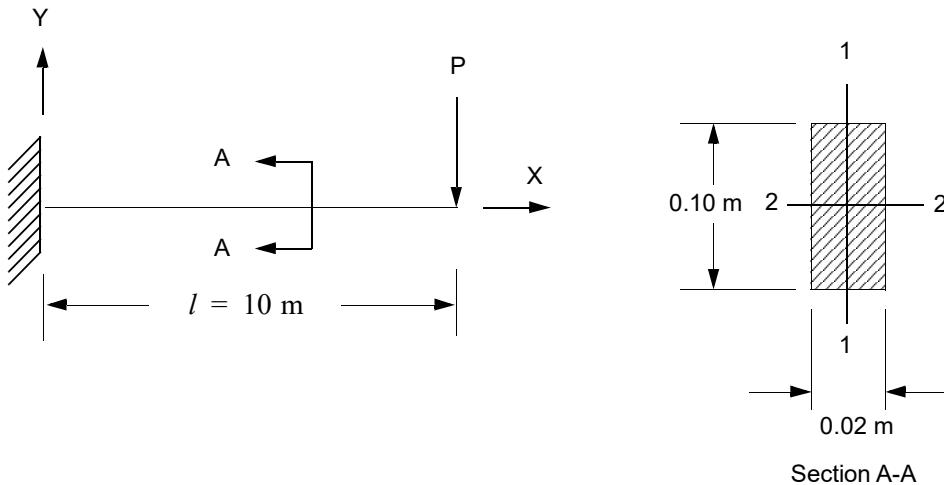


Figure 13-4 Lateral Buckling

Results for Example 2

[Listing 13-5](#) contains the corresponding model for this lateral buckling problem. (See MSC_DOC_DIR/doc/linstat/bucklat.dat) 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. [Listing 13-6](#) contains the eigenvalue table and corresponding eigenvectors. The static deflection and buckling mode shapes are shown in [Figure 13-5](#).

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 13-5 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  
$  
PBEAM11.0026.667-81.667-62.332-7  
$  
SPC1    10      123456  1  
$  
ENDDATA
```

Listing 13-6 Eigenvalue Table and Eigenvectors for Lateral Beam Buckling

R E A L E I G E N V A L U E S		RADIANS		CYCLES		GENERALIZED	GENERALIZED
MODE NO.	EXTRACTION ORDER	EIGENVALUE				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							
POINT ID.	TYPE	T1	T2	T3	R1	1	R2
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	-1.207668E-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	-6.074866E-08
100	G	.0	.0	.0	.0	.0	.0

EIGENVALUE = -2.183456E-01							
POINT ID.	TYPE	T1	T2	T3	R1	2	R2
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



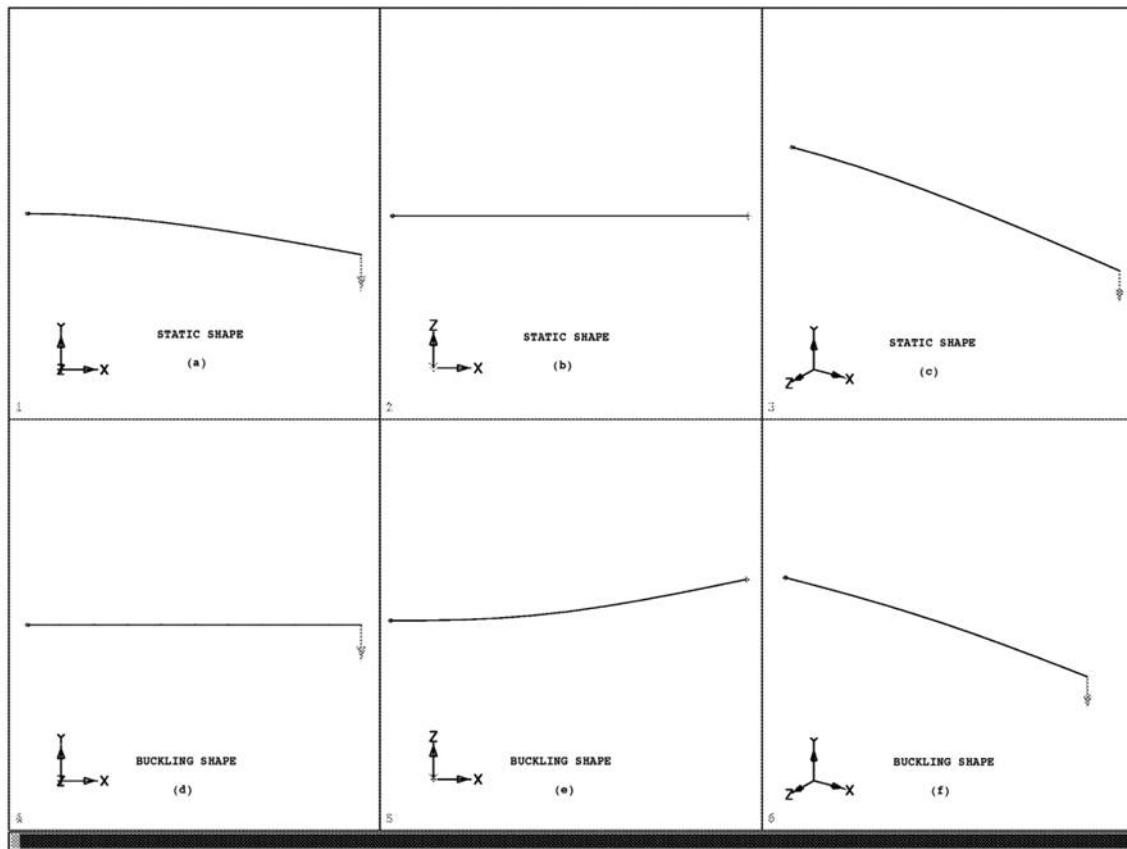


Figure 13-5 Static and Buckling Shapes

The lowest buckling load calculated by MSC Nastran is

$$P_{cr_1} = \pm 218(1000) = \pm 218N$$

Theoretical Results for Example 2

From Reference 8., the theoretical buckling load can be calculated as follows:

$$\begin{aligned} P_{cr} &= \frac{4.013}{l^2} \sqrt{EIGJ} \\ &= \frac{4.013}{100} \sqrt{7.1 \cdot 10^{10} (6.667 \cdot 10^{-8})(2.67 \cdot 10^{10})(2.332 \cdot 10^{-7})} \\ &= 218 N \end{aligned}$$

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 13-6](#). 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 13-7\(a\)](#) and S. P. Timoshenko and J. M. Gere, *Theory of Elastic Stability* (Reference 8.), this is not the case. The lowest buckling mode is actually antisymmetric. The second buckling mode is symmetric ([Figure 13-7\(b\)](#)).

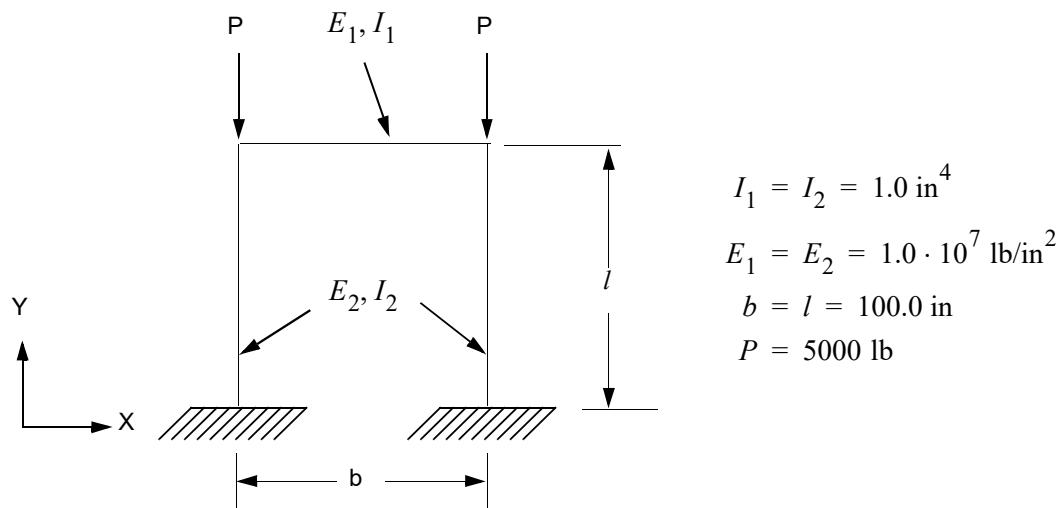


Figure 13-6 Planar Frame Buckling



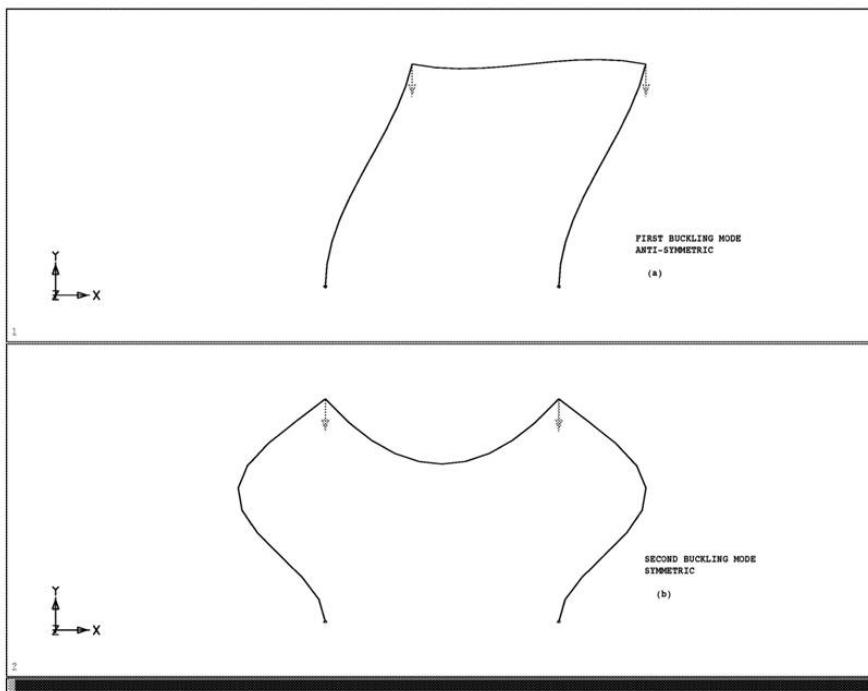


Figure 13-7 Frame Buckling

Results for Example 3

A partial listing of the MSC Nastran input file used to model this problem is shown in [Listing 13-7](#). (See `MSC_DOC_DIR/doc/linstat/bukframe.doc`)



Listing 13-7 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 8.), the buckling load P_{cr} can be calculated as

$$P_{cr} = k^2 E_2 I_2 \quad (13-10)$$



where the value of k can be obtained by solving the following transcendental equation

$$\frac{kl}{tan kl} = \frac{-6 l I_1}{b I_2} \quad (13-11)$$

Solving [Equation \(13-11\)](#), the lowest value of k is equal to

$$k_1 = 0.02716 \quad (13-12)$$

Note that all the roots from the above transcendental equation only yield the anti-symmetric modes. In this case, the lowest buckling load is

$$P_{cr_1} = (.02716)^2 (1 \cdot 10)^7 (1.0) = -7377 \text{ lb}$$

[Table 13-4](#) contains a comparison between the theoretical results versus the MSC Nastran results for the antisymmetric frame buckling.

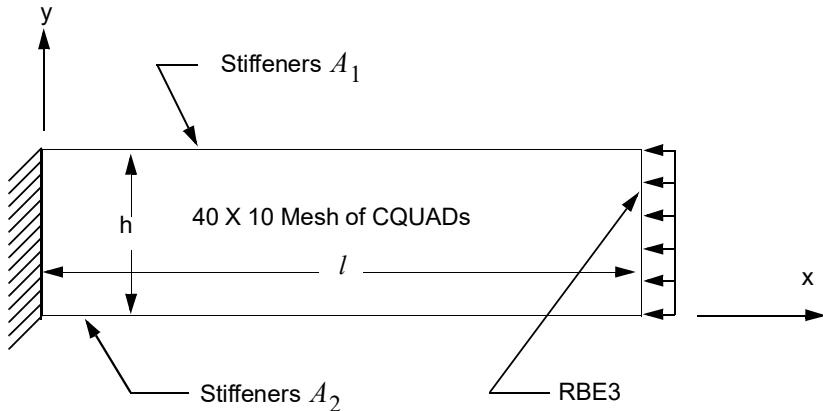
Table 13-4 Frame Buckling Results Comparison

MSC Nastran (lb)	Theoretical (lb)	% Difference
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 13-8](#) 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.





$$A_1 = A_2 = 0.6 \text{ in}^2 ; h = 2.0 \text{ in}$$

$$A_t = 1.2 + 0.4 = 1.6 \text{ in}^2 ; A_w = t(h) = 0.2(2.0) = 0.4 \text{ in}^2$$

$$E = 1.0 \cdot 10^7 \text{ psi} ; I = 1.333 \text{ in}^4$$

$$l = 20 \text{ in} ; G = 3.79 \cdot 10^6 \text{ psi}$$

$$t = 0.2 \text{ in} ; n = A_t/A_w = 4$$

Figure 13-8 Buckling of a Stiffened Panel

Results for Example 4

Part of the corresponding input file for this problem is shown in [Listing 13-8](#). (See `MSC_DOC_DIR/doc/linstat/stffqud4.dat`) 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 [Listing 13-9](#) and [Figure 13-9](#), respectively.

Listing 13-8 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  

$  

SEIGRL101  

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  

$  

*****  

$  

ENDDATA

```

Listing 13-9 Eigenvalues for a Stiffened Panel

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	7.810897E-01	8.837928E-01	1.406600E-01	6.161842E+03	4.812951E+03
2	3	4.636940E+00	2.153356E+00	3.427172E-01	1.022307E+07	4.740374E+07
3	2	4.640286E+00	2.154132E+00	3.428408E-01	1.106009E+07	5.132200E+07



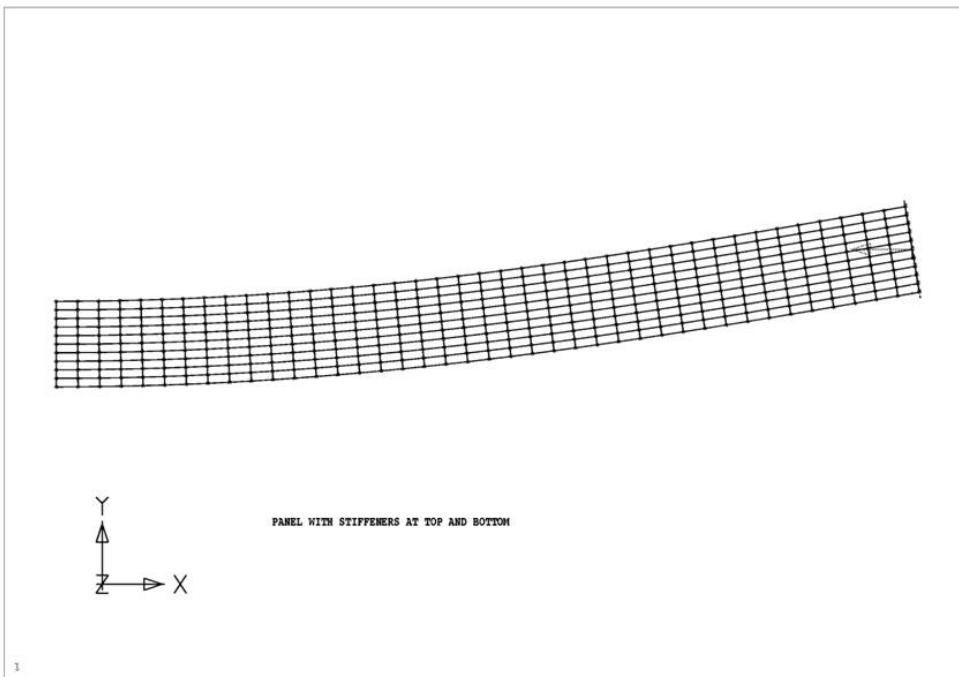


Figure 13-9 Stiffened Panel Buckling Mode

The lowest buckling load calculated by MSC Nastran is

$$P_{cr_1} = (\lambda_1)(P_a) = 0.78118(-(-100,000)) = -78,118 \text{ lb}$$

Theoretical Results for Example 4

If this problem is treated as an Euler beam, the critical buckling load P_e is equal to

$$P_e = \frac{\pi^2 EI}{4 l^2} = 82,245 \text{ lb} \quad (13-13)$$

If the effect of the transverse shear flexibility is included, then (from Reference 8.) can be calculated as follows:

$$P_{cr} = \frac{\sqrt{1 + \frac{4nPe}{A_t G}} - 1}{\frac{2n}{A_t G}} = 78,210 \text{ lb} \quad (13-14)$$

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 13-10](#). This problem illustrates the phenomenon of the buckling of a thin curved shell structure.

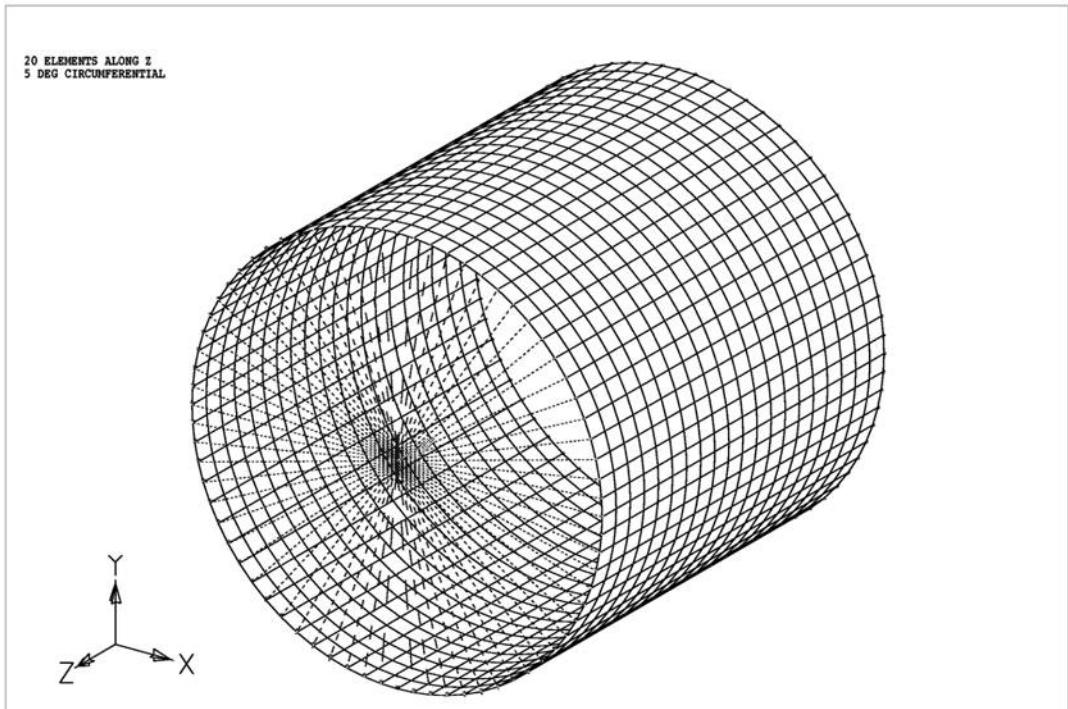


Figure 13-10 Buckling of a Cylinder Under a Compressive Load



Results for Example 5

The CQUAD4 elements are used for the 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 13-10](#). (See MSC_DOC_DIR/doc/linstat/buckcy20r)

Listing 13-10 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  

$  

$ THIS SECTION CONTAINS ALL DEFINED COORDINATE SYSTEMS  

$  

CORD2C 1 0 0.0 0.0 0.0 0.0 0.0 0.0 1. +  

$  

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.  
$  
$ CYLINDER ELEMENT AND GRID DATA  
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 13-11](#).

The Lanczos eigenvalue extraction method is used in this case to request the lowest mode. The lowest calculated eigenvalue is equal to 0.34262. 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 13-12](#). 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 wave.

This model can be generated with Patran; the procedure is shown in [Cylinder Buckling with Patran, 728](#), this model can be generated with SimXpert; the procedure is shown in [Cylinder Buckling with SimXpert, 729](#).

This problem is then remeshed with 40 elements along the z-direction keeping the same mesh density in the circumferential direction. (See MSC_DOC_DIR/doc/linstat/bucky) The new eigenvalue calculated in this case is equal to 0.34234. The revised buckling load is, therefore, equal to

$$P_{cr} = (.34234)(-100,000) = -34,234 \text{ lb}$$

The corresponding mode shape is shown in [Figure 13-13](#). In this case, there are three grid points per half sine wave (six grid points per sine wave), which is still below the minimum requirement. To obtain better accuracy, you can certainly further refine this model until you meet the minimum number of grid points requirement. However, this mesh is sufficient for our goal of demonstrating the linear buckling features of MSC Nastran.



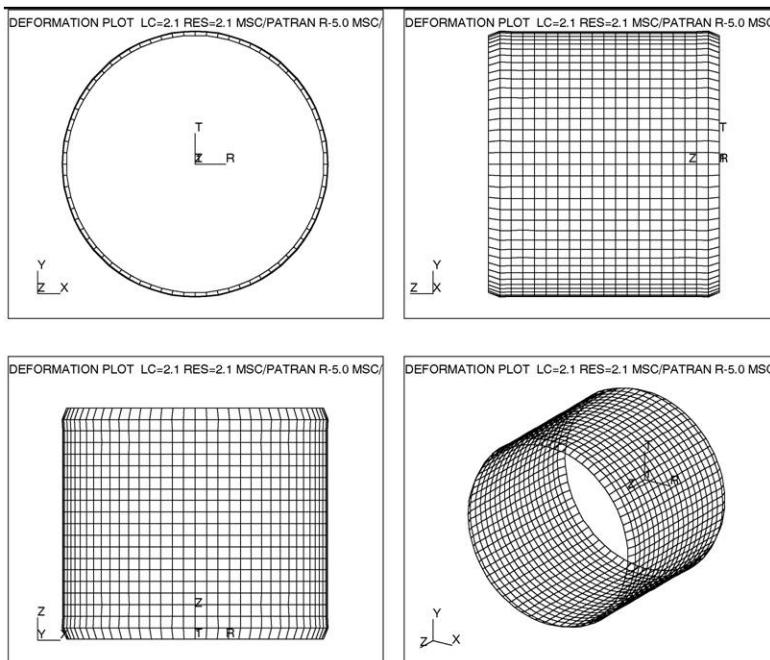


Figure 13-11 Static Deflection of a Cylinder Due to a Compressive Load

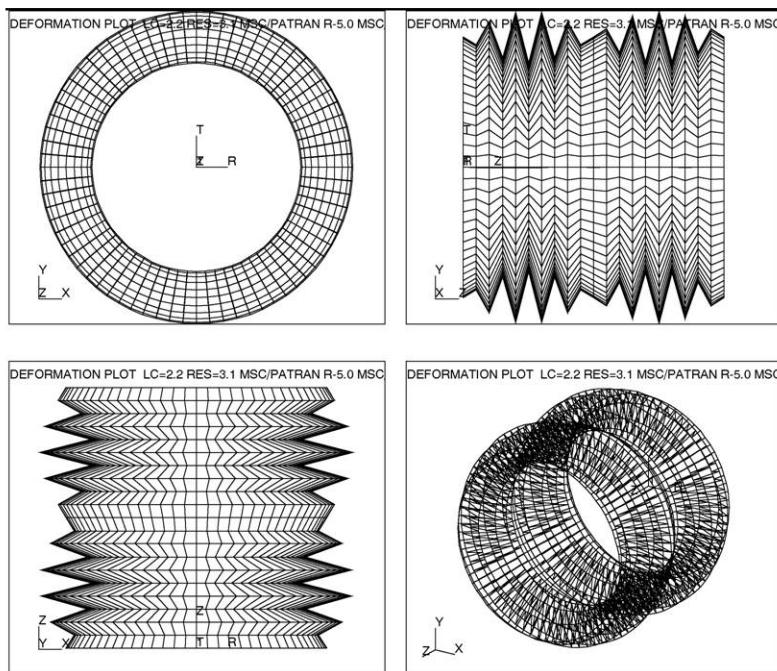


Figure 13-12 Buckling Shapes of a Cylinder with 20 Elements Along the z-Direction

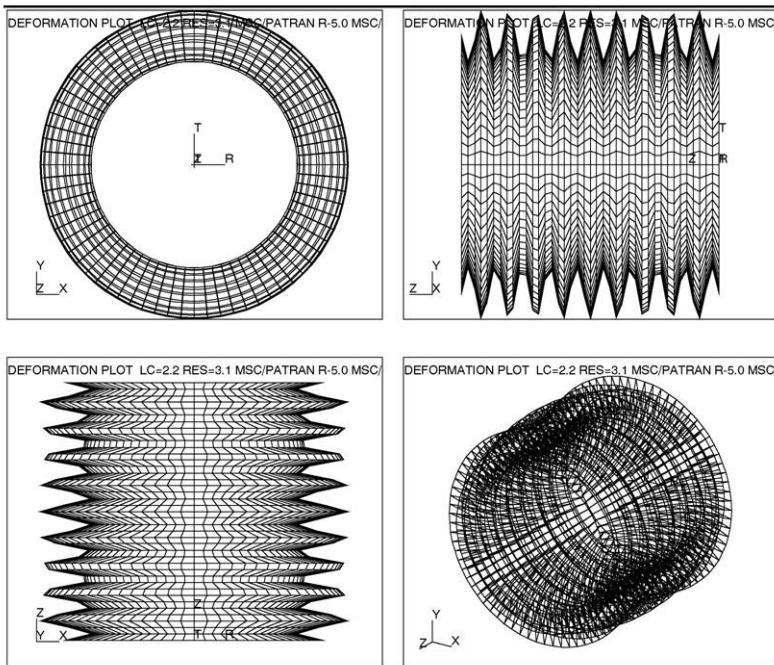


Figure 13-13 Buckling Shape of a Cylinder with 40 Elements Along the z-Direction

Theoretical Results for Example 5

The first buckling load (Reference 8.) can be calculated as follows:

$$N_x = \frac{E(h)^2}{r\sqrt{3(1-v^2)}} = \frac{1 \times 10^7 (0.03)^2}{10\sqrt{3(1-0.3^2)}} = 544.7 \text{ lb/in}$$

$$P_{cr} = N_x(2)(\pi)r = 544.7 (2)(\pi)(10) = 34,225 \text{ lb}$$

[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 using CBEAM elements. Part of the corresponding input file is shown in [Listing 13-11](#). (See `MSC_DOC_DIR/doc/linstat/multbuck.dat`)

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 13-4](#)). 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 Example 1. Note that different boundary conditions are allowed for different subcases.



Listing 13-11 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.
$
```





14

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



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.

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.

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. This includes removing matrices or tables that would be affected by the changes and retrieving those not affected and then recalculating only what is necessary. No user directions are required. 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. 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](#)) and read-only restart ([Read-Only Restarts](#)).

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, 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 .	
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

For more information regarding the MSC Nastran input file structure, see [Organization of MSC Nastran Files](#).

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 the [Chapter 1: Introduction](#) for the exact syntax for your



machine. For a typical Linux 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 three database files (see [Overview of the MSC Nastran Input File](#)) are created as a result of the above command:

```
stat1.DBALL
stat1.MASTER
stat1.IFPDAT
```

Restart Run

NASTRAN Statement Section

This section is normally the same as your cold start run. Note that the BUFSIZE must not be changed in a restart run. See [Database Management](#) 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 Linux 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. (See MSC_DOC_DIR/doc/linstat/stat1.dat)

Listing 14-1 Input File for a Cold Start Run

```
$  
$   filename -  stat1.dat  
$  
ID ROD,TEST  
SOL 101  
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. (See `MSC_DOC_DIR/doc/linstat/stat2.dat`) 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 an 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 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  
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. (See `MSC_DOC_DIR/doc/linstat/stat3.dat`) The same two loading conditions are used as run number two. A replacement material entry is added in this run and the old material entry, from the stat2.dat run, is deleted. See the `/ Bulk Data` entry. Since `UPDATE` is the default for processing entries on restarts, only the replacement material entry is needed in the Bulk Data Section.

You can have as many of these entries as you need.

The unsorted Bulk Data is output by default. Use 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`).

The general format for deleting an entry from the database is

```
/DELETE NAME
```

in this case it would be

```
/DELETE MAT1
```



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:

```
/DELETE ALL
$
$    include a complete new bulk data listing
$
```

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
DIAG 8
CEND
TITLE= RESTART RUN
$
echo = both
SPC = 1
$
SUBCASE 1
LABEL = ADDITIONAL STRESS OUTPUT REQUEST
LOAD = 1
DISP = ALL
STRESS = ALL
$
SUBCASE 2
LABEL = ADDITIONAL LOAD CASE
LOAD = 2
DISP = ALL
STRESS = ALL
$
BEGIN BULK
$
$/delete mat1 $ will delete all the mat1 entries
$ or just adding the entry below will replace the "mat1 1"
entry
mat1,1,3.+7,,.32
$
ENDDATA
```

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 update or 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.

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='dddddd.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](#).

Read-Only Restarts

Read-only restart is very similar to read-write restart as described in [User Interface for Read-Write Restart](#) 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.

Mini-Database Restarts

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, use the DBS= or RESTART+ assign method to select your cold start database, modify your Case Control output requests and remove your Bulk Data entries. The restart 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:



DISPLACEMENT	STRAIN
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, 557](#)) the HOUTPUT and NOUTPUT cannot be changed in the restart run.

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 14-4](#)) 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 14-4 Input File for Printing/Punching a Listing of Bulk Data File from the Database

```
$  
$ FILE - bulklist.dat  
$  
restart  
assign master='STAT1.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 time--by not having to resolve a problem from the beginning. One disadvantage is that you have to save the database. 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.



15

Database Management

- Introduction to Database Management
- MSC Nastran Database
- File Management Section
- Guidelines for Large Problems



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 contains the basic use of FMS statements, for complex, large problems see [Using the Basic Functions of MSC Nastran](#) (Ch. 4), [Using the Advanced Functions of MSC Nastran](#) (Ch. 5) in the and [Database Concepts](#) (Ch. 10) in the *MSC Nastran Reference Guide*.

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.
BUFFSIZE	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 BUFFSIZE is machine dependent. The default value is recommended except for large problems. It can be modified by using the following NASTRAN statement: NASTRAN BUFFSIZE = xxxxx
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 BUFFSIZE-1 words. (GINO is an acronym for general input/output.)
{ }	A brace indicates that the quantity within this bracket is mandatory.
[]	A square bracket indicates that the quantity within this bracket is optional.

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 three scratch (SCRATCH, SGR300, and OBJSCR) DBset are created during the run. The default blocks shown is for the smallest memory and bufsize, see [Table 4-6](#) in the *MSC Nastran Installation and Operations Guide* for the various default sizes.



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.
OBJSCR	Temporary DBset for DMAP compilation. The default size is 5000 blocks.

For a typical Linux-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.

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:

- Byte (B), Word (W)
- Kilobyte (KB), Kiloword (K or KW)
- Megabyte (MB), Megaword (M or MW)
- Gigabyte (GB), Gigaword (G or GW)



Terabyte (TB) or Teraword (T or TW)

MSC Nastran BUFFSIZE 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

Creates/initializes permanent and/or temporary DBsets. The File Management statement **INIT** in the *MSC Nastran Quick Reference Guide* 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-siznen))]
```

DBset-name	The logical name of the DBset being used (e.g., DBALL).
log-namei	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-sizei	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.

Unless you need to split DBALL into logical/physical files, the SDBALL=size nastran command keyword (see [Keywords and Environment Variables](#) in the *MSC Nastran Installation and Operations Guide*) is a quick way to set DBALL to a specific size, such as SDBSIZE=50000.

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-siznen))]
```

Log-name1 through log-namei 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.

Again, unless you need to split SCRATCH into logical/physical files, the SCCR=size nastran command keyword (see [Keywords and Environment Variables](#) in the *MSC Nastran Installation and Operations Guide*) is a quick way to set SCRATCH to a specific size, such as SCCR=800MW.

ASSIGN

The File Management statement [ASSIGN](#) in the *MSC Nastran Installation and Operations Guide* 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. Generally this is used when a file will not fit on one disk drive.

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 Installation and Operations Guide*.

EXPAND

The File Management statement **EXPAND** in the *MSC Nastran Quick Reference Guide* 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-namei	The logical name of the i-th member of the DBset. An ASSIGN statement should be used to point this logical name to a physical file.
max-sizei	The maximum size of the 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

The File Management statement **RESTART** in the *MSC Nastran Installation and Operations Guide* 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

The File Management statement **DBCLEAN** in the *MSC Nastran Installation and Operations Guide* 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

The File Management statement **DBDIR** in the *MSC Nastran Installation and Operations Guide* 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

The File Management statement **INCLUDE** in the *MSC Nastran Installation and Operations Guide* 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 editing and disk space by using the INCLUDE statement instead of having multiple files with duplicate input data.

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. Resources for computers are much more plentiful than when NASTRAN was initially developed, so only very large problems need resource estimation. 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 Linux 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
$ 
$ in file 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. An equivalent submittal command is:

```
NASTRAN RUNA DBALL=300000 SSCR=400000 SCR=NO
```

and the INIT FMS statements are not needed.

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
$ 
$ in file runb.dat

INIT DBALL,LOGICAL=(DBALL(50MB))
INIT SCRATCH,LOGICAL=(SCRATCH(500MB)) ,
    SCR300=(SCR300(800MB))
$ 
$ THE REST OF YOUR INPUT FILE
$ 
```

or

```
NASTRAN RUNB SSCR=1.3GB SCR=YES
```



Example K

If you have sufficient disk space in your current and default scratch file systems (disk packs), the methods shown in Example I or Example J are the preferred procedures since it is the simplest method. However, if you do not have sufficient disk space in a file system, 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
$
$ in file runc.dat
$
ASSIGN DBALL1='runc.db1'
ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrtch2'
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.scrtch2" 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.

Example L

Even though the setup in Example K looks virtually the same as that in Example L, 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.

```

nastran rund scr=yes
$ in file rund.dat
$ ASSIGN DBALL1='rund.db1'
ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrtch2'
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 (1) 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](#)).

```
NASTRAN BUFFSIZE=xxxxx
```

where xxxx = $(n \cdot \text{disk-block-size}) + 1 \leq \text{limit}$ and n is a positive integer. Default is 8193.

The acceptable values of disk-block-size and limit are machine dependent and are defined in the *MSC Nastran Installation and Operations Guide*.

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.

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:

$$\text{Number of words} = \text{Number of GINO blocks} \cdot (\text{BUFFSIZE} - 1) \quad (15-1)$$

Example M illustrates the use of a non-default BUFFSIZE.

Example M



Assume the machine that you are running on has a default BUFFSIZE of 8193. If you increase the BUFFSIZE to 65537, using [Equation \(15-1\)](#), this example allocates the same maximum amount of physical disk space as Example I, which uses the default BUFFSIZE.

```
$  
$ in file rune.dat  
  
NASTRAN BUFFSIZE=65537  
INIT DBALL,LOGICAL=(DBALL(37500))  
INIT SCRATCH,LOGICAL=(SCRATCH(25000)),SCR300=(SCR300(25000))  
$  
$ THE REST OF YOUR INPUT FILE  
$
```

BUFFSIZE=65537 is the maximum allowed. Total memory allocation can be controlled by the “mem” keyword on the submittal line. The following example allocates 2 GB of total memory for the run.

```
nastran stat1 mem=2GB
```

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 [Equation \(15-2\)](#).

$$\text{User OPENCORE} \cong \text{mem} - (\text{BUFFSIZE} \cdot 150) - 500\text{KW} \quad (15-2)$$

As you can see for [Equation \(15-2\)](#), 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. You can also let MSC Nastran estimate the amount of required memory by using the following submittal command:

```
nastran stat1 mem=estimate
```

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.



16

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



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).

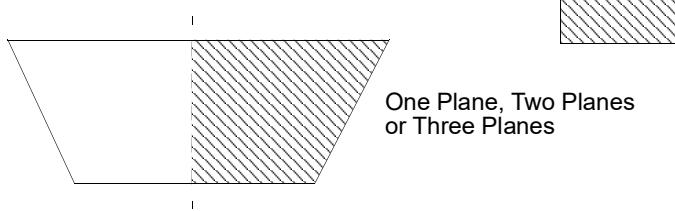
MSC Nastran also includes a way of handling the special case of unsymmetric loads on a structure with one or two planes of symmetry (discussed in [Modeling Guidelines](#)). With this method you must perform Tasks 1 and 2 in the above list by hand. Task 3 is accomplished by creating subcases for each symmetry condition, with different SPC sets and loading conditions for each subcase. Task 4 is accomplished with [SYMCOM \(Case\)](#) and [SYMSEQ \(Case\)](#) Case Control commands or with [SUBCOM \(Case\)](#) and [SUBSEQ \(Case\)](#) Case Control commands (see [Organization of MSC Nastran Files](#)).

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](#).

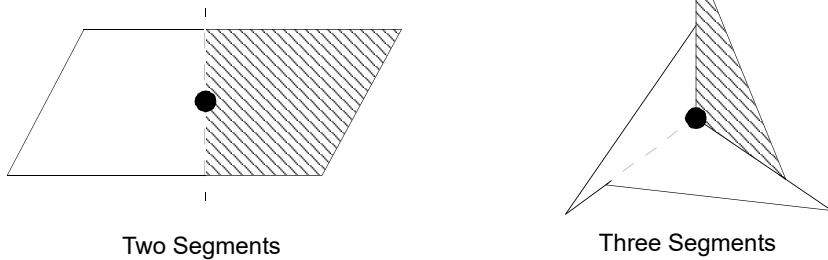


(a) Reflective Symmetry



Fundamental Region

(b) Rotational Symmetry



(c) Dihedral Symmetry

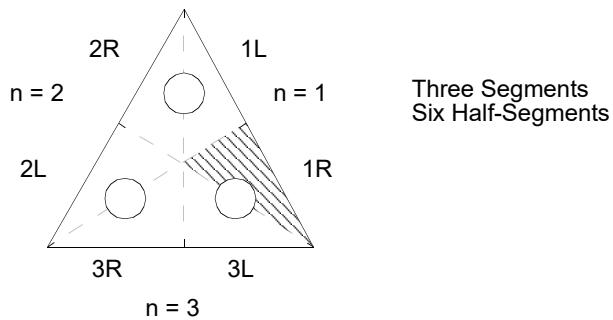
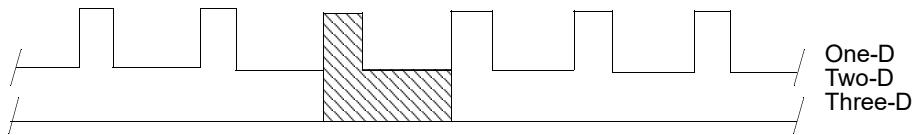


Figure 16-1 Types of Geometrical Symmetry



(d) Translational Symmetry



(e) Axisymmetry

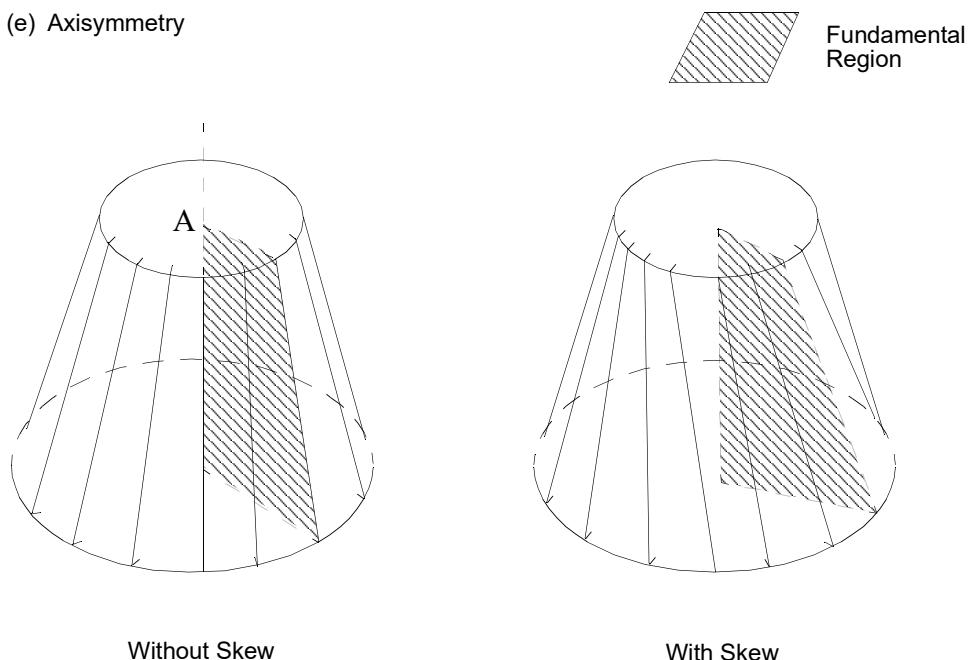


Figure 16-1 Types of Geometrical Symmetry (continued)

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.



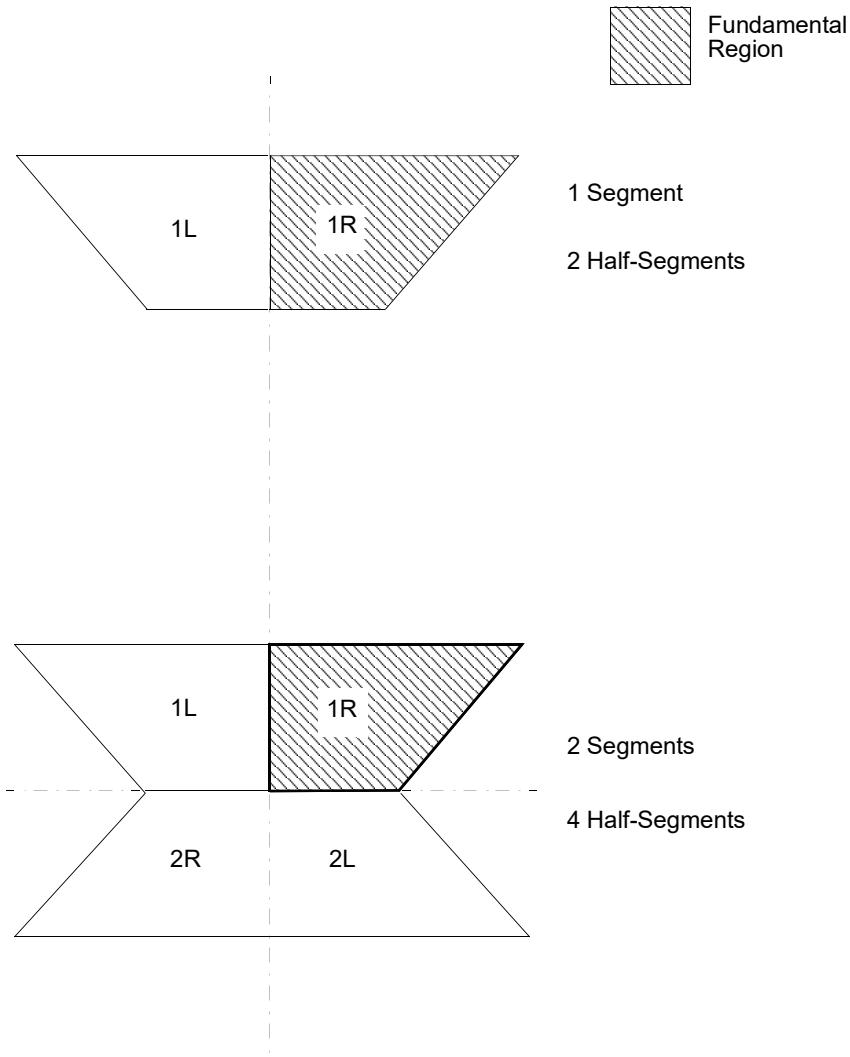


Figure 16-2 Reflective Symmetry with One or Two Planes of Symmetry --
A Special Case of Dihedral Symmetry

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 [Figure 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 SOL 114 is discussed in this Guide.



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

$$Q(x, y, z) = Q(-x, y, z) \quad (16-1)$$

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 [Equation \(16-1\)](#), but their average u_s clearly does. Thus,

$$u_s = \frac{1}{2}(u_r + u_l) \quad (16-2)$$

so that

$$u_s(x, y, z) = \frac{1}{2}(u(x, y, z) + u(-x, y, z)) \quad (16-3)$$

and

$$u_s(-x, y, z) = \frac{1}{2}(u(-x, y, z) + u(x, y, z)) \quad (16-4)$$

Since the right-hand sides of [Equation \(16-3\)](#) and [Equation \(16-4\)](#) are equal, it follows that

$$u_s(x, y, z) = u_s(-x, y, z) \quad (16-5)$$

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,



$$u_a = \frac{1}{2}(u_r - u_l) \quad (16-6)$$

The antisymmetrical part of the response has the property that

$$u_a(x, y, z) = -u_a(-x, y, z) \quad (16-7)$$

[Equation \(16-2\)](#) and [Equation \(16-6\)](#) 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

$$\begin{Bmatrix} u_s \\ u_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} u_r \\ u_l \end{Bmatrix} \quad (16-8)$$

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 [Equation \(16-8\)](#).

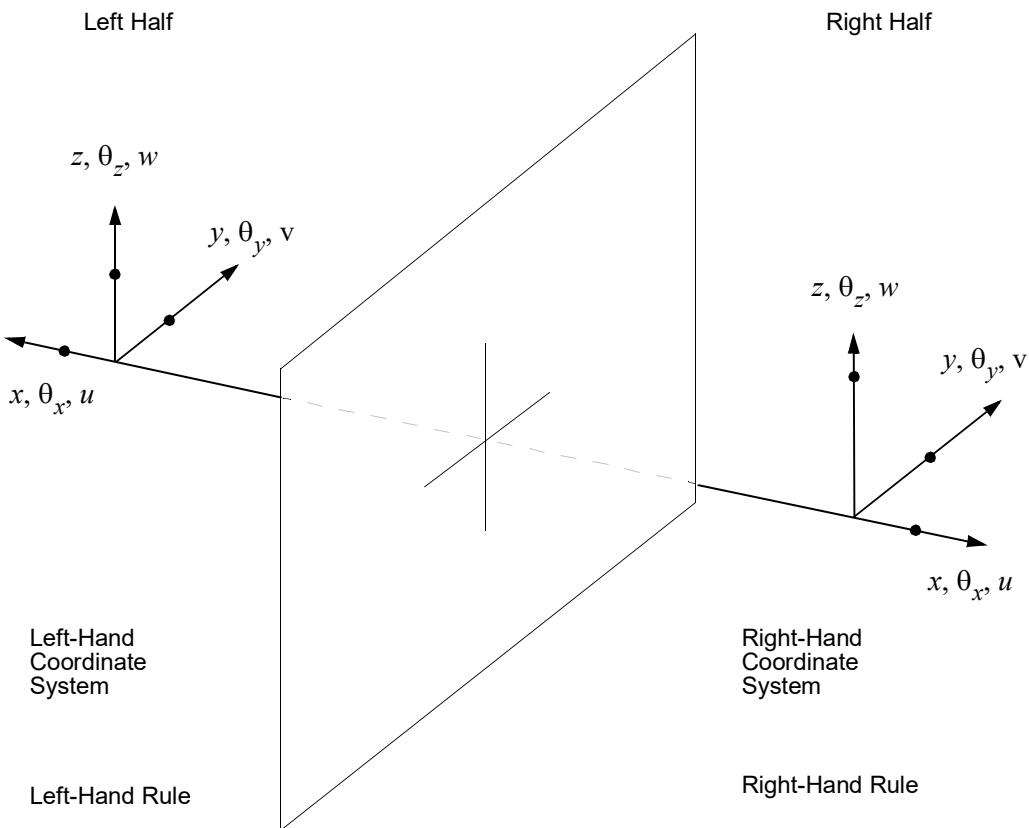
$$\begin{Bmatrix} u_r \\ u_l \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} u_s \\ u_a \end{Bmatrix} \quad (16-9)$$

[Equation \(16-8\)](#) and [Equation \(16-9\)](#) 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:



$$\begin{Bmatrix} P_s \\ P_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} P_r \\ P_l \end{Bmatrix} \quad (16-10)$$

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 [Equation \(16-10\)](#).
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 [Equation \(16-9\)](#).

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](#) (Ch. 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

$$Q(r, \theta, z) = Q(r, 0, z) \quad (16-11)$$

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 [Equation \(16-11\)](#).



Note: Axisymmetric without skew is the terminology introduced in [Description of Geometric Symmetry in MSC Nastran, 558](#),

The response of the structure and its applied loads do not necessarily satisfy [Equation \(16-11\)](#). 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 be the cylindrical coordinate system used in connection with [Equation \(16-11\)](#). Then the Fourier series representation of $u(r, \theta, z)$ may be written as

$$u(r, \theta, z) = \bar{u}^0(r, z) + \sum_{k=1}^{\infty} (\bar{u}^{ks}(r, z) \sin(k\theta) + \bar{u}^{kc}(r, z) \cos(k\theta)) \quad (16-12)$$

or, dropping the explicit dependence on r and z ,

$$u(\theta) = \bar{u}^0 + \sum_{k=1}^{\infty} (\bar{u}^{ks} \sin(k\theta) + \bar{u}^{kc} \cos(k\theta)) \quad (16-13)$$

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:

$$\begin{aligned} \bar{u}^0 &= \frac{1}{2\pi} \int_0^{2\pi} u(\theta) d\theta \\ \bar{u}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \sin(k\theta) d\theta \\ \bar{u}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \cos(k\theta) d\theta \end{aligned} \quad (16-14)$$

Applied loads $P(\theta)$ can also be expanded in a Fourier series. The symmetrical component load sets are, by analogy with [Equation \(16-14\)](#),



$$\begin{aligned}\bar{P}^0 &= \frac{1}{2\pi} \int_0^{2\pi} P(\theta) d\theta \\ \bar{P}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \sin(k\theta) d\theta \\ \bar{P}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \cos(k\theta) d\theta\end{aligned}\tag{16-15}$$

Fourier series are often used in the analysis of axisymmetric structures. The first step is to find the harmonic load coefficients from [Equation \(16-15\)](#). 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 [Equation \(16-12\)](#).

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](#),



$$Q(\theta + \theta_n) = Q(\theta) \quad (16-16)$$

where:

$$\theta_n = \frac{2\pi(n-1)}{N} \quad \text{and } n = 1, 2, 3, \dots, N.$$

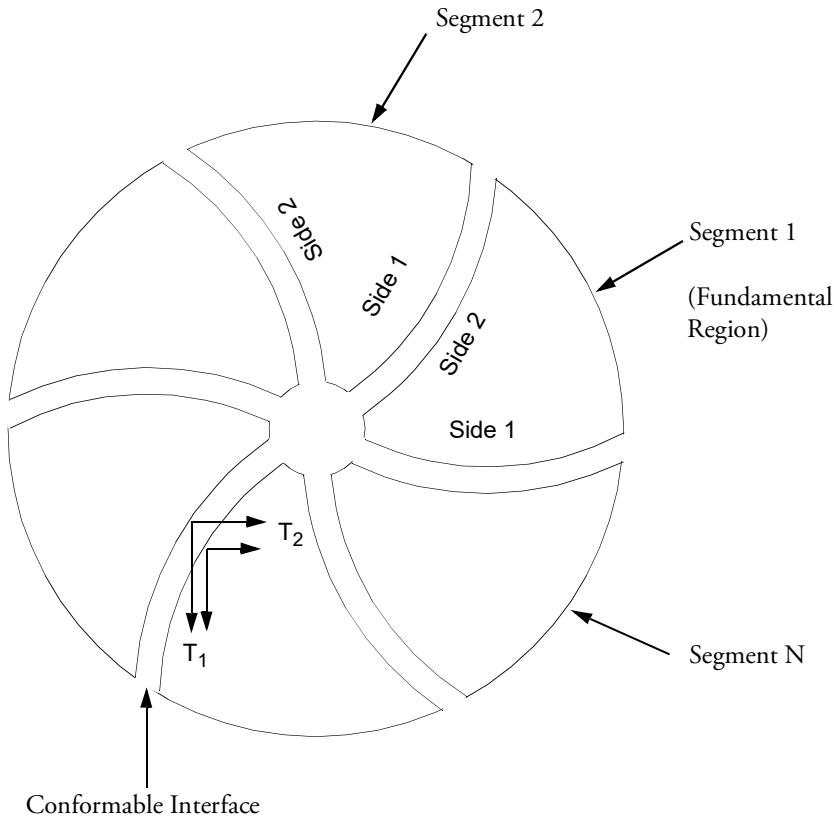


Figure 16-4 Conforming Boundaries for the ROT Option

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

$$u^n(p) = \bar{u}^0(p) + \sum_{k=1}^{k_L} (\bar{u}^{ks}(p)\sin(k\theta_n) + \bar{u}^{kc}(p)\cos(k\theta_n)) + (-1)^{n-1} u^{N/2}(p) \quad (16-17)$$

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 (ϕ) is dropped for convenience.

$$\begin{aligned} \bar{u}^0 &= \frac{1}{N} \sum_{n=1}^N u^n \\ \bar{u}^{ks} &= \frac{2}{N} \sum_{n=1}^N u^n \sin(k\theta_n) \quad k = 1, 2, \dots, k_L \\ \bar{u}^{kc} &= \frac{2}{N} \sum_{n=1}^N u^n \cos(k\theta_n) \quad k = 1, 2, \dots, k_L \\ \bar{u}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} u^n \end{aligned} \quad (16-18)$$



You should note that stiffness damping, and mass matrices become uncoupled for \bar{u}^k .

[Equation \(16-18\)](#) also applies to the expansion of applied loads into symmetrical component sets. Thus

$$\begin{aligned}\bar{P}^0 &= \frac{1}{N} \sum_{n=1}^N P^n \\ \bar{P}^{ks} &= \frac{2}{N} \sum_{n=1}^N P^n \sin(k\theta_n) \quad k = 1, 2, \dots, k_L \\ \bar{P}^{kc} &= \frac{2}{N} \sum_{n=1}^N P^n \cos(k\theta_n) \quad k = 1, 2, \dots, k_L \\ \bar{P}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} P^n\end{aligned}\tag{16-19}$$

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 [Equation \(16-16\)](#). This fact leads directly to an efficient method of analysis in which each symmetric load component is computed from [Equation \(16-19\)](#) 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).



$$\begin{aligned}\bar{u}_1^0 &= \bar{u}_2^0 \\ \bar{u}_1^{kc} \cos(ka) + \bar{u}_1^{ks} \sin(ka) &= \bar{u}_2^{kc} \\ -\bar{u}_1^{kc} \sin(ka) + \bar{u}_1^{ks} \cos(ka) &= \bar{u}_2^{kc} \\ k &= 1, 2, \dots, k_L \\ -\bar{u}_1^{N/2} &= \bar{u}_2^{N/2}\end{aligned}\tag{16-20}$$

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.



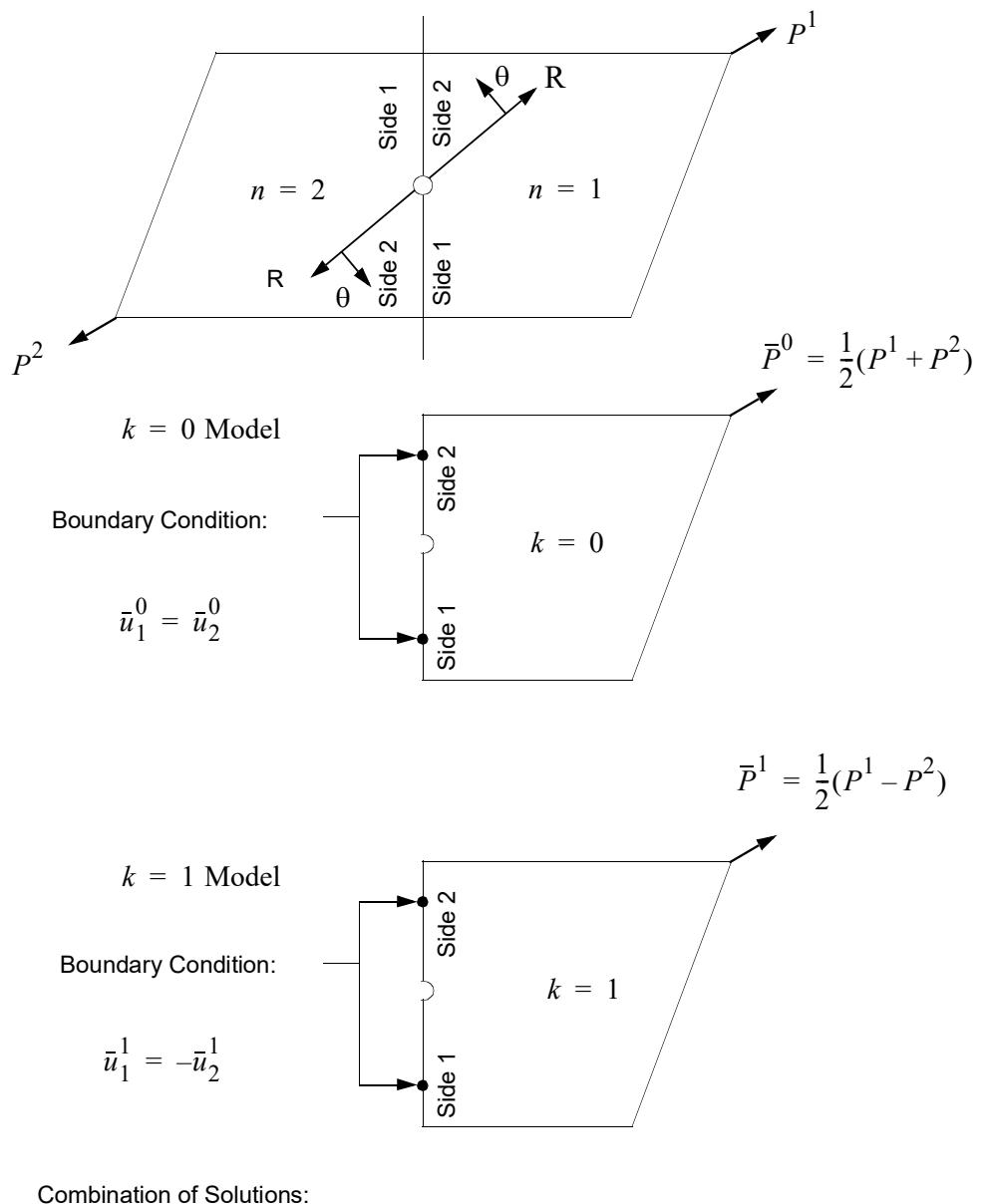


Figure 16-5

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 [Equation \(16-19\)](#), are

$$\bar{P}^0 = \frac{1}{2}(P^1 + P^2) \quad (16-21)$$

$$\bar{P}^1 = \frac{1}{2}(P^1 - P^2)$$

\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 [Equation \(16-20\)](#), are

$$\begin{aligned}\bar{u}_1^0 &= \bar{u}_2^0 \\ \bar{u}_1^1 &= -\bar{u}_2^1\end{aligned} \quad (16-22)$$

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 [Equation \(16-17\)](#)

$$\begin{aligned}u^1 &= \bar{u}^0 + \bar{u}^1 \\ u^2 &= \bar{u}^0 - \bar{u}^1\end{aligned} \quad (16-23)$$

The static condensation procedure described in [Matrix Operations, 469](#) 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.



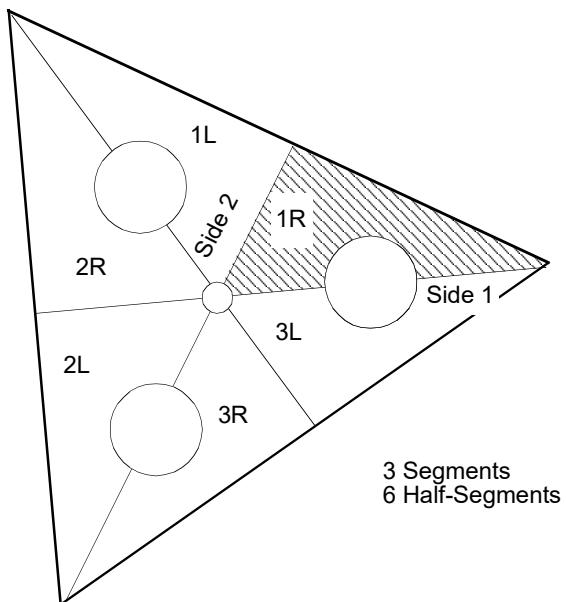


Figure 16-6 Dihedral Symmetry

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

$$u^{nR} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} + \bar{u}^{kc*}) \cos(k\theta_n) + (\bar{u}^{ks} + \bar{u}^{ks*}) \sin(k\theta_n)] \quad (16-24)$$

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} , and \bar{u}^{ks*}

are symmetrical component sets. The series for response quantities in the left half-segments is

$$u^{nL} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} - \bar{u}^{kc*}) \cos(k(\theta_n + a)) - (\bar{u}^{ks} - \bar{u}^{ks*}) \sin(k(\theta_n + a))] \quad (16-25)$$

where, in addition to previously defined terms, $a = 2\pi/N$.

The symmetrical component sets are evaluated from the physical components as follows:

$$\begin{Bmatrix} \bar{u}^{kc} \\ \bar{u}^{kc*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \cos(k\theta_n) \pm u^{nL} \cos(k(\theta_n + a))] \quad (16-26)$$

$$\begin{Bmatrix} \bar{u}^{ks} \\ \bar{u}^{ks*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \sin(k\theta_n) \mp u^{nL} \sin(k(\theta_n + a))] \quad (16-27)$$

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 [Equation \(16-26\)](#) and [Equation \(16-27\)](#), 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 $\alpha = \pi$.
- $\theta_1 = 0$.
- $\theta_2 = \pi$.
- The harmonic indices $k = 0, 1$.

Substitution of these values into [Equation \(16-26\)](#) and [Equation \(16-27\)](#) gives

$$\begin{aligned}\bar{u}^{0c} &= \frac{1}{4}(u^{1R} + u^{1L} + u^{2R} + u^{2L}) \\ \bar{u}^{1c} &= \frac{1}{4}(u^{1R} - u^{1L} - u^{2R} + u^{2L}) \\ \bar{u}^{0c*} &= \frac{1}{4}(u^{1R} - u^{1L} + u^{2R} - u^{2L}) \\ \bar{u}^{1c*} &= \frac{1}{4}(u^{1R} + u^{1L} - u^{2R} - u^{2L}) \\ \bar{u}^{0s} &= \bar{u}^{0s*} = \bar{u}^{1s} = \bar{u}^{1s*} = 0\end{aligned}\tag{16-28}$$

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.

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.

CYJOIN, CYSYM, CYAX, and CYSUP Bulk Data Entries

The boundary points of a segment in cyclic symmetry problems using the [CYJOIN](#) entry are as follows:

CYJOIN

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.

- 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.

- 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.
- 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.
SCALAR	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:

CYSYM

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:

CYAX



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 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 α -set.

The [CYSUP](#) Bulk Data entry provides for free body supports in cyclic symmetry analysis.

The format of the CYSUP entry is as follows:

CYSUP

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](#).



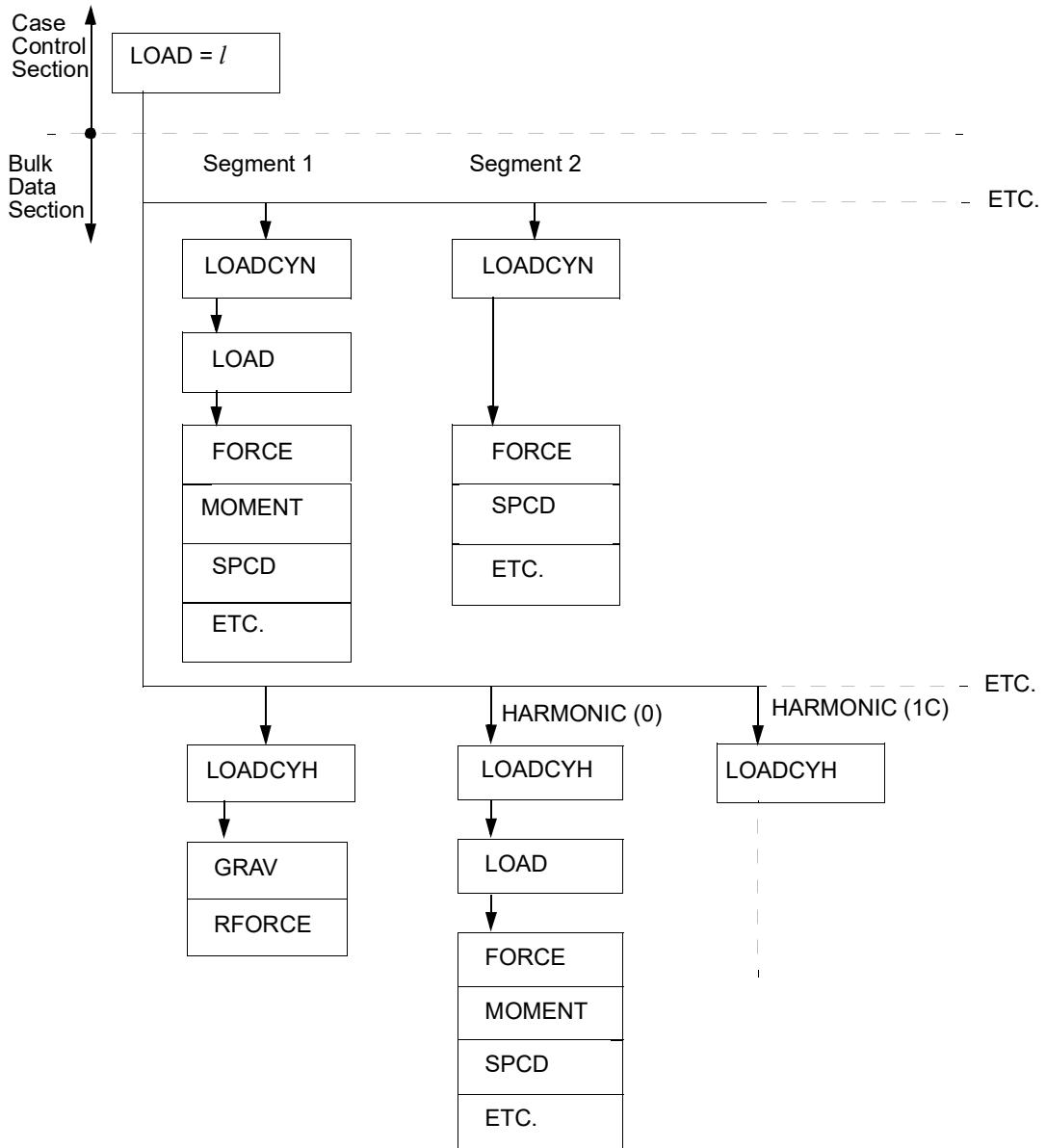


Figure 16-7 Cyclic Symmetry Flowchart

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:

LOADCYN



1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYP	S1	L1	S2	L2	

Field	Contents
SID	Load set identification number.
S	Scale factor.
SEGID	Segment identification number.
SEGTYP	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		TEM	
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:

LOADCYH



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:

LOADCYT

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.
TABLEID _i	Table ID for table load input for load set Li.
LOADSET _i	Load set Li.
METHOD _i	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.

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 (Case)** 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)** 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 (Case)** and **NOUTPUT (Case)** 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 [Figure 16-11](#) and [Listing 16-2](#), respectively.



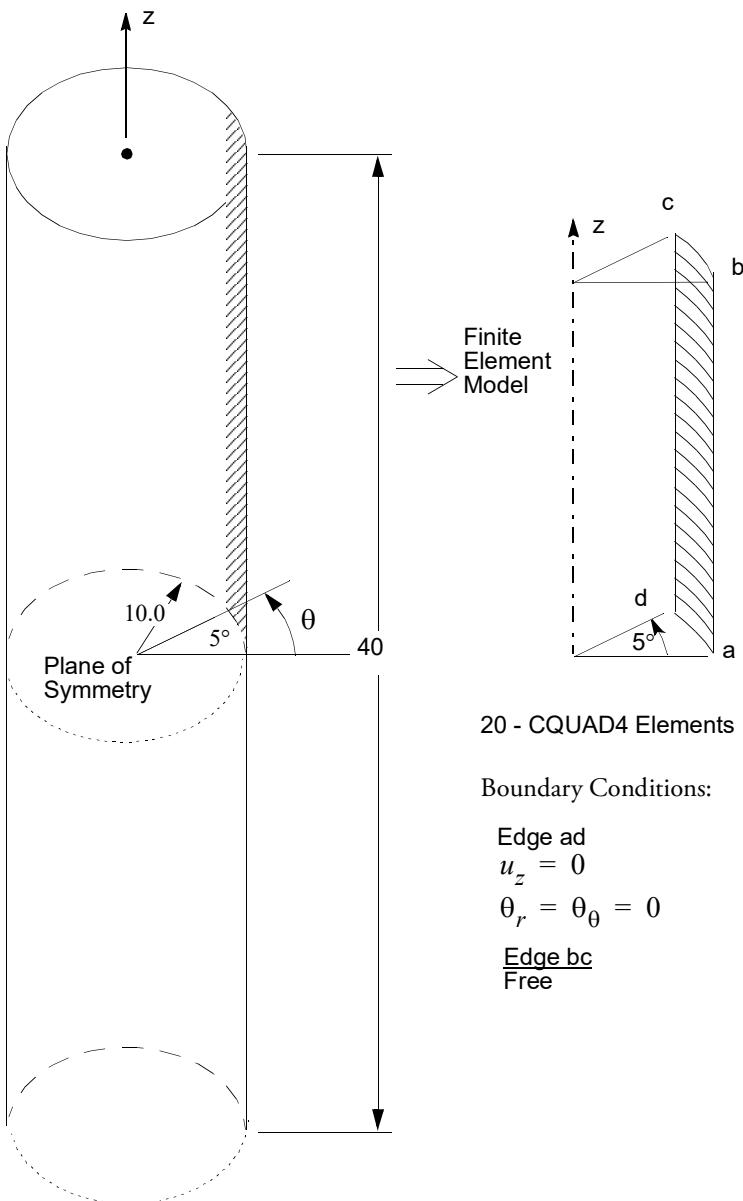


Figure 16-8 Cylinder Example



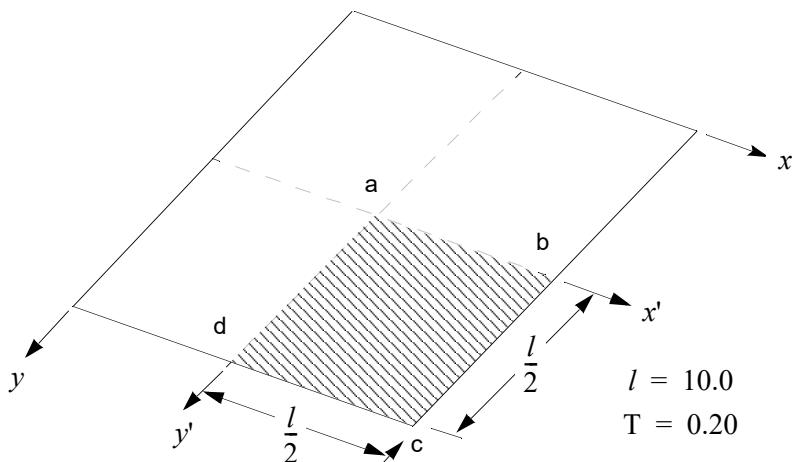


Figure 16-9 Square Plate Example

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 NOUTPUT 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.

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, 523](#). (See MSC_DOC_DIR/doc/linstat/cyclic2.dat)

A five degrees strip, as shown in [Figure 16-10](#), is modeled using the axisymmetric features.



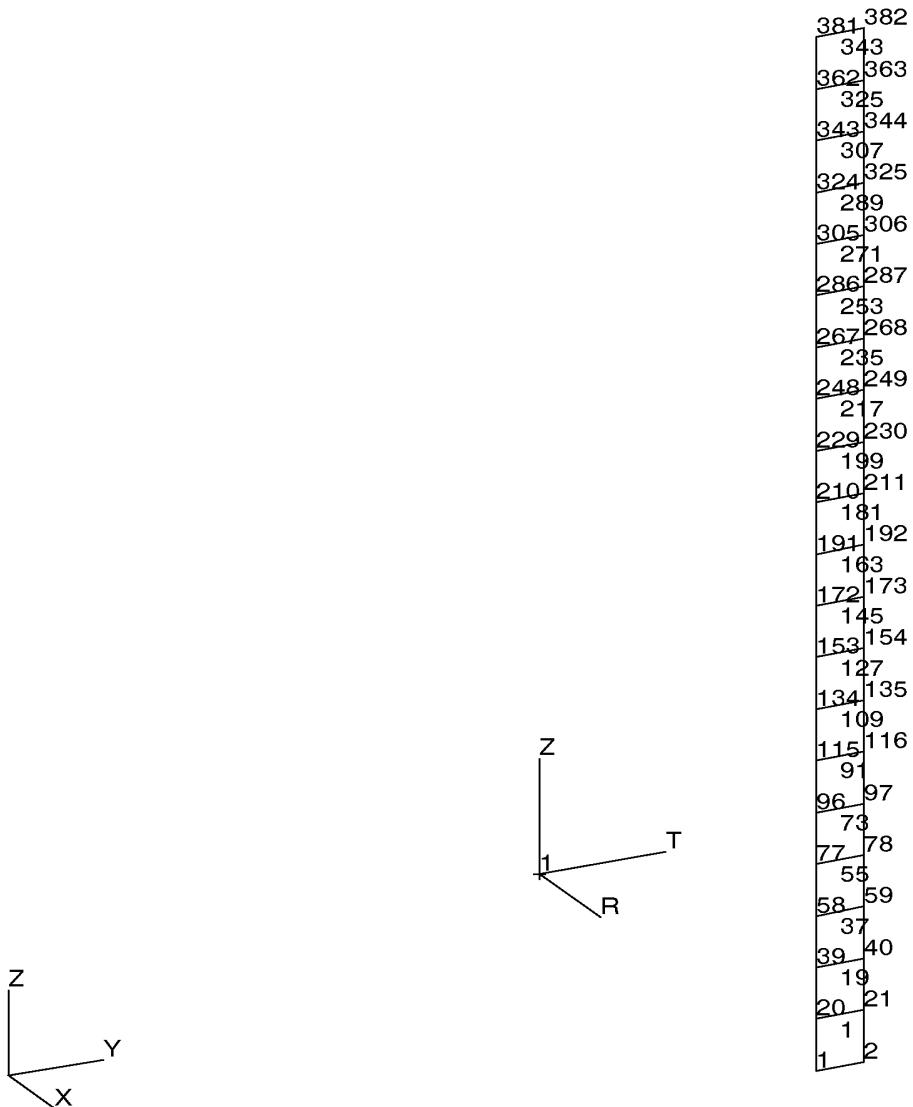


Figure 16-10 A Five Degrees Strip of a Cylinder Using Axisymmetry

The input file is shown in [Figure 16-3](#). (See `MSC_DOC_DIR/doc/linstat/cyclic1.dat`) 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 16-12](#). The same output using the full model is shown in [Figure 16-13](#) for comparison.

Figure 16-11 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.
$
PSHELL 1 1 .03 1 1 1 0. 0.
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
GRID   192     1      10.   5.     10.1
GRID   210     1      10.   0.     11.1
GRID   211     1      10.   5.     11.1

```

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									
ELEMENT	FIBRE	STRESSES IN QUADRILATERAL ELEMENT COORD SYSTEM			ELEMENTS (QUAD4)					
ID.	DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	PRINCIPAL STRESSES (ZERO SHEAR)		
0	1 -1.500000E-02	-7.481326E+03	-5.592213E+04	2.894851E-10	.0000	-7.481326E+03	-5.592213E+04	VON MISES		
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 16-12 Abridged Output Using Axisymmetry



DISPLACEMENT VECTOR									
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		
STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)									
ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)				
0 1 5.258343E+04	-1.500000E-02	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		VON MISES
4.759125E+04	1.500000E-02	-7.482788E+03	-5.592399E+04	-1.636876E-10	.0000	-7.482788E+03	-5.592399E+04		
0 343 5.258343E+04	-1.500000E-02	-7.482788E+03	-5.592399E+04	-4.949596E-10	.0000	-7.482788E+03	-5.592399E+04		
4.759125E+04	1.500000E-02	-5.769486E+03	-5.021298E+04	-2.932691E-10	.0000	-5.769486E+03	-5.021298E+04		

Figure 16-13 Abridged Output Using Full Model





17

Contact in Linear Static Analysis

- Introduction
- Simplified Support Method
- Contact in SOL 101 and Permanent Glued Contact



Introduction

The contact between parts is an important engineering process that requires analysis. This contact may be due to physical gaps in the geometry, interference fits, or do to the assembly process. In the most general sense contact is a nonlinear process potentially involving large geometric relative motion, friction and separation. MSC Nastran has multiple ways to address this phenomena. This includes:

1. General nonlinear contact capability
2. Nonlinear gap element CGAP
3. Slideline technology
4. Simplified Support
5. Contact in SOL 101 and Permanent Glue

General nonlinear contact capability is available in SOL 400 and SOL 700 and will not be discussed here. The nonlinear gap element (CGAP) is available in SOL 106, SOL 129 and SOL 400. In SOL 400 it should not be used in conjunction with the general nonlinear contact capability. While the CGAP element has some useful capabilities it is not recommended when large sliding occurs over the surfaces. Slideline technology (bulk data options BLSEG and BWIDTH) are available in SOL 106 and SOL 129 and is not the recommended approach. Note that SOL 700 also uses Slideline technology and is a general capability for explicit dynamic nonlinear analysis. The Simplified Support method and Contact may be used in SOL 101 and are discussed in this chapter. Besides the determination of contact no other material or geometric nonlinearity may occur in the analysis. The contact conditions are determined using an iterative procedure.

Simplified Support Method

The constraints are applied to grid points or spoints. The constraint ensures that:

1. The relative displacement (UR- relative distance between grid points on each contact surface) 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 contact 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 user can specify a starting vector that defines the open/shut status of the gaps by specifying DMIG, CDSHUT. If the CDSHUT vector is not specified, then all gaps will be assumed to be closed for the initial iteration. A solution is obtained when all the gap constraints are satisfied, i.e., there's no penetration and no tension forces. If job does not converge during the iterations, a new random start vector is generated and the iterative procedure will repeat until the number of iterations reaches the maximum iteration number or there are no more random vectors to try.

This approach provides an alternative method to the use of GAP elements in SOL 106. The advantage is that it is unnecessary to learn either how to calculate the GAP stiffness or 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 arranging one of the degrees-of-freedom of the boundary grid points to represent the motion perpendicular to the boundary. A positive displacement represents the motion away from the boundary. If, on the other hand, the constraint represents



the relative motion between two bodies, MPC equations are needed to define this relative motion degrees-of-freedom, which is then constrained to have a non-negative displacement.

A PARAM,CDITER entry along with a SUPPORT entry are required. The CDPRT and CDPCH parameters are used to control the output and an optional DMIG,CDSHUT can help to control the analysis.

To define CDITER entry, user may use "PARAM, CDITER, n" where n has a default value 0. When n > 0, whose value is the maximum iteration number, means to perform constrained displacement iteration in the analysis.

A SUPPORT entry can be defined as Table 17-1,

Table 17-1

SUPPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4
---------	-----	----	-----	----	-----	----	-----	----

IDi means the identification number of Grid Points. Ci means component number. This SUPPORT entry specifies the reference degrees of freedom for a rigid body motion.

CDPRT is the parameter to control the print of constraint violations during the iterations. When CDPRT=YES (Default), the negative displacements and tension forces which do not satisfy constraints will be printed out.

CDPCH controls the PUNCH output of DMIG, CDSHUT for the final state. When CDPCH=YES, it will create a separate .pch file to print PUNCH output. Default value is "NO"

The linear gap element is supported in Patran under the Utilities Menu as shown in [Figure 17-1](#).



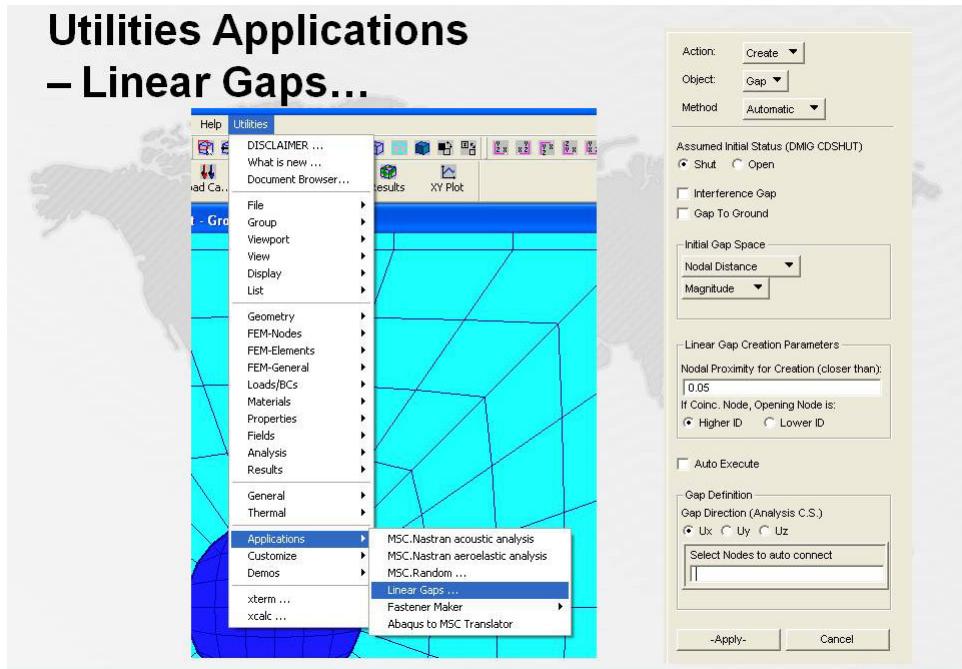


Figure 17-1 Patran Linear Gap

The linear gap utility in Patran provides a convenient method of generating the linear gap elements by graphically selecting the GRIDs that will be used in the formulation of the MPC equations.

The 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 output should be examined to ensure that the job has converged in the given iterations. 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 explanations are included in the actual examples.

```
DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1 CD_1

      QRI
POINT   VALUE     POINT   VALUE     POINT   VALUE     POINT   VALUE
COLUMN      1
      5 T2 -2.37141E-02
^END ITERATION 3 , CHANGES 1
^END ITERATION 4 , CHANGES 0
^USER INFORMATION MESSAGE 9097 (CSTRUISP)
^CONSTRAINED DISPLACEMENT ITERATIONS CONVERGED LOAD CASE = 1
```

The following is a list of limitations for the linear gap elements.

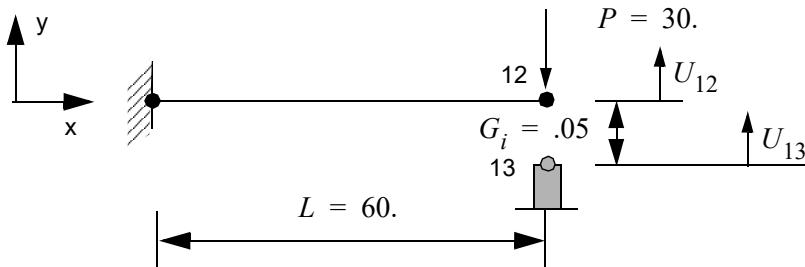
- The only nonlinearity allowed is the constrained displacements.
- There is no sliding friction.



- Free bodies--using the SUPPORT 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 will be issued if both parameters are present.
- There is no guarantee that the solution will converge or that all systems will follow the same path.

Example 1

As an example of using SUPPORT's for contact, please see the following diagram. The 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 accomplishes this goal.

$$S_{51} = U_{12} + G_i - U_{13} \quad (17-1)$$

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 grid 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. Equation (17-1) can be simplified and recast as

$$-U_{12} + S_{51} - G_{101} = 0 \quad (17-2)$$

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 17-1](#). (See `MSC_DOC_DIR/doc/linstat/cd_0.dat`)

Listing 17-1 Input file for Cantilever Beam with Stopper.

```
$  
$     filename = 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  
suport,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,cdprt,yes  
$  
enddata
```

[Equation \(17-2\)](#) is represented by “MPC, 77” in [Listing 17-1](#). Scalar point 51 must also be placed on the SUPPORT 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 will 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 [Listing 17-2](#). The large non-zero values for epsilon and strain energy for the SUPPORT 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.

```
^/^ END ITERATION      1,  CHANGES      0
```

The gap is in contact as indicated by the following scalar point 51 output.

POINT	SHUT COLUMN	POINT	POINT	POINT	POINT	POINT	POINT	POINT	POINT
POINT	VALUE	POINT	VALUE	POINT	VALUE	POINT	VALUE	POINT	VALUE
51	1								
	51	S	1.00000E+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:

$$P_g = \frac{3 \cdot EI \cdot U_{12}}{L^3} = \frac{3(3E7)(1)(0.05)}{60 \cdot 60 \cdot 60} = 20.833 \quad (17-3)$$

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.

Listing 17-2 Abridged Output of Cantilever Beam with Stopper.

```

0
*** 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 2.500000E-01 2.083333E+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 0.000000E+00 1.562500E+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

0
0
POINT SHUT
POINT VALUE POINT VALUE POINT VALUE POINT VALUE POINT VALUE
COLUMN 1
51 S 1.000000E+00
.
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
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 0.0
51 S 0.0 5.000000E-02
101 S
.
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
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
.
F O R C E S O F M U L T I P O I N T C O N S T R A I N T
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

```

Example 2

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.

[Figure 17-2](#) depicts the FE model. The problem is to determine where the lift off occurs along the base (grid points 1 through 7.) This is considered as a planar problem and DOFs 3456 on the GRDSET entry are constrained. “SPC1, 200” applies the symmetric boundary condition at the center line of the structure. The



potential contact points are specified by the seven SUPPORT 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 defined 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 will be created in a separate punch file. PARAM,CDPRT,YES requests the printing of the constraint violations during the iterations, which is the default. Part of the input file for this example is shown in [Listing 17-3](#). (See MSC_DOC_DIR/doc/linstat/cd_1.dat for the whole input)

Listing 17-3 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
SPCF0 = 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
$ SUPPORT  1       2
SUPPORT  2       2
SUPPORT  3       2
SUPPORT  4       2
SUPPORT  5       2
SUPPORT  6       2
SUPPORT  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.
$ geometry data below
```



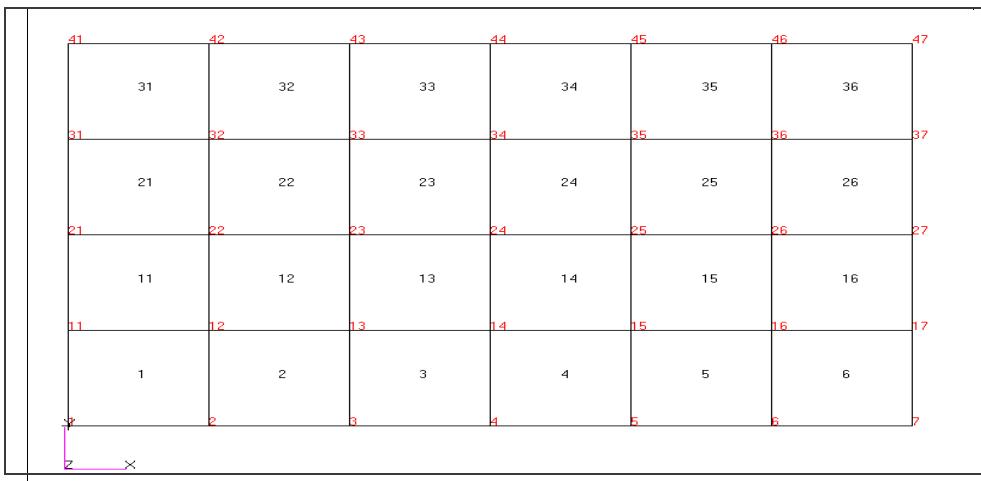


Figure 17-2 Undeformed Plot of the Thick Pad.

The problem converged at the 4th iteration. Looking at the output ([Listing 17-4](#)), it is obvious to 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 which indicates 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.



Listing 17-4 Abridged Output of Thick Pad.

```

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
0
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
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

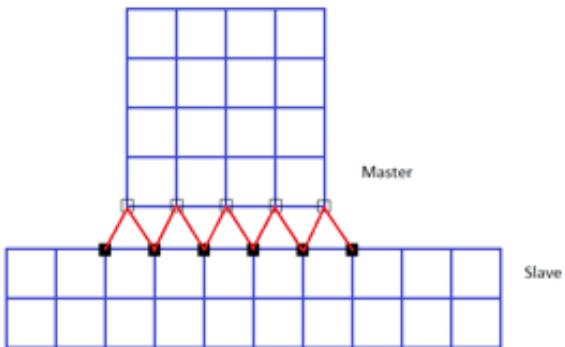
```

Contact in SOL 101 and Permanent Glued Contact

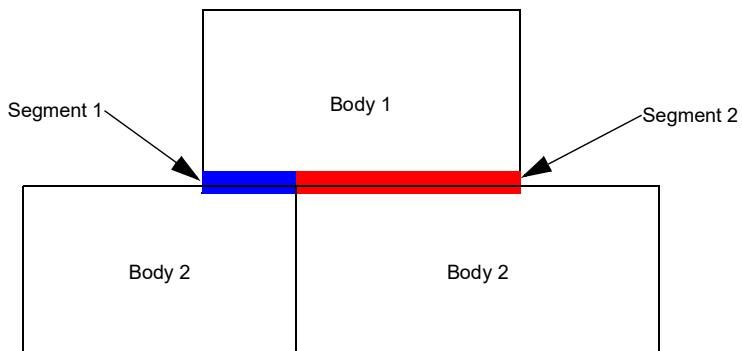
MSC Nastran provides users two contact schemes, Node-to-Segment contact and Segment-to-Segment contact.

For Node-to-Segment (N2S) contact, MSC Nastran will create MPC (Multi-Points Constraints) equations among the nodes which come into contact between Primary body and Secondary body. Then, augmented Lagrange Multiplier Method will be used to embed constraints into analysis see the *MSC Nastran Reference Guide*, [Augmented Lagrange Multiplier Method](#) (Ch. 3) for more information. As shown in the picture below, a secondary body and a primary body have to be defined. Once nodes satisfy the conditions in contact search, they will be linked together by MPC equations. Normally, each active node has three constraint equations, but when beam element or shell element comes into contact, the number of constraint equations increases to 6.





For Segment-to-Segment (S2S) contact, MSC Nastran will introduce auxiliary points for each contact segment for contact search. Each contact segment has a specified number of auxiliary points which are located at fixed positions on contact segments. Both distance and direction detections will be performed for contact search. When segments are in contact, polygons or polylines will be generated. They can be treated as normal interface elements representing the contact stiffness. Then, a weighted stiffness related to a penalty factor is added to a normal stiffness matrix. A tangential stiffness induced by glue contact is similarly treated. For more details on segment-to-segment contact, see the Nonlinear User's Guide, [Segment-to-Segment Contact](#) (Ch. 9).



Contact in SOL 101

Contact in SOL 101 utilizes the full nonlinear contact algorithm of SOL 400 with all the typical linear assumptions of SOL 101, i.e. no material nonlinearity, small displacement, and small strain. This general contact capability has the following features

1. Multiple contact bodies are allowed.
2. Initial contact is unnecessary for the contact bodies.
3. Contact grid points may separate from each other.



4. The grid points of the contacting bodies do not need to be aligned, and the contact algorithm may be used to join dissimilar meshes.
5. Both deformable-deformable and deformable-rigid contact are allowed.¹
6. Friction contact is allowed. Bilinear Coulomb or bilinear shear friction may be applied in the contact analysis. In the general bilinear friction algorithms, the relative tangential displacement is split into stick (elastic contribution) and slip (plastic contribution) components. In SOL 101, only stick (elastic contribution) to relative tangential displacement is computed.

The primary benefit of Contact in SOL 101 is that you can use the full SOL 400 nonlinear contact algorithm and capabilities for “linear” structural models where the only nonlinearity is from the contact. Also, user may keep the same model for both linear and nonlinear analyses. For example, after running a model in SOL 101, if user determines that there are other nonlinear effects, such as material nonlinearity or large rotation, the model can simply be switched to SOL 400 without any requirement to redefine the contact control parameters, tables, and other entries as both SOL 400 and SOL 101 use the same contact definitions.

Full nonlinear contact may output control parameters and iteration messages in the .f04 file and an STS file.

Note that the linear GAP contact defined by PARAM,CDITER,n is still supported, but it should not be used in association with the surfaces defined for general contact.

Permanent Glued Contact Modeling in SOL 101

Permanent Glued contact is a special type of contact model which imposes the condition of no relative (both normal and tangential) motion between the contacting surfaces and no separation occurs.

For Permanent Glued contact, MSC Nastran will form the required constraints without entering the full nonlinear contact algorithm. It should be seen that if there is no initial contact between the contacting bodies, these constraints cannot be formed and the run may fail in permanent glued contact analysis due to insufficient constraints. If the contact surfaces cannot be explicitly established, tolerance of the contact surface may be set and any grid points falling within these tolerances will be GLUED. With this option you can also require that the algorithm bring the surfaces into contact before applying the constraints. The primary benefit of the Permanent Glued contact is to join dissimilar meshes.

Interface to General and Permanent Glued Contact

As with most real world engineering problems, the majority of users will be using a pre- and post-processors like Patran or SimXpert to build the FE model. Of course, the controls and definitions of the contact may be built in the input file directly. Here is the brief description of the interface to contact control and definition.

Input for the General Contact and Permanent Glued contact starts with the Case Control command BC CONTACT in Case Control Section as

BC CONTACT = n

Where n is the Identification number of contact table BCTABLE or BCTABLE1 in Bulk Data Section.

¹Chapter 9 Contact in MSC Nastran Nonlinear User's Guide.



The following commands and entries briefly describe the user interface defined in Bulk Data Section for SOL 101.

Referring the inputs in the first example of this chapter,

```
BCTABL1 1      10
BCONECT 10     11          3      4
BCONPRG 11           ISEARCH 1

$ Deform Body Contact LBC set: left
BCBODY1 3           3D      DEFORM 3      0

BSURF   3      1      2      3      4      5      6      7
        8      9      10     11     12     13     14     15
        16     17     18     19     20     21     22     23
        24     25     26     27     28     29     30     31
        32     33     34     35     36     37     38     39
        40

To analyze a contact problem, users need to define
```

Primary definitions control -

- BCONTACT (Case Control Command) – Selects the contact table.

Primary definitions -

- BCTABL1 (Bulk Data Entry) – Defines a Contact Table
- BCONECT (Bulk Data Entry) – Defines the Touching and Touched Contact Bodies
- BCONPRG (Bulk Data Entry) – Defines geometric contact parameters used to determine if contact occurs
- BCONPRP (Bulk Data Entry) – Defines physical contact parameters of touching bodies
- BCPARA (Bulk Data Entry) – Defines Contact Parameters
- BCBODY1 (Bulk Data Entry) – Defines flexible or rigid contact body in 2D and 3D
- BSURF (Bulk Data Entry) – Defines a contact body or surface

Optional definitions -

- BCRIGID (Bulk Data Entry) – Defines a Rigid Contact Body
- BCRGSRF (Bulk Data Entry) – Defines Rigid Contact Surface
- BCPATCH (Bulk Data Entry) – Defines Rigid Contact Body made up of Quad Patches.
- BCBZIER (Bulk Data Entry) – Defines Rigid Contact Body made up of Bezier Surface
- BCNURB2 (Bulk Data Entry) – Defines a 2D Rigid Contact Body made up of NURBS

Flow chart for the referring relationship



Case Control Command:

BCONTACT = BCID

Bulk Data Entries:

BCTABL1	BCID	ID1	ID2	ID3	ID4	ID5	ID6	ID7
		ID8	-etc..					
BCONNECT	ID	BCGPID	BCPPID	IDSLAVE	IDMASTER			
	"SLAVES"	IDSL1	IDSL2	IDSL3	IDSL4	IDSL5	IDSL6	IDSL7
		IDSL8	IDSL9	-etc.-				
	"MASTER S"	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7
		IDMA8	IDMA9	-etc.-				
BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3
		PARAM4	VAL4	PARAM5	VAL5	-etc.-		
BCONPRP	BCPPID		PARAM1	VAL1	PARAM 2	VAL2	PARAM3	VAL3
		PARAM4	VAL4	PARAM5	VAL5	-etc.-		
BCBODY1	BID (IDSLAVE Or IDMASTER)	BPID	DIM	BEHAV	BSID	BCRGID		
BSURF	BSID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7
		ELID8	ELID9	-etc.-				

Bulk Data entry - BCTABLE1

Table 17-2

BCTABL1	BCID	ID1	ID2	ID3	ID4	ID5	ID6	ID7
		ID8	-etc..					

BCTABL1 is referred by case control command BCONTACT. The IDi in BCTABL1 is the identification number of BCONNECT which defines different contact pair.

If BCTABL1 and BCONNECT have the same ID number, BCTABL1 will be chosen primarily. If BCTABL1 does not exist, the Case Control Command BCONTACT may directly refer to BCONNECT. If BCONTACT refers to a BCTABL1 ID or BCONNECT ID but no respective BCTABL1 or BCONNECT input is given, MSC



Nastran will issue a Fatal Error Message. If BCONTACT=ALLBODY, any BCTABL1 with ID>0 will be ignored. If none of BCONTACT, BCTABL1, and BCONECT exists, no contact will be defined.

When BCTABL1 with ID=0 is defined, it will be implemented no matter if BCONTACT=0 exist or not. With BCTABL1=0, it may be used for initial contact (move rigid contact bodies to just touch flexible bodies) or initial stress free (adjust coordinates to remove pre-stressed condition).

Bulk Data entry – BCONECT

Table 17-3

BCONECT	ID	BCGPID	BCPPID	IDSCND	IDPRIM			
	“SECNDRY”	IDSCN1	IDSCN2	IDSCN3	IDSCN4	IDSCN5	IDSCN6	IDSCN7
		IDSCN8	IDSCN9	-etc.-				
	“PRIMARY”	IDPRM1	IDPRM2	IDPRM3	IDPRM4	IDPRM5	IDPRM6	IDPRM7
		IDPRM8	IDPRM9	-etc.-				

Field	Contents
ID	Unique identification number referenced by a BCTABL1 entry (Integer ≥ 0).
BCGPID	Parameter identification number of a BCONPRG entry (Integer > 0 or blank).
BCPPID	Parameter identification number of a BCONPRP entry (Integer > 0 or blank).
IDSCND	Identification number of BCBODY1 entry defining the touching body (Integer > 0 or blank).
IDPRIM	Identification number of BCBODY1 entry defining the touched body (Integer > 0 or blank).
“SECNDRY”	Indicates the start of the list of the touching bodies.
IDSCNi	Identification number of BCBODY1 entry defining the touching bodies (Integer > 0).
“PRIMARY”	Indicates the start of the list of bodies touched by touching bodies.
IDPRMi	Identification number of BCBODY1 entry defining touched bodies (Integer > 0).

BCGPID and BCPPID refer to the BCONPRG entry and BCONPRP entry respectively. But, if neither BCONPRG nor BCONPRP is defined, all default values will be set for touching body.

IDSCND and IDPRIM are the short inputs to define a single touching and touched body. Once these two entries are given, “SECNDRY”, “IDSCNi”, “PRIMARY” and “IDPRMi” will be ignored. On the other hand, if these two entries are blank, “SECNDRY”, “IDSCNi”, “PRIMARY” and “IDPRMi” must be defined.

Bulk Data entry - BCONPRG

BCONPRG

Geometric Contact Parameters of Touching Bodies



Defines geometric contact parameters used to determine if contact occurs between bodies. For segment-to-segment contact it also defines additional numerical parameters used to apply the constraints. The parameters defined here are referenced by the BCONECT entry. This entry is used in conjunction with the BCONPRP entry.

Geometric Contact Parameters of Touching Bodies in SOLs 101 and 400 for General Contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the Permanently Glued or Tied Contact.

See [BCONPRG](#) in the *MSC Nastran Quick Reference Guide* for the complete bulk data entry.

Bulk Data entry - BCONPRP

BCONPRP

Physical Contact Parameters of Touching Bodies in SOLs 101 and 400

Defines physical contact parameters of touching bodies used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCONECT entry. This entry is used in conjunction with the BCONPRG entry.

See [BCONPRP](#) in the *MSC Nastran Quick Reference Guide* for the complete bulk data entry.

Bulk Data entry - BCBODY1

Table 17-4

BCBODY1	BID (IDSCND Or IDPRIM)	BPID	DIM	BEHAV	BSID	BCRGID			
---------	---------------------------------	------	-----	-------	------	--------	--	--	--

Field	Contents
BID	Unique contact body identification number referenced by BCONECT, BCHANGE, or BCMOVE
BPID	Parameter identification number of a BCBDRP entry. (Integer > 0 or blank)
DIM	Dimension of body. (Character; Default= 3D)
BEHAV	Behavior of curve or surface (Character; Default = DEFORM)
BSID	Identification number of a BSURF or BCPROP entry
BCRGID	Identification number of a BCRIGID entry

The default setting of DIM is 3D which means any 3D body composed of rigid surfaces, shell elements or solid elements.

Another setting of DIM is 2D including planar body composed of 2D elements or curves in x-y plane of basic coordinate system.



Bulk Data entry - BSURF

Table 17-5

BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	
	ELID8	ELID9	etc.						

Field	Contents
BID	Identification of a deformable surface corresponding to a BSID value on BCBODY
ELID _i	Element identification numbers.

User may also use "THRU" and "BY" to define the elements that are on contact surface as the format shown in [Table 17-6](#).

Table 17-6

BSURF	ID	ELID1	THRU	ELID2	BY	INC			
	ELID3	THRU	ELID4	BY	INC2				

Bulk Data entry - BCPARA

Defines contact parameters used in SOL 101 and SOL 400.



Table 17-7

BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3	
	Param4	Value4	Param5	Value5	etc.			

Field	Contents
ID	Subcase to which the defined parameters belong. If ID is zero, the parameters belong to all subcases. (Integer > 0; no Default)
Param(i)	Name of a parameter. Allowable names are given in Table 17-8 . (Character)
Value(i)	Value of the parameter. See Table 17-8 . (Real or Integer)

Table 17-8

ERROR	Distance below which a node is considered touching a body. Automatically calculated if left blank. (Real; Default = blank)	
BIAS	Contact tolerance bias factor. (Real value between 0.0 and 1.0.; Default = 0.9 for IGLUE=0, if field left blank or 0.0 (to obtain a near zero value, enter 1.0E-16). Default = 0.0 for IGLUE <>0, if field left blank or 0.0. Default = 0.0 for BEHAVE=SYMM on BCBODY, if field left blank or 0.0.)	
NLGLUE (SOLs 101 and 400 only)	If any secondary's for the BCTABLE corresponding to the first subcase (and first step) contain IGLUE=1, permanent glued contact will be used for all SECNDRY entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified. For further discussions of NLGLUE (Integer; Default = 0)	
FNTOL	Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank)	
THKOFF	Ignore thickness from the tolerance used by ISEARCH=2 in node-to-surface contact or from the characteristic length (for PENALT and AUGDIST) in segment-to-segment contact. (Integer 0 = do not ignore thickness or 1 = remove thickness; Default = 0)	
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real > 0.0)	
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real > 0)	
IBSEP	Flag for separation based on stresses or forces. (Integer > 0; Default = 0)	
	0	Separation based on forces.
METHOD	Flag to select Contact methods. (Character)	
	NODESURF	Regular 3D Contact (Default: node to surface contact)
	SEGSMALL	Segment to segment contact with small sliding.
	SEGLARGE	Segment to segment contact with finite sliding.
AUGMENT	Augmentation method used in a segment-to-segment contact analysis. (Integer)	
	0	No augmentation (Default)
	1	Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements
	2	Augmentation based on a constant Lagrange multiplier field
	3	Augmentation based on a (bi)linear Lagrange multiplier field



IBSEP	Flag for separation based on stresses or forces. (Integer ≥ 0 ; Default = 0)	
	0	Separation based on forces.
	1	Separation based on absolute stresses (force/area)
	2	Separation based on absolute stress (extrapolating integration point stresses)
	3	Relative nodal stress (force/area)
	4	Separation based on relative stress (extrapolating integration point stresses)
Only option 2 and 4 can be used with mid-side node elements where the mid-side nodes contact (LINQUAD=-1). For segment to segment contact, the program will set IBSEP to 2 internally.		
FTYPE	Friction type. See Remark 5. (Integer)	
	0	No friction. (Default)
	6	Bilinear Coulomb friction.
	7	Bilinear Shear friction.

Prior to bodies coming into contact due to load application, it may be desirable to initially identify contacting bodies or specify initial stress-free contact. Thus, Bulk Data entries such as BCPARA have an ID field which may be 0 or blank. (For the BCPARA entry, BCPARA,0 is usually the typical and only usage, as it is used to define initial contact parameters.) For other entries such as a BCTABL1 entry, a BCTABL1,0 entry would be used to identify the contact bodies that can possibly contact each other at the start of the analysis. Then BCTABL1, n with $n > 0$ and selected in Case Control by a BCONTACT - n entry, is used to control contact during loading.

BCPARA defines global parameters. It means these entries might be overwritten by BCONPRG and BCONPRP. For example, FNTOL might be overwritten by the same entries inputs in BCONPRP.

To turn on segment-to-segment contact, METHOD need to be used.

When quadratic element is detected and user does not define IBSEP, the default IBSEP=0 will be set to 2 to guarantee successful analysis.

Guidelines and Limitations

1. Surface-to-surface 3-D contact is currently supported. For solids contacting solids, the contact surfaces contain the grid points. For shell surfaces contacting each other, the contacting surfaces do not contain the grid points. Taking the shell thickness into consideration, for contacting shell surfaces, the distance between grid points is, $(t_1 + t_2)/2$ where t_1 and t_2 are the thickness of the two contacting shells. For deformable-rigid contact, contact occurs when the position of the shell grid plus or minus half the thickness projected with the normal comes into contact with the rigid segment. Note that, if you specify that the shell grid points are touching surface, the algorithm will assume contact penetration.



2. Contact in SOL 101 means standard linear small strain, small rotation and full nonlinear contact algorithm without material nonlinearity. Linear Contact (LINCNT) is preferred method in SOL 101 to achieve consistent linear solution, in which contact relation is built upon initial configuration, but not updated configuration through iterations. It is recommended to set SYSTEM(786)=1 to turn on Linear Contact as default in SOL 101. Refer to Nonlinear User's Guide for detailed information of Linear Contact (LINCNT).
3. Permanent Glued Contact is defined when the IGLUE field of the BCTABLE or BCONPRG is set > 0 or < 0 . In this option, all degrees of freedom of the contact grid points are constrained with MPCs or contact constraint matrix in the case of deformable-deformable contact once the grid points have come in contact. The relative tangential motion of a contact grid is zero in the case of deformable-rigid contact. If IGLUE > 0 on the BCTABLE or BCONPRG, MSC Nastran will form the required constraints without entering the full nonlinear contact algorithm. If IGLUE < 0 , MSC Nastran will form constraint matrix based on initial contact, but it will enter the full nonlinear contact algorithm in order to handle large displacement or large rotations. If there is no initial contact between the contact bodies, these constraints cannot be formed and the run will fail. Therefore special cases arise, described as follows:
 - a. To run Permanent Glue contact, with no initial contact between the contact bodies, user may define entry BCPARA,0,NLGLUE,1 to turn on the general nonlinear contact algorithm of nonlinear analysis (SOL 400). Use Case Control BCONTACT etc. to define possible contacting surfaces. The algorithm will then determine the contact surfaces and "glue" the bodies together.
 - b. Or optionally on the BCONPRG entry, specify a value for the ERROR field. Any grid point within this error tolerance will be considered to be in contact. If the user sets ICOORD=1 on the BCONPRG entry and this BCONPRG is referred by BCTABL1,0 then initial stress free contact can be turned on and the grid points will be physically moved so that the surfaces are actually in contact. You may also use the Bulk Data entry BCPARA,0 field THKOFF to have the algorithm to ignore shell thickness when computing contact penetration.
4. Each grid and element should be, at most, in one body. Solids and shells should not be mixed in the same body.
5. Deformable bodies can contact rigid bodies. Contact between rigid bodies is not supported.
6. In the following text, examples of contact bulk data entries and their fields are given. Each "Entries" and "Fields" were defined once. However, the parameter may appear on multiple "Entries." "Field" values on the BCPARA entry may be overridden by "Field" values on the BCONPRG and BCONPRP.
7. When defining deformable-deformable contact, the default rule is that the grid points of a body may contact any other body and the searching starts with the body having the lowest ID number and ends with the body having the highest ID number. Depending on the contact constraints found, it is possible that grid points of a body with a higher ID number contact surfaces of bodies with a lower ID number. This default behavior (so called double sided contact) can be overruled by the user in different ways, namely via the BCBDRP option or the BCONPRG option.



Via the BCBDPRP - ISTYP option, two different global search orders can be defined. The first is so called single sided contact, which means that the search for contact for grid points of a deformable body is only done with respect to surfaces of the body itself and of bodies with a higher ID number. The other activates a process which tries to optimize the set of contacting grid points and contacted surfaces and is only recommended for rare cases where a proper set of "PRIMARY" and "SECNDRY" bodies cannot be defined by the user.

The recommended way of overruling the default search process is via the BCONPRG - ISEARCH option, where a local search method can be defined for each body combination. The BCONECT option allows the user to define which pair of contact bodies can potentially come into contact and, if they can come into contact, it is also possible to specify that grid points of the body indicated as "SECNDRY" can only contact surfaces of the body indicated as "PRIMARY", and not the other way around. In this way one can define single sided contact per body combination, thus overruling the default double sided contact.

8. When defining deformable-deformable contact using the node-to-segment method, it is important to ensure that grid points of the body with the finer mesh contacting surfaces of a body with a coarser mesh. When defining deformable-rigid contact, it is important that the deformable bodies all have ID numbers lower than the rigid bodies.
9. When using the segment-to-segment method softer bodies should be contacting stiffer bodies. E.g., rubber should be a SECNDRY and steel should be a contacted body or PRIMARY.
10. Avoid sharp corners in contacted surfaces. It is better to have a finer mesh and smooth surface where possible.
11. In sliding contact try to have smooth surfaces. Use the SPLINE option, which generates a Coons surface (where the mesh surfaces are "patched"). This helps considerably by avoiding scattering and accelerating convergence.
12. NLPARM (Case Control) and NLSTEP (Case Control) are both supported in SOL101. For SOL101 general contact analysis, since the only nonlinearity comes from contact, some fields are not user-controllable in NLPARM. These fields include DT(=0.0), KMETHOD(AUTO), KSTEP(=500), INTOUT(=NO) and MAXLS(=0). NLPARM is not required for running SOL101 contact jobs. If users want to use NLSTEP, the keyword "LCNT" needs to claim for the Contact in SOL101. Users may refer to the *MSC Nastran Quick Reference Guide* for more explanations about the parameters which are used in NLSTEP for Contact in SOL101.
13. The motion of deformable bodies is prescribed by applying displacements or loads to the bodies. It is NOT recommended that enforced displacements or point loads are placed on grid points that might come into contact with a rigid body.
14. It is recommended that MPC relationships should be avoided for grid points that might come into contact.
15. Rigid bodies may have prescribed velocity, position, or load. See BCRIGID entry.
16. The tolerance value lies equally above and below the contacted surface. That is to say, the "thickness" of the contacted surface is twice the value of the Tolerance. Often, the grid points are almost touching the contacted surface. The algorithm provides for a biased tolerance with a smaller distance on the outside (the side nearest the contacting surface) and a larger distance on the inside. This avoids the



close grid points from coming into contact and separating again. There is a default bias of 0 in the SOL 400 and SOL 101 algorithm. The user may set the bias. See BCONPRG entry field BIAS. The outside contact area is computed as $(1-\text{BIAS})*\text{ERROR}$ and the inside contact area is $(1+\text{BIAS})*\text{ERROR}$.

17. In SOL 101 and SOL 400, bilinear Coulomb (see BCPARA entry, FTYPE = 6) or bilinear shear (see BCPARA entry, FTYPE = 7) friction is available. Bilinear shear is usually used for forging processes. Bilinear Coulomb is used for most friction applications.
18. Currently, quadratic (8-noded) shell elements are not recommended for use with contact.

Example 1: Permanent Glued Contact

As an example of Permanent Glued Contact, [Figure 17-3](#) shows a vertical shell plate lying on a solid beam. They are to be glued together, then have a FORCE applied to the upper edge of the shell sufficient to bend both parts. Note on [Figure 17-3](#) the dissimilar meshes and the force direction. Part of the input file (see [MSC_DOC_DIR/doc/linstat/lnrcmc02.dat](#)) is shown as [Listing 17-5](#). Permanent glued contact is selected when the IGLUE field in the BCONPRG entry is $\neq 0$.

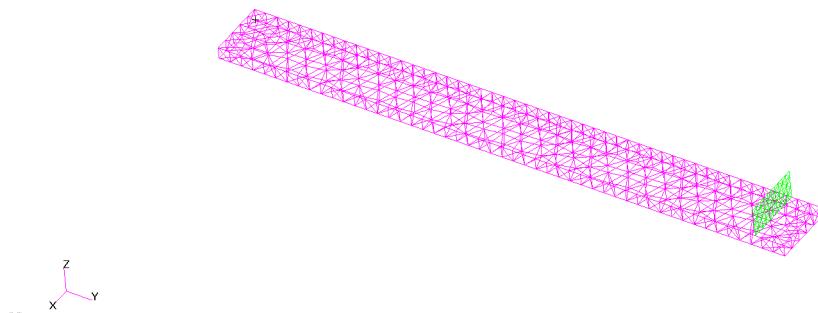


Figure 17-3 Permanent Glued Contact in SOL 101

Listing 17-5 Part of lnrcmc02.dat

```
$
$ filename = lnrcmc02.dat
$
SOL 101
CEND
TITLE =Linear contact with moment-carrying permanent glue
ECHO = NONE
$ Ground check on N-set after applying GLUING
groundcheck(set=N)=yes
$
SUBCASE 1
$ Subcase name : General contact
SUBTITLE=T3 shells glued to TET10 solids
LABEL    =Default NLPARM
```



```

BCONTACT = 1
nlparm=1
SPC = 2
LOAD = 2
$  

DISPLACEMENT(plot)= ALL
STRESS(plot)= ALL
    autospc(noprint)=YES
$  

REPCASE 2
    $ Selected disp and stress sets
    set 1 = 2812,2818,2824,2830,2836,2842,2848,2854,2860,2866,2872,
            2806,1257,1258,1259,1260,1261,1262,1263,1264,1265,1375
    set 2 = 124, 127, 128, 130, 1282,1283,1292,1293,1302,1303,1312,
            1313,1322,1323,1332,1333,1342,1343,1352,1353,1362,1363
    DISPLACEMENT = 1
    STRESS      = 2
$  

BEGIN BULK
PARAM     POST      -1
PARAM     PRTMAXIM YES
$  

NLPARM   1
$-----2-----3-----4-----5-----6-----7-----8-----9-----0-----  

PV          NO
BCTABL1 1      10
BCONNECT 10     11      12      3      5
BCONPRG 11           IGLUE   3      ISEARCH 1
BCONPRP 12           FRLIM   1.+20
$ Deform Body Contact LBC set: Shell body
BCBODY1  3           3D      DEFORM  3      0
$ Deform Body Contact LBC set: Solid body
BCBODY1  5           3D      DEFORM  5      0
$
```

The setup for this problem is similar to the previous example for selecting Contact and defining the contact bodies (note the BSURF entries are not shown). The main difference is the inclusion of IGLUE=3 on the BCONPRG entry to select permanent gluing. A request for ground check, a set selection for printed output and full output for plotting has been included. The plot in Figure 17-4 shows the displacement result of the glued bodies.

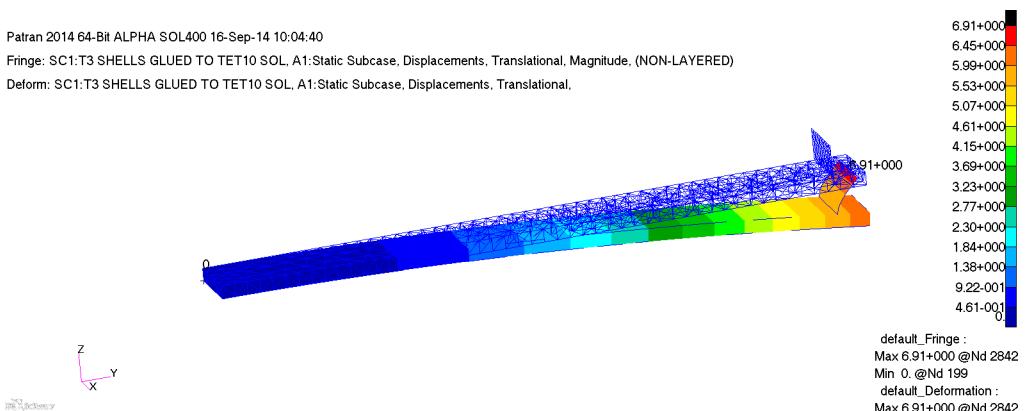


Figure 17-4 Result of the Glued Bodies

As a second example of permanent glue, the plates on the CINTC example shown in [Figure 4-63](#) will be joined with permanent glued contact. The Bulk Data entries shown in [Listing 4-12](#) can be replaced by the Bulk Data entry shown in [Listing 17-6](#). (See MSC_DOC_DIR/doc/linstat/itfpltq4glue.dat) plus the Case Control command BCONTACT=1.

Listing 17-6 Permanent Glued Bodies

```
BCONTACT = 1
$
$ definition by glued bodies
$
$ define bodies
bctabl1,1
bcconnect,1,10,,1,2
bconprg,10,,error,.01,iglue,3,icoord,3,copts1,60,coptm1,60
bcbbody,2,,3d,deform,10
bcbbody,1,,3d,deform,11
$ body properties
bcprop,10,1
bcprop,11,2
```

The BCPARA entry selects the nonlinear contact algorithm because there is no initial contact. In the BCONPRG, IGLUE=3 is selected for full moment carrying permanent glue with shell contact. Also COPTS1 and COPTM1=60, which are the setting for permanent glue contact with shell elements. The results are virtually identical to [Figure 4-65](#).



18

Adaptive Meshing

- Local Adaptive Mesh Refinement



Local Adaptive Mesh Refinement

Introduction

MSC Nastran has a Local Adaptive Mesh Refinement capability which can be used to increase accuracy when starting from “rough” initial meshes, and can be used in situations involving contact and/or superelements.

Adaptive mesh refinement is an automatic mechanism for altering and controlling locally the size of the finite element mesh. Beginning with an initial mesh provided by the user, a sequence of new meshes is automatically generated. Each new mesh of this sequence is an offspring of the previous, coarser mesh and is obtained by refining (by subdivision) a subset of their elements.

Figure 18-1 illustrates this mechanism, it is a 2D elastic analysis of a Mode-I fracture specimen.

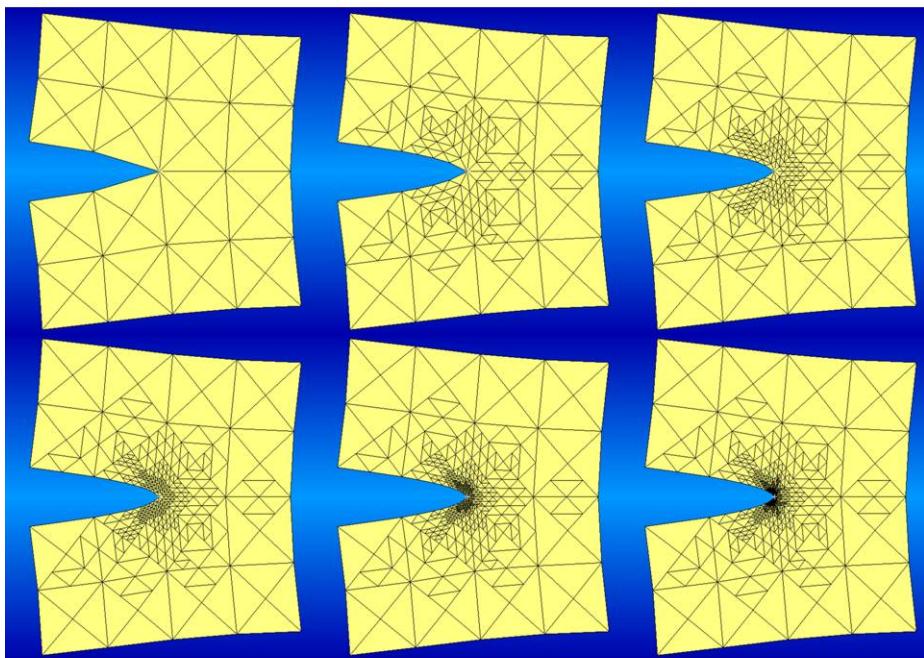


Figure 18-1 2D Elastic analysis of a mode-I fracture specimen

Adaptive mesh refinement can be applied to meshes that combine elements of different types (triangular or quadrilateral surface elements, tetrahedral, pentahedral or hexahedral volume elements), different interpolation orders (linear or quadratic), different dimensionality (line, surface or volume elements), or models substructured into different superelements.

The following elements are supported:

- Line elements: CBEAM, CBEAM3 (with no offsets or warping), CBEND, CBAR (with no offsets), CONROD, CROD, CTUBE, CVISC.
- Surface elements: CTRIA3, CTRIAR, CTRIA6, CQUAD4, CQUADR, CQUAD8



- Volume elements: CTETRA, CPENTA, CHEXA.

Adaptive mesh refinement (or *h-adaptivity*) attempts to change the element size while keeping interpolation order unaltered.

When an element is refined (subdivided) but its adjacent elements are not refined a non conforming mesh is generated. Nodes created on the boundary between a refined and a non refined element are referred to as *hanging-nodes* (Figure 18-2).

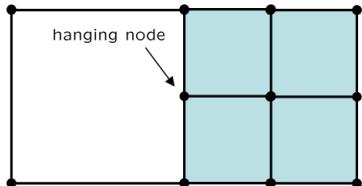


Figure 18-2 Hanging node

In MSC Nastran, all degrees of freedom (1 to 6) associated to a hanging node are automatically constrained using internal Multipoint Constraint (MPC) equations derived from the isoparametric mapping.

Local adaptive mesh refinement is activated by the Case Control command HADAPT and controlled by the two Bulk Data entries HADAPTL and HADACRI along with the optional feature angle parameter VARPHI (Figure 18-3).

- The Case Control command **HADAPT (Case)** in the *MSC Nastran Quick Reference Guide* must reference the Bulk Data entry HADAPTL.
- The Bulk Data entry **HADAPTL** in the *MSC Nastran Quick Reference Guide* provides an interface to control the number of iterations in the adaptive mesh refinement loop (REPEAT field), the refinement criteria (CRITID field), which must reference a Bulk Data entry HADACRI, the refinement region where the latter will be applied (WHEREMETHOD and WHEREID fields), the placement method for new mid-edge nodes (SNAPMETHOD field), and the maximum levels of refinements permitted to any individual element in the mesh (MAXLEVEL field).
- The Bulk Data entry **HADACRI** in the *MSC Nastran Quick Reference Guide* provides an interface for the specification of the refinement criterion along with criteria specific parameters.
- The parameter **VARPHI** in the *MSC Nastran Quick Reference Guide*, (feature angle) can be optionally adjusted when corners and edges are not satisfactory detected to control how sharp a mesh edge or vertex should be in order to be consider a split edge or vertex between two otherwise continuous curves or surfaces.
- Different mesh refinement criteria might be applied to different refinement regions by combining two pairs of Bulk Data entries HADAPTL and HADACRI. Furthermore, mesh refinement can be driven by a combined error indicator based on stresses arising from multiple load cases.



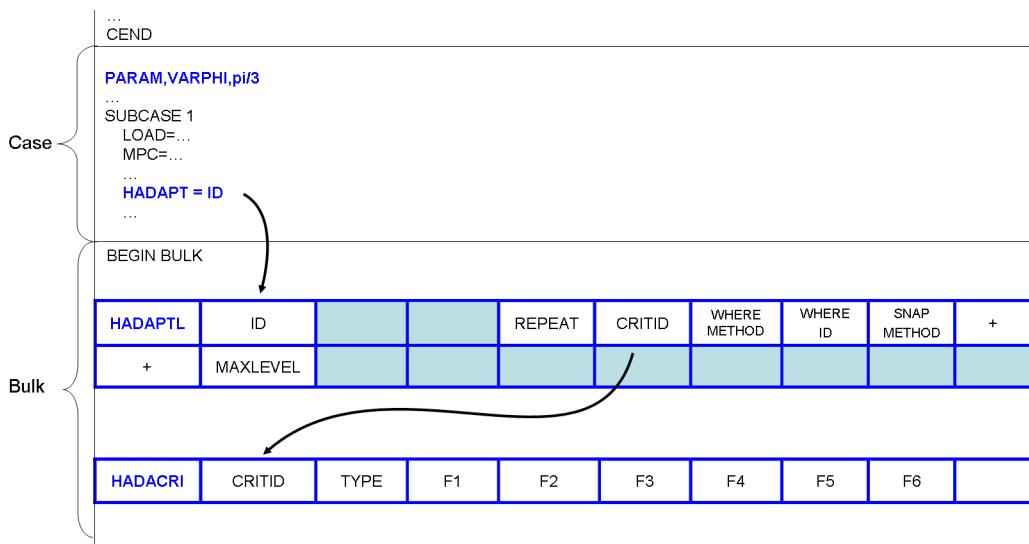


Figure 18-3 User interface to activate and control the adaptive mesh refinement capability

Selection of Refinement Region

Adaptive mesh refinement can be either requested for all elements in the mesh or for a subset of elements. Two different mesh refinement subsets are supported:

- elements sharing a given property ID
- elements belonging to a given superelement.

The refinement region can be specified by the user via the pair of fields (WHEREMET, WHEREID) in the HADAPTL Bulk Data entry as follows:

- If mesh refinement must be restricted to all elements sharing a given property identified with property ID “PID”, then the WHEREMET field must be set to the keyword “PROP” and WHEREID field must be set to the integer PID:

1	2	3	4	5	6	7	8	9	10
HADAPTL	1				101	PROP	PID		

- If mesh refinement must be restricted to a particular superelement identified with superelement ID “SEID”, then the WHEREMET field must be set to the keyword “SUPER” and the WHEREID field must be set to the integer SEID:

1	2	3	4	5	6	7	8	9	10
HADAPTL	1				101	SUPER	SEID		



- Finally, if mesh refinement is requested for all elements in the mesh, then the field WHEREMET must be set to the keyword “ALL” and the WHEREID field is ignored.

1	2	3	4	5	6	7	8	9	10
HADAPTL	1				101	ALL			

Consider by way of example a cylindrical shell subjected to a concentrated force as depicted in (Figure 18-4). Two different properties (labeled with IDs 1 and 2) have been assigned to the top and bottom halves of the shell. The concentrated force is applied on the center node of the shell (located at the interface between both regions) and in the direction normal to the shell.

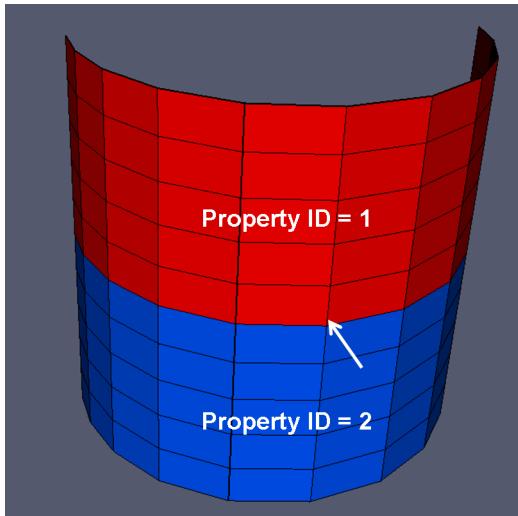


Figure 18-4 Pinched cylindrical shell. Different properties have been assigned to elements in the top and bottom halves of the shell.

Mesh refinement using the error indicator based criterion have been requested for the bottom half (property 2):

1	2	3	4	5	6	7	8	9	10
HADAPTL	1				101	PROP	2		
HADACRI	101	1	0.9						

Figure 18-5 shows the sequence of meshes and deformed configuration obtained during the adaptive mesh refinement process. Notice that even though the refinement is mainly confined to the bottom half, it also propagates a few layers into the top half due to the 2-to-1 rule.



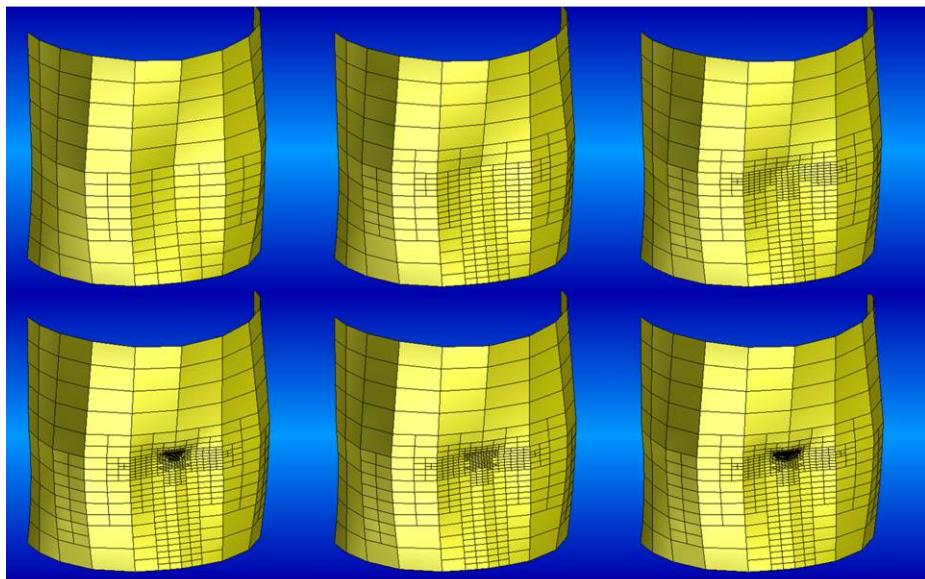


Figure 18-5 Pinched cylindrical shell. Sequence of meshes and deformed configuration obtained during the mesh refinement process.

Selection of Refinement Criterion

The refinement criterion that will be applied to the refinement region is selected by specifying a refinement criteria ID on the CRITID field in the HADAPTL Bulk Data entry:

1	2	3	4	5	6	7	8	9	10
HADAPTL	1				CRITID	ALL			

along with a corresponding HADACRI Bulk Data entry,

1	2	3	4	5	6	7	8	9	10
HADACRI	CRITID	TYPE	F1	F2	F3	F4	F5	F6	

The particular refinement criteria is specified using the TYPE field in the HADACRI Bulk Data entry. Four different refinement criteria, can be selected, namely:

TYPE	Name of Mesh Refinement Criterion
1	Error indicator based criterion
2	Element within a spatial spherical region criterion
3	Elements within a spatial cubic region criterion
4	Elements in contact criterion.



See Remark 2 under [HADACRI](#), CRITID, TYPE in the *MSC Nastran Quick Reference Guide* for a description of fields F1 to F6.

Combination of Subcases with Error Indicator Based Criterion

The error indicator is computed using the finite element stresses and measures indirectly the stress discontinuity across interelement boundaries. When multiple load cases are defined, multiple finite element stress solutions are obtained (one for each load case) and therefore, multiple instances of the error indicator are computed.

Every load case that should be considered for the computation of the error indicator must include an HADAPT Case Control command referencing a unique HADAPTL Bulk Data entry:

Error Indicator Based on the Combination of Load Case 1 and Load Case 2 ([Figure 18-6](#))

SUBCASE 1

...
HADAPT = 1

SUBCASE 2

...
HADAPT = 1

BEGIN BULK								
...								
HADAPTL	1			111	ALL			
HADAPTL	1			222	ALL			
ENDDATA								

One single pair of Bulk Data entries HADPTL and HADACRI are required. The unique HADAPTL entry must be referenced by an HADAPT Case Control command included in both load cases ([Figure 18-6](#)).

[Figure 18-6](#) shows the sequence of meshes obtained during the adaptive mesh refinement process on the cylindrical shell subjected to two independent load cases.



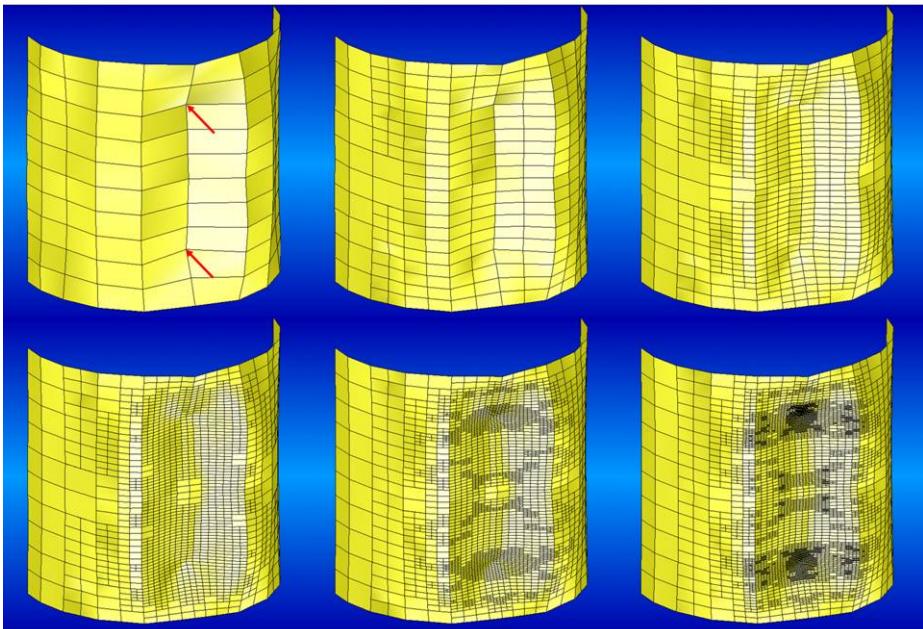


Figure 18-6 Sequence of meshes and deformed configuration obtained using the error indicator based criterion applied to both the first and second load cases

User Information Messages (.f06 File)

The output requests for displacements, stresses, forces, etc., are honored for all the iterations of the adaptive mesh refinement loop ([Figure 18-7](#)). Thus, for example, if DISPLACEMENT=ALL is specified in the Case Control section of the input file, then the grid point displacements will be written to the .f06 file not only for the initial mesh, but for all the subsequent meshes created during the mesh refinement process.

At the end of each refinement cycle in the adaptive mesh refinement loop ([Figure 18-7](#)) the following user information message is printed to the .f06 file to signal the end of the analysis supported on the current mesh and beginning of a new analysis supported on the refined mesh obtained from the previous:

```
-----  
* * * E N D O F A N A L Y S I S #: 2 * * *  
-----
```

The total number of elements meeting the user's specified criterion and the total number of elements actually refined is reported to the .f06 at the end of each successful refinement instance (step 3 and 4 in [Figure 18-7](#)) and prior to the transference of analysis data between unrefined and refined meshes (step 5 in [Figure 18-7](#)).



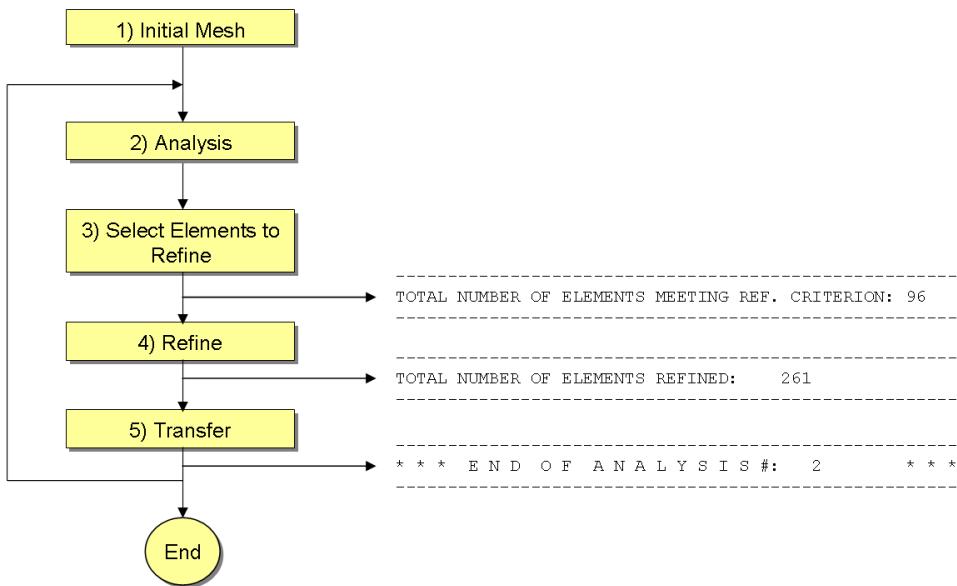


Figure 18-7 User information messages reporting the progress of the adaptive mesh refinement loop

Notice that the number of elements actually refined will be in general different from the number of elements meeting the refinement criterion because the refinement is propagated from the latter to the neighbors.

When the error indicator based criterion is selected, a user message is printed to the .f06 file informing the total number of elements scanned for the computation of the error indicator, the mean square average over the whole mesh of the local error indicator and the relative change of this magnitude with respect to the previous iteration in the adaptive mesh refinement loop:

```

-----
GLOBAL NUMBER OF ELEMENTS:          64
AVERAGE ERROR INDICATOR:          3.522044E+04
CHANGE IN AVERAGE ERROR INDICATOR: 1.389418E+01 %
-----

```

It bears emphasis that the error *indicator* is not an error *estimator* in the sense that its numerical value does not measure the actual (absolute) error but gives rather a relative assessment of where the mesh should be refined.

If no elements meet the user specified criterion or no elements are actually refined, the adaptive mesh refinement loop is terminated and corresponding messages are printed to the F06 file.

Output Files for Postprocessing in MSC Patran or SimX

A request for the creation of post processing files (“.xdb” using PARAM,POST,0 or “.op2” using PARAM,POST,-2) is automatically honored for each and all analysis instances in the adaptive mesh



refinement loop. Different postprocessing files are automatically created in the same directory and with the same name as the input file and with the extension “.xdb” or “.op2” where i is the iteration counter in the adaptive mesh refinement loop. Thus, for example, if the input file is `fender.dat`, then, the sequence of files will be:

```
fender.xdb or fender.op2
fender.1.xdb or fender.1.op2
...
fender.i.xdb or fender.i.op2
```

The first file in the sequence will contain postprocessing data corresponding to the initial mesh and analysis results and subsequent files will contain postprocessing information for each refined mesh created during the adaptive mesh refinement process.

Bulk Data File Images of the Sequence of Refined Meshes

During the adaptive mesh refinement process, new elements, new grid points, new boundary conditions, new multipoint constraints and new pressure loads are created automatically. Furthermore, contact bodies are internally redefined to subtract refined elements and replace them by their children elements.

Bulk data file images containing the new mesh and analysis data created after each refinement cycle is automatically generated prior to the beginning of each analysis cycle.

Each bulk data file image is created in the same directory and with the same name as the input file and with the extension “.seid.i.bdf” where `seid` is the superelement ID (0 for models with no superelements) and i is the iteration counter in the adaptive mesh refinement loop. Thus, for example, if the input file is `fender.dat` and contains no upstream superelements (only the residual structure, i.e. SEID=0), then the sequence of bulk data files created automatically will be named:

```
fender.0.1.bdf
...
fender.0.i.bdf
...
```

These bulk data file images might be used as the starting point for the creation of a new analysis input files supported on any of the refined mesh obtained during the adaptive mesh refinement cycle.

Modeling Guidelines

- The number of elements created during refinement grows exponentially. If on each refinement iteration i , a fraction f_i of the total number of elements N_i is refined by subdividing each element of this fraction into M children elements (with $M = 2$ for line elements, $M = 4$ for surface elements and $M = 8$ for volume elements), then, the total number of element at iteration $i + 1$ will be:

$$N_{i+1} = N_i - f_i N_i + f_i N_i \cdot M = N_i(1 + f_i(M - 1))$$



Therefore, if the fraction f_i remains approximately constant during the mesh refinement process, i.e., if $f_i \approx f$ with f a constant independent of i , then the number of elements at iteration i will be given by the estimate

$$N_i = N_0(1 + f(M - 1))^i$$

where N_0 is the number of elements in the initial mesh (the mesh provided by the user). For example, in a surface mesh ($M = 4$) and if approximately 1/3 of the elements are refined on each refinement iteration ($f = 1/3$), then the number of elements at iteration i will be roughly

$N_i = N_0 2^i$. Thus an initial structure with thousands of shell elements ($N_0 = O(10^3)$) will be refined into millions of shell elements ($N_0 = O(10^6)$) in about 10 iterations. In a 3D mesh ($M = 8$), and if approximately 1/3 of the elements are refined on each iteration, the total number of elements expected at iteration i will be approximately $N_i = N_0 (10/3)^i$. Thus, a mesh with thousands of 3D elements will be refined into millions of 3D elements in about 6 iterations.

- Exponential growth of the number of elements implies that adaptive mesh refinement is memory intensive. As a rough estimate, each refinement iteration i requires on the order of $100 \cdot N_i$ integer words of memory where N_i is the number of elements of the mesh created during refinement iteration i .
- Instead of the traditional modeling practice, the user should start the process with an initial mesh preferably coarse which will be refined automatically and selectively according to the refinement criterion.
- The effectiveness of the refinement process depends on an appropriate detection of geometric corners, creases and edges and interfaces between elements of different properties. Detection of geometric features requires the selection of a proper value for the Geometric Feature parameter (PARAM,VARPHI).
- When the initial mesh is very coarse and the boundary of the structure under analysis is poorly approximated, it is recommended to activate the automatic projection of mid-edge nodes onto a smooth approximation of the mesh boundary using SNAPMETH=1. Convergence of the mesh refinement process might be dramatically improved using this method.
- The user should avoid the use of MPC sets 90000000 to 99999999 which are reserved for hanging nodes constraints generated during the adaptive mesh refinement process.
- When mesh refinement is restricted to a specific mesh refinement region (by selecting WHEREMET=PROP or WHEREMET=SUPER in the HADAPTL Bulk Data entry), the user should expect refinement also in a few layers away from the refinement region due to the enforced implicit propagation ruleMs.



- In partitioned superelements, the HADAPT entry must be specified in the main bulk data section. Entries specified in the Bulk Data Section corresponding to individual parts (sections beginning with BEGIN SUPER) will be ignored.
- When using regular superelements, the Bulk Data Section must begin with BEGIN SUPER as opposed to BEGIN BULK in order to the refinement to be appropriately propagated across superelement boundaries. If BEGIN BULK is used, grid points on the superelement boundaries will be duplicated and not shared by the joining superelements.

Limitations

- The error indicator based refinement criterion can be used with surface or volume elements, but not with line elements. The latter can be subdivided with any other of the refinement criteria, or when they are attached to the boundary of a surface or volume elements
- Refinement of CBEAM, CBEAM3 with offsets or warping are not supported. Refinement of CBAR with offsets is not supported.
- Temperature loads are not supported. Similarly, the HEATSTAT=YES option in SOL 101 that runs a preliminary thermal analysis to compute thermal loads for a subsequent structural analysis is not supported.



19

Nastran Smart Superelement

- Introduction to Smart Superelement
- Benefits
- Workflow
- The SSE Process and the User Interface
- Linear Statics SSE Consumption in a MSC Nastran Assembly Run
- References



Introduction to Smart Superelement

MSC Nastran smart superelement (SSE) capability employs ODYSSEE machine learning technology for reduced order modeling (ROM) to create parameterized superelements [1]. The SSE is parameterized with model (geometric, material, etc.) parameters so that the user can readily change the superelement's properties in each simulation to explore the design space without having to completely reprocess the superelement (SE).

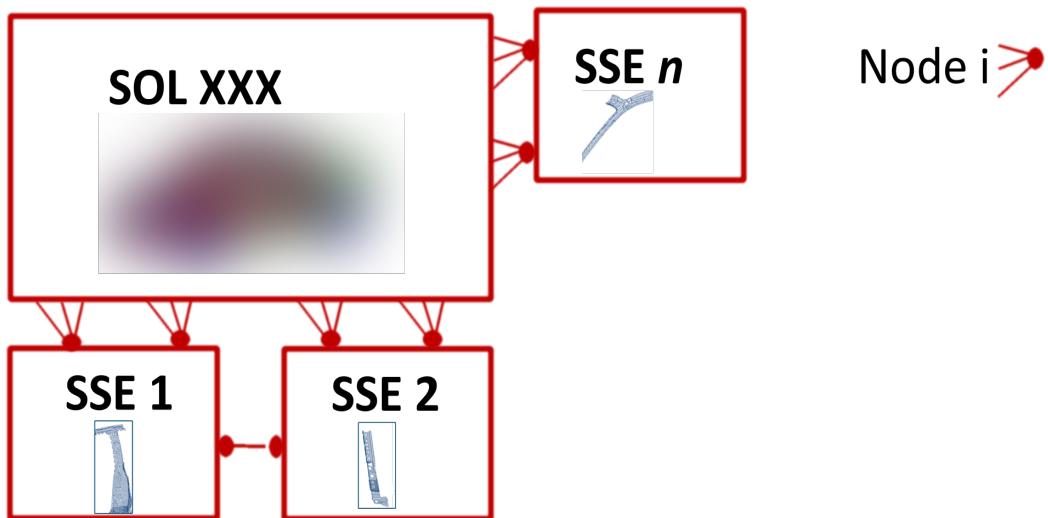


Figure 19-1 Schematic Diagram of MSC Nastran Model using multiple SSEs

The SSE capability is supported in the following MSC Nastran solution sequences:

- Sol 101: Linear Static analysis.
- Sol 103, 107, 110: Normal modes and complex modes.
- Sol 108, 109: Linear direct frequency and transient response.
- Sol 111, 112: Linear modal frequency and transient response.
- Sol 400: Nonlinear static and transient analysis

Here, we discuss the SSE capability for the linear statics. For linear dynamics, please see [2].

Benefits

Using the SSE capability the user will be able to explore the design space by conducting MSC Nastran simulation with different SSE parameters. Additionally, an analysis using SSE is typically much faster than a traditional MSC Nastran assembly model as there is no need to reprocess the entire superelement each time.



Workflow

The basic SSE process is a 2-Step procedure:

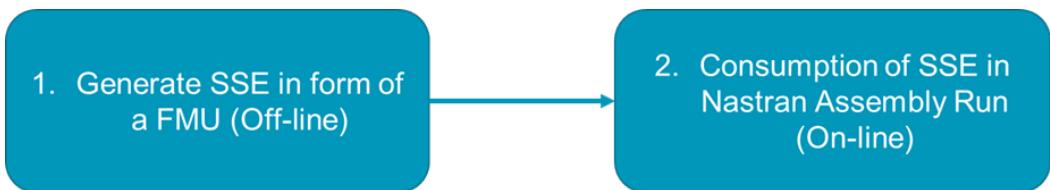


Figure 19-2 2-Step Generate/Consume SSE Procedure

1. Generate Linear Statics SSE (done outside MSC Nastran, i.e. off-line):

- A MSC Nastran base external superelement generation (EXTSEOOUT) finite element model of the SSE component is developed. The input file is the same as a traditional EXTSEOOUT input file with the additional option to output the reduced matrices, loads, displacement OTM required to generate the SSE to a HDF5 file.
- User identifies SSE parameters in ODYSSEE CAE – Lunar^[3] using the DOE generator tool.
- In the Lunar DOE generator, the user creates a design-of-experiments (DOEs) table to train the SSE (the choice of DOEs is model specific and requires domain knowledge). The SSE is trained to predict the output matrices for a given set of input parameters. A batch simulation of DOEs is run: each instance/experiment of this step requires an updated valid MSC Nastran input file (automatically created by Lunar). The output of each experiment is automatically extracted (Lunar supports extracting output data from MSC Nastran HDF5 format) to create a dataset (X, Y) for training and validating the SSE:

X: User identified, input parameters of SSE.

Y: Output to be supplied by SSE. For the linear case this clearly is,

$[K]_{axa}$: reduced stiffness matrix (a-size)

$[F]_{axnl}$: reduced load vector (a row size \times nl), where nl is the number of loads (referred to as PA in HDF5 file)

$[DISP-OTM]_{a \times a}$: the two displacement output transformation matrices (OTMs),

a. $[MUG1]_{*xa}$: The free boundary displacement OTM, (* = number of user requested output grids \times a-row size)

b. $[MUG10]_{*xa}$: The free boundary displacement OTM, (* = number of user requested output grids \times nl)

- Creating the SSE Functional Mock-up Unit (FMU) in Lunar by training it with dataset (X,Y): After training, a good SSE model should be able to predict Y for a yet unseen value of X with acceptable accuracy. In essence, the linear SSE is a parametrized SE which can change its representation instantly by changing the input parameters X.



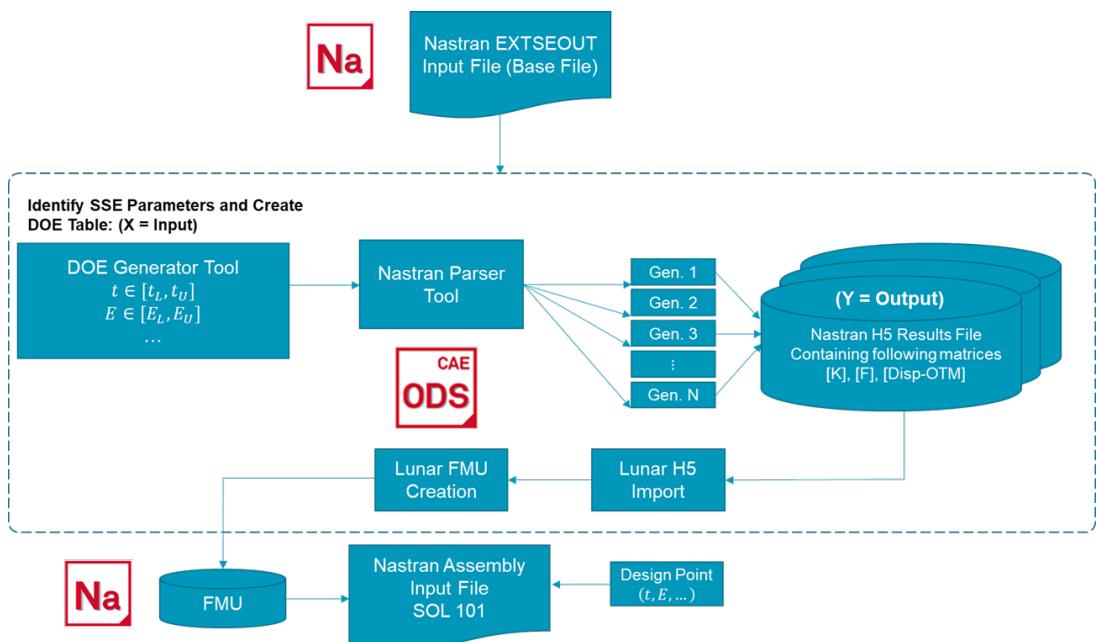


Figure 19-3 Linear Statics SSE Workflow



Figure 19-4 ODYSSEE Linear Statics SSE FMU

2. Consumption of SSE (done in MSC Nastran, On-line):

The SSE will be consumed in MSC Nastran with a process that is similar to the inclusion of a Part SE in a MSC Nastran Analysis except that a SSE is indicated by the presence of a SEBULK entry with SETYPE=EXTROM. The SSE parameters are specified inside the SE's bulk data through the presence of pairs of DESVAR/DVMREL1 or DESVAR/DVPREL1 bulk data entries.



The SSE Process and the User Interface

EXTSEOUT Input File

After the generation of baseline modes, the user prepares the EXTSEOUT analysis file which will be used by Lunar to generate the SSE. This file is just a traditional EXTSEOUT analysis file for the SSE component which outputs the external superelement data to a HDF5 (instead of OP2/OP4/etc.) file supported by Lunar as follows:

```
SOL 101
CEND
$ DISPLACEMENT output request is required to generate displacement OTMs
$ which will be employed in interior data recovery for the SSE
$ in the assembly run
SET 200 = 34,566, 445,...
DISP=200
SPC=10
LOAD=20
...
EXTSEOUT (ASMBULK=MAN ,EXTBULK ,EXTID=20 ,HDF5)
BEGIN BULK
$ Attachment points (A-set) selection:
ASET1,123456,70801, THRU, 70850
...
$ Element Property : Thickness, Tagged as SSE Parameter in Lunar
PSHELL 1 1 .56 1
$ Referenced Material Records: E, Tagged as SSE Parameter in Lunar
MAT1 1 10.E6 .3 2.588-4 1.E-6 0..05
$ Load Vector
FORCE,20,111,,100.,1.,1.,1.
...
$ Displacement Constraints
SPC1 10 123456 101 201
...
ENDDATA
```

The EXTSEOUT case control with HDF5 option will store the output matrices on a HDF5 (.h5) file which is required as Lunar currently only supports the import of HDF5 file from MSC Nastran, e.g.,

EXTSEOUT (ASMBULK=MAN ,EXTBULK ,EXTID=20 ,HDF5)

After running the EXTSEOUT analysis, MSC Nastran will produce two files (.asm and .pch) that define the necessary connection information for consuming the SSE in Assembly run. These files assume that the connection nodes in the assembly run will have the same IDs as the grids in the reduction run. If this is not the case, the user can modify the SEBULK entry to use the AUTO method, or manually provide the pairs.

The EXTSEOUT input file is then imported into Lunar to generate the SSE FMU using the workflow shown in [Figure 19-3](#) (Please see [2] for further details)



Linear Statics SSE Consumption in a MSC Nastran Assembly Run

The process of consuming a SSE in MSC Nastran SOL 101 assembly run is similar to linear dynamics (Please see [2] for further details).

Guidelines and Limitation

The following guidelines and limitation are associated with the capability:

1. In the EXTSEOUT (SSE generation) analysis the displacement output request is required to generate displacement OTMs which will be employed in interior data recovery for the SSE in the assembly run. If the size of these OTMs are very large then both the training time and time required to conduct interior data recovery for the SSE may be long. It is therefore recommended that the user only request the displacement at grids of interest in EXTSEOUT analysis, e.g. using,

```
SET 200 = 34,566, 445, ...
DISP=200
```

2. Repeating or minoring of a SSE is not supported. As a workaround, the user can reuse same FMU multiple times for creating a new SSE together with a SELOC entry and a modified SECONCT entry.
3. All the FMU files used in the assembly run should be placed in the same directory as the MSC Nastran input file.

Example

Examples of a door frame assembly using the SSE capability in SOL 101 are provided in the MSC Nastran Documentation Installation directory: `tp1\odyssee` and `tp16\odyssee` (Note that running these examples requires an existing installation of ODYSSEE CAE).

Model	Description
<code>tp16\odyssee\asm_101_sse_wre.dat</code>	SOL 101 Analysis with Pillar modeled as a SSE. Data recovery in the pillar is enabled through the use of OTMs.
<code>tp1\odyssee\asm_101_sse.dat</code>	SOL 101 Analysis with Pillar modeled as a SSE. Data recovery is not enabled inside the Pillar (OTMs are not output by the FMU) to reduce the solve time.

The basic model is a door frame assembly with four components roof, pillar, front and bottom, refer [Figure 19-5](#). The pillar is modeled as a linear SSE.



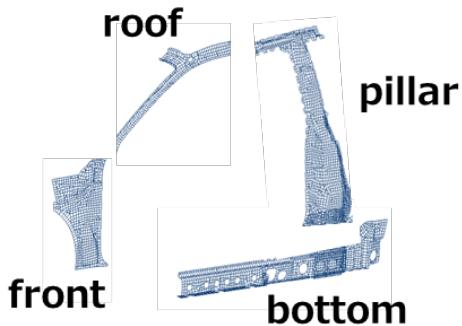


Figure 19-5 Door Frame Assembly Components

References

1. MSC Nastran Superelements and Modules User's Guide.
2. MSC Nastran Dynamic Analysis User's Guide.
3. ODYSSEE CAE – Lunar User's Guide.





20

Grid Point Weight Generator

- Introduction to Grid Point Weight Generation
- Commonly Used Features
- Example with Direction-Dependent Masses



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](#), a basic discussion is given that should satisfy most users. If you need additional information, read [Example with Direction-Dependent Masses](#).

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.

Commonly Used Features

To demonstrate the typical output generated by the GPWG, a small model consisting of four concentrated masses as shown in [Figure 20-1](#) 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.

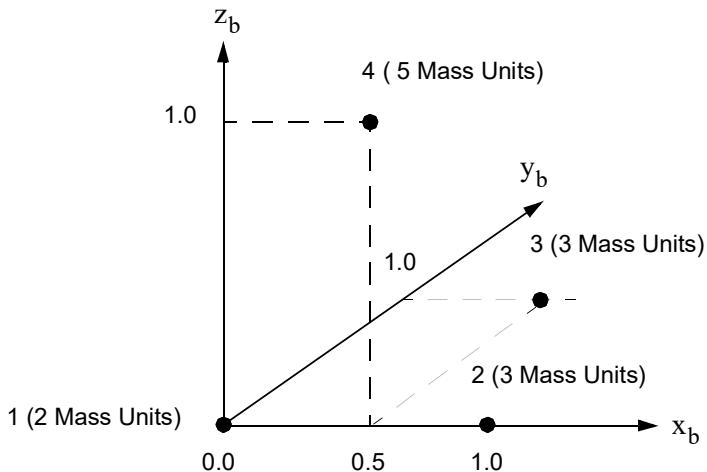


Figure 20-1 Four Concentrated Mass Model



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 20-1](#), the resulting GPWG output is given in [Figure 20-2](#).

```
OUTPUT FROM GRID POINT WEIGHT GENERATOR
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 20-2 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 \(Case\)](#) = 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.

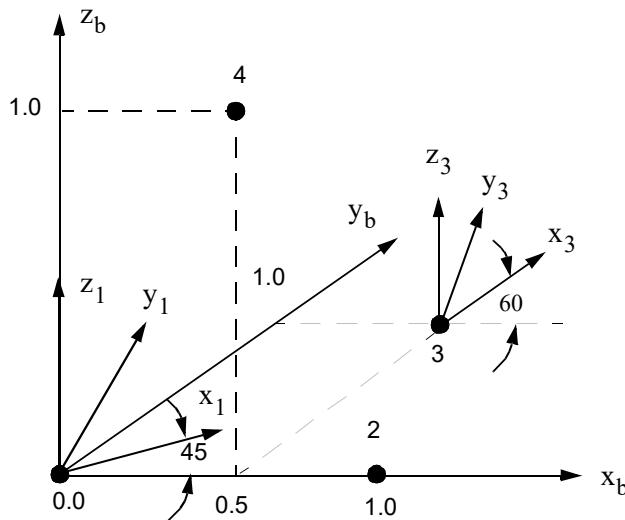
Example with Direction-Dependent Masses

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 20-3](#)). 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 20-1](#).



[Figure 20-3](#) Four Concentrated Mass Model.

[Table 20-1](#) 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 20-4](#).



```
*** 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 O

* 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 *

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 *

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

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 *

I (Q)
* 4.463246E+00      *
* 6.075616E+00      *
*           2.893001E+00 *

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 *
```

Rigid-Body Mass Properties Matrix About the Reference Point

Transformation from the Principal Mass to the Basic Direction

Center of Gravity Relative to the Reference Point in the Principal Mass Axes System

Moments of Inertia with Respect to Principal Mass Axes for the Center of Gravity

Principal Moments of Inertia

Transformation from the Principal Direction of the Momental Ellipsoid to the Principal Mass Axes

Figure 20-4 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 point 1 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 20-4](#). 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 20-5](#).



2	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1
0	3	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2
0	0	5	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	3
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	4
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	5
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	6
0	0	0	0	0	0	2	0	0	0	0	0	0	0	0	0	0	0	0	0	7
0	0	0	0	0	0	0	3	0	0	0	0	0	0	0	0	0	0	0	0	8
0	0	0	0	0	0	0	0	5	0	0	0	0	0	0	0	0	0	0	0	9
0	0	0	0	0	0	0	0	0	2	0	0	0	0	0	0	0	0	0	0	10
0	0	0	0	0	0	0	0	0	0	3	0	0	0	0	0	0	0	0	0	11
0	0	0	0	0	0	0	0	0	0	0	5	0	0	0	0	0	0	0	0	12
0	0	0	0	0	0	0	0	0	0	0	0	2	0	0	0	0	0	0	0	13
0	0	0	0	0	0	0	0	0	0	0	0	0	3	0	0	0	0	0	0	14
0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	0	0	0	0	0	15
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	3	0	0	0	0	16
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	0	0	0	17
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	0	0	18
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	3	0	0	0	19
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	0	0	20
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	0	21
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	22
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	23
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	2	24

Figure 20-5 Global Mass Matrix

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.

- To generate the 6×6 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 [Equation \(20-1\)](#).

$$\{u_g\} = [D]\{u_O\} \text{ (Reference Point)} \quad (20-1)$$



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 [Equation \(20-2\)](#).

$$\{r\}_i = \{R\}_i - \{R\}_0 = \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \end{Bmatrix} \quad (20-2)$$

The location vectors for the example are as follows:

$$\{r\}_1 = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_2 = \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_3 = \begin{Bmatrix} .5 \\ 1 \\ 0 \end{Bmatrix}; \{r\}_4 = \begin{Bmatrix} .5 \\ 0 \\ 1 \end{Bmatrix} \quad (20-3)$$

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 [Equation \(20-4\)](#).

$$[T_r]_i = \begin{bmatrix} 0 & r_3 & -r_2 \\ -r_3 & 0 & r_1 \\ r_2 & -r_1 & 0 \end{bmatrix} \quad (20-4)$$

For the example problem, the grid point transformation matrices are

$$[Tr]_1 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}; [Tr]_2 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & -1 & 0 \end{bmatrix}; [Tr]_3 = \begin{bmatrix} 0 & 0 & -1 \\ 0 & 0 & .5 \\ 1 & -.5 & 0 \end{bmatrix}; [Tr]_4 = \begin{bmatrix} 0 & 1 & 0 \\ -1 & 0 & .5 \\ 0 & -.5 & 0 \end{bmatrix} \quad (20-5)$$

The coordinate system transformation matrices from the global coordinate system to the basic system coordinates are given by the direction cosine matrices as follows:



$$[Ti]_1 = \begin{bmatrix} \cos(45^\circ) & -\sin(45^\circ) & 0 \\ \sin(45^\circ) & \cos(45^\circ) & 0 \\ 0 & 0 & 1 \end{bmatrix}; [Ti]_2 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad (20-6)$$

$$[Ti]_3 = \begin{bmatrix} \cos(60^\circ) & -\sin(60^\circ) & 0 \\ \sin(60^\circ) & \cos(60^\circ) & 0 \\ 0 & 0 & 1 \end{bmatrix}; [Ti]_4 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

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 [Equation \(20-7\)](#).

$$[d]_i = \begin{bmatrix} Ti^T & | & Ti^T \cdot Tr \\ \hline 0 & | & Ti^T \end{bmatrix} \quad (20-7)$$

The rows of the each $[d]_i$ form the columns of the global transformation matrix $[D]^T$ as shown in [Equation \(20-8\)](#).

$$[D]^T = [d_1^T | d_2^T | \dots | d_n^T] \quad (20-8)$$

Using [Equation \(20-8\)](#), the global transformation matrix for the example is

$$[D]^T = \left[\begin{array}{cccccccc|ccccc|ccccc|ccccc} .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & .5 & -.866 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ .707 & .707 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & .866 & .5 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & .5 & -.866 & 0 & 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & .707 & .707 & 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 & -0.5 & .866 & .5 & 0 & 1 & 0 & -0.5 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 1 & -0.067 & 1.116 & 0 & 0 & 0 & 1 & 0 & .5 & 0 & 0 & 0 & 1 \end{array} \right] \quad (20-9)$$

Using the global transformation matrix $[D]$, the rigid-body mass matrix about the reference point in the basic coordinate system $[MO]$ is obtained by [Equation \(20-10\)](#).

$$[MO] = [D]^T [MJJ][D] \quad (20-10)$$

For the example, $[MO]$ is determined to be



$$[MO] = \begin{bmatrix} 9.25 & -0.933 & 0. & 0. & 2. & -2.967 \\ -0.933 & 10.75 & 0. & -3. & 0. & 6.058 \\ 0. & 0 & 20. & 5. & -10. & 0. \\ 0. & -3 & 5. & 8. & -2.5 & -1.5 \\ 2. & 0 & -10. & -2.5 & 9.5 & 0. \\ -2.967 & 6.058 & 0. & -1.5 & 0. & 7.496 \end{bmatrix} \quad (20-11)$$

Comparing the results shown in [Equation \(20-11\)](#) to $[MO]$ generated by the GPWG module ([Figure 20-4](#)) 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 [Equation \(20-12\)](#).

$$[MO] = \begin{bmatrix} \bar{M}^t & \bar{M}^{tr} \\ \bar{M}^{rt} & \bar{M}^r \end{bmatrix} \quad (20-12)$$

where the superscripts t and r refer to translation and rotation, respectively.

For this example, the translational mass partition is given by

$$[\bar{M}^t] = \begin{bmatrix} 9.25 & -0.933 & 0 \\ -0.933 & 10.75 & 0 \\ 0 & 0 & 20. \end{bmatrix} \quad (20-13)$$

A check is made for coupling as follows:

$$\delta = \sqrt{\sum (\bar{M}_{ij}^t)^2} \quad i = j \quad (20-14)$$

$$\varepsilon = \sqrt{\sum (\bar{M}_{ij}^t)^2} \quad i > j$$

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 $\varepsilon = (.933^2 + 0^2 + 0^2)^{.5}$. The ratio $\varepsilon/\delta = .038$ agrees with the System Warning Message 3042 shown in [Figure 20-4](#).



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 $[S]$.

$$[S] = [\{e_1\}, \{e_2\}, \{e_3\}] \quad (20-15)$$

Using this eigenvector matrix, the partitions of the rigid-body mass matrix with respect to the principal mass direction are computed by [Equation \(20-16\)](#).

$$\begin{aligned} [\bar{M}^t] &= [S]^T [\bar{M}^t] [S] \\ [\bar{M}^{tr}] &= [S]^T [\bar{M}^{tr}] [S] \\ [\bar{M}^r] &= [S]^T [\bar{M}^r] [S] \end{aligned} \quad (20-16)$$

The $[S]$ matrix for the example is given by

$$[S] = \begin{bmatrix} .432 & .902 & 0. \\ -.902 & .432 & 0. \\ 0. & 0. & 1. \end{bmatrix} \quad (20-17)$$

The $[S]$ matrix is printed after the \bar{M} matrix as shown in [Figure 20-4](#). 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 [Equation \(20-18\)](#).

$$M_x = M_{11}^t$$

$$M_y = M_{22}^t$$

$$M_z = M_{33}^t$$

The “centers of gravity” are



$$\begin{aligned}
 X_x &= \frac{M_{11}^{tr}}{M_x}, \quad Y_x = \frac{-M_{13}^{tr}}{M_x}, \quad Z_x = \frac{M_{12}^{tr}}{M_x} \\
 X_y &= \frac{M_{23}^{tr}}{M_y}, \quad Y_y = \frac{M_{22}^{tr}}{M_y}, \quad Z_y = \frac{-M_{21}^{tr}}{M_y} \\
 X_z &= \frac{-M_{32}^{tr}}{M_z}, \quad Y_z = \frac{M_{31}^{tr}}{M_z}, \quad Z_z = \frac{M_{33}^{tr}}{M_z}
 \end{aligned} \tag{20-18}$$

As can be seen, the center of gravity location is not a unique location. The center of gravity location is computed separately for the 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 20-2 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 20-2 is the same as shown in Figure 20-4.

5. 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 Equation (20-19).

$$\begin{aligned}
 I_{11}^{(S)} &= M_{11}^r - M_y Z_y^2 - M_z Y_z^2 \\
 I_{12}^{(S)} &= I_{21}^{(S)} = -M_{12}^r - M_z X_z Y_z \\
 I_{13}^{(S)} &= I_{31}^{(S)} = -M_{13}^r - M_y X_y Z_y \\
 I_{22}^{(S)} &= M_{22}^r - M_z X_z^2 - M_x Z_x^2 \\
 I_{32}^{(S)} &= I_{23}^{(S)} = -M_{23}^r - M_x Y_x Z_x \\
 I_{33}^{(S)} &= M_{33}^r - M_x Y_x^2 - M_y X_y^2
 \end{aligned} \tag{20-19}$$

For the example, the inertia matrix $[I(S)]$ is given by



$$[I(S)] = \begin{bmatrix} 4.377 & -0.877 & 0.662 \\ -0.877 & 5.632 & -0.342 \\ 0.662 & -0.342 & 3.432 \end{bmatrix} \quad (20-20)$$

6. 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 $[I(S)]$. This is because historically, Nastran defined the off-diagonal terms with the opposite sign than most text books. For the inertia matrix given in $[\tilde{I}]$, is given by

$$[I(S)] = \begin{bmatrix} 4.377 & 0.877 & -0.662 \\ 0.877 & 5.632 & 0.342 \\ -0.662 & 0.342 & 3.432 \end{bmatrix} \quad (20-21)$$

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 [Equation \(20-22\)](#).

$$[I(Q)] = [Q]^T [\tilde{I}] [Q] \quad (20-22)$$

For the example, $[Q]$ and $[I(Q)]$ are given by

$$[I(Q)] = \begin{bmatrix} 4.463 & 0 & 0 \\ 0 & 6.076 & 0 \\ 0 & 0 & 2.893 \end{bmatrix} \quad (20-23)$$

$$[Q] = \begin{bmatrix} 0.720 & 0.459 & 0.521 \\ -0.372 & 0.889 & -0.269 \\ -0.586 & 0 & 0.810 \end{bmatrix}$$

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 [Equation \(20-23\)](#) are in agreement with those given in [Figure 20-4](#). The example for the four mass model is now complete.



21 Special Topics

- Multiple Mass Configuration



Multiple Mass Configuration

The multiple mass configurations (MMC) capability is described here which provides the user with a convenient method for constructing and analyzing additional mass cases in MSC Nastran. This can be used, for example, to analyze several different payloads or fuel loadings in a single run. The mass increments are specified using the MASSID bulk data section. Mass combinations are then specified using MASSSET bulk data entry. The mass combinations are subcase selectable to provide an appropriate method similar to selecting SPCs and MPCs boundary conditions in a subcase to conduct the analysis using appropriate mass. The complete discussion of the MMC capability is provided in the [Multiple Mass Configuration](#) (Ch. 18) in the *MSC Nastran Dynamic Analysis User's Guide*. It should be noted that MMC capability does not support inertia relief in SOL101 with part superelements.

MMC capability in Sol 101 can be used to conduct:

1. Static analysis with gravity (mass dependent) loading applied to user specified mass cases:

```
SOL      101
CEND
SUBCASE  1
    SUBTITLE=Gravity Load, Base Case
    LOAD = 990
    ...
SUBCASE  2
    SUBTITLE=Gravity Load, Mass Case (MASSSET) 11
    MASSSET = 11
    LOAD = 990
    ...
BEGIN BULK
MASSSET  11      1.0      1.0      0       1.0      101
...
$ Grav Load
GRAV,990,,386.09,0.,0.0,-1.0
...
BEGIN massid = 101
  conn2   6002     70        250.0    0.        0.        -12.
          4500.0   0.        50250.   0.        0.        50250.
  conn2   6004     31        250.0    0.        0.        -12.
          4500.0   0.        50250.   0.        0.        50250.
...
ENDDATA
```

Example of MMC with Sol 101 gravity loading can be found in qa/nastran/tpl/mmc/mmc101s1.dat.



2. Static analysis with inertia relief applied to user specified mass cases.

```
SOL      101
CEND
SUBCASE 1
    SUBTITLE= Inrel with Base Mass Case
    LOAD = 2
    ...
SUBCASE 2
    SUBTITLE= Inrel with Mass Case (MASSSET) 2000
MASSSET = 11
    LOAD = 2
    ...
BEGIN BULK
PARAM,INREL,-2
MASSSET 200      1.0      1.0      0       1.0      2
...
BEGIN massid = 2
conm2   6002      70          250.0    0.        0.        -12.
        4500.0   0.        50250.    0.        0.        50250.
conm2   6004      31          250.0    0.        0.        -12.
        4500.0   0.        50250.    0.        0.        50250.
...
ENDDATA
```

Examples of MMC with Sol 101 inertia relief can be found in qa/nastran/tpl/mmc/mmcinrel1_1 and mmcinrel2_2.





A

Sample Output Listings

- Examples List



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 [Table 1-1](#).

Table 1-1 List of Figures for Output Formats

Figure 1-1	Forces in Scalar Springs
Figure 1-2	Stresses in Scalar Springs
Figure 1-3	Forces in CROD Elements
Figure 1-4	Stresses in CROD Elements
Figure 1-5	Forces in CBAR Elements
Figure 1-6	Stresses in CBAR Elements
Figure 1-7	Force Distribution in CBAR Elements (CBARAO)
Figure 1-8	Stress Distribution in CBAR Elements (CBARAO)
Figure 1-9	Forces in CBEAM Elements
Figure 1-10	Stresses in CBEAM Elements
Figure 1-11	Forces in CBEAM3 Elements
Figure 1-12	Stresses in CBEAM3 Elements
Figure 1-13	Strains in CBEAM3 Elements
Figure 1-14	Forces in CBEND Elements
Figure 1-15	Stresses in CBEND Elements
Figure 1-16	Forces in CBUSH Element
Figure 1-17	Forces Acting on Shear Panel Elements
Figure 1-18	Stresses in Shear Panels
Figure 1-19	Forces in CQUAD4 Elements (Without Corner Option - Default) Forces in CQUADR Elements (With Center Option)
Figure 1-20	Stresses in CQUAD4 Elements (Without Corner Option - Default) Stresses in CQUADR Elements (With Center Option)
Figure 1-21	Strains in CQUAD4 Elements (Without Corner Option - Default) Strains in CQUADR Elements (With Center Option)
Figure 1-22	Forces in CQUAD4 Elements (With Corner Option) Forces in CQUADR Elements (With Corner Option - Default)
Figure 1-23	Stresses in CQUAD4 Elements (With Corner Option) Stresses in CQUADR Elements (With Corner Option - Default)
Figure 1-24	Strains in CQUAD4 Elements (With Corner Option) Strains in CQUADR Elements (With Corner Option - Default)



Table 1-1 List of Figures for Output Formats

Figure 1-25	Forces in CQUAD8 Elements
Figure 1-26	Stresses in CQUAD8 Elements
Figure 1-27	Strains in CQUAD8 Elements
Figure 1-28	Stresses in CHEXA Elements
Figure 1-29	Strains in CHEXA Elements
Figure 1-30	Stresses and Strains for Layered HEXA Elements (PCOMPLS - SOL 400)
Figure 1-31	Interlaminar Stresses for Layered HEXA Elements (PCOMPLS - SOL 400)
Figure 1-32	Forces in CWELD Elements
Figure 1-33	Forces and Stresses in CFAST Element
Figure 1-34	Grid Point Force Balance
Figure 1-35	Grid Point Stress Output
Figure 1-36	Grid Point Stress Discontinuities Output
Figure 1-37	Element Stress Discontinuities Output
Figure 1-38	Screened Stresses in CBAR/CBEAM Elements
Figure 1-39	Stresses in Composite CBEAM3 Elements
Figure 1-40	Element Strain Energy
Figure 1-41	Grid Point Singularity Table
Figure 1-42	User Information Message 5293
Figure 1-43	OLOAD Resultant
Figure 1-44	SPCFORCE Resultant
Figure 1-45	Maximum SPCFORCES, DISPLACEMENT, and Applied Loads
Figure 1-46	MAXMIN (old form) DISPLACEMENT, VELOCITY, ACCELERATION, SPCFORCE, and MPCFORCE
Figure 1-47	Sample of MAXMIN(DEF) Grid Point Output
Figure 1-48	Sample of MAXMIN(DEF) Element Output
Figure 1-49	Displacement Vectors
Figure 1-50	Displacement Vectors with DISP(CID)=all
Figure 1-51	Load Vectors
Figure 1-52	Database Summary Table
Figure 1-53	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
1	-6.366724E+02	2	3.633277E+02
		3	-5.537512E+02
		4	3.653757E+02

Figure 1-1 Forces in Scalar Springs

Notes: Standard FORCE output, requested in Case Control Section.

See [Scalar Elements, 79](#) for the interpretation of the CELASi output.

This output is typical of all scalar elements (CELAS1, CELAS2, CELAS3, CELAS4).

S T R E S S 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.	STRESS	ELEMENT ID.	STRESS
1	-6.366723E+04	2	3.633277E+04
		3	-5.537512E+04
		4	3.653757E+04

Notes: 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.

This output applies only to the CELAS1 and CELAS2 Bulk Data entries.

Figure 1-2 Stresses in Scalar Springs



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
1	2.495156E+03	0
3	-2.544570E+03	0
5	4.990313E+02	0

ELEMENT ID.	AXIAL FORCE	TORQUE
2	2.554449E+03	0
4	-2.504844E+03	0

Figure 1-3 Forces in CROD Elements

Notes: See [One-Dimensional Elements, 90](#) for definition of positive torque (same as CBAR).

Positive axial force is tension.

Standard FORCE output, requested in Case Control Section.



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
1	6.237891E+02	.0		
3	-6.361426E+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.

Notes: 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 1-4 Stresses in CROD Elements

F O R C E S I N B A R E L E M E N T S (C B A R)				
ELEMENT ID.	BEND-MOMENT END-A	BEND-MOMENT END-B	- SHEAR -	AXIAL FORCE TORQUE
1	PLANE 1	PLANE 2	PLANE 1	PLANE 2
2	2.586560E+01	-1.519694E+01	1.619619E+00	4.542251E+00
3	2.586560E+01	1.519694E+01	1.619619E+00	-4.542251E+00
	-5.172854E+01	0.0	-3.240443E+00	0.0
				-4.336907E-02 0.0
				3.726698E+03 2.810185E+00
				3.726698E+03 -2.810185E+00
				-7.453428E+03 0.0

Notes: See Figures 4-9 and 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 1-5 Forces in CBAR Elements



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
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.

Notes: **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 1-6 Stresses in CBAR Elements



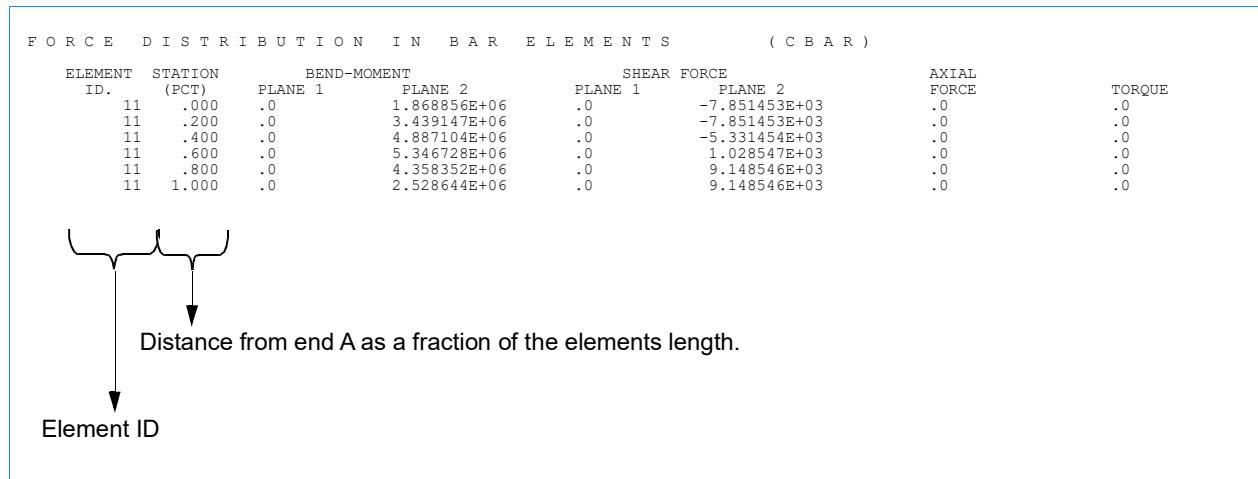


Figure 1-7 Force Distribution in CBAR Elements (CBARAO)

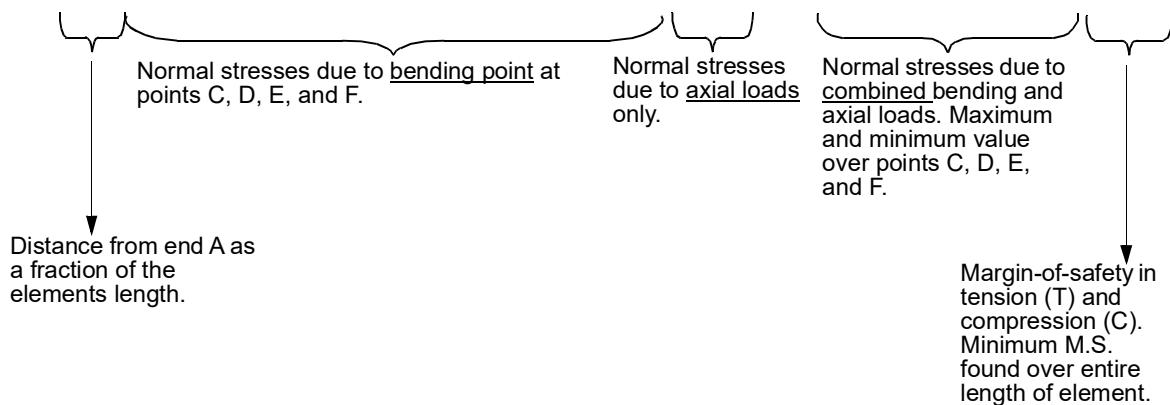
Notes: Optional output for the CBAR element using the CBARAO entry.

See [One-Dimensional Elements, 90](#) for a discussion of plane 1 and 2.

Force output must be requested in Case Control Section.



STRESS DISTRIBUTION IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	STATION (PCT)	SXC	SXD	SXE	SXF	AXIAL	S-MAX	S-MIN	M.S.-T M.S.-C
11	.000	-1.868856E+04	-1.868856E+04	1.868856E+04	1.868856E+04	.0	1.868856E+04	-1.868856E+04	6.1E-01
11	.200	-3.439147E+04	-3.439147E+04	3.439147E+04	3.439147E+04	.0	3.439147E+04	-3.439147E+04	-1.3E-01
11	.400	-4.887105E+04	-4.887105E+04	4.887105E+04	4.887105E+04	.0	4.887105E+04	-4.887105E+04	-3.9E-01
11	.600	-5.346729E+04	-5.346729E+04	5.346729E+04	5.346729E+04	.0	5.346729E+04	-5.346729E+04	-4.4E-01
11	.800	-4.358352E+04	-4.358352E+04	4.358352E+04	4.358352E+04	.0	4.358352E+04	-4.358352E+04	-3.1E-01
11	1.000	-2.528644E+04	-2.528644E+04	2.528644E+04	2.528644E+04	.0	2.528644E+04	-2.528644E+04	1.9E-01



Notes: The default position for points C, D, E and F is on the neutral axis.

This is special output requested by including CBARAO and/or PLOAD1 entries in the Bulk Data Section along with a STRESS request in the 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. A 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. An 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 1-8 Stress Distribution in CBAR Elements (CBARAO)



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)									
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	

Figure 1-9 Forces in CBEAM Elements

Notes:

See [One-Dimensional Elements, 90](#) 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.



ELEMENT-ID	GRID	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 LENGTH	DIST/ 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			

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.

Distance from end A as a fraction of the element's length.

Element ID

Notes: 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 1-10 Stresses in CBEAM Elements



F O R C E S		I N	B E A M 3	E L E M E N T S	(C B E A M 3)							
ELEMENT-ID	GRID/ GAUSS	- BENDING MOMENTS -			- SHEAR FORCES -			AXIAL FORCE	TOTAL TORQUE	BI-SHEAR FORCE	BI-MOMENT	
		MY	MZ	QY	QZ							
0	2901	2901	0.0	-1.156075E+03	-2.348231E+01	0.0	1.087807E+02	0.0	0.0	0.0	0.0	
		2902	0.0	-1.125649E+02	-1.096482E+02	0.0	1.902465E+01	0.0	0.0	0.0	0.0	
		2903	0.0	-6.343201E+02	-6.656523E+01	0.0	6.390266E+01	0.0	0.0	0.0	0.0	
0	3901	3901	0.0	-1.048109E+03	-3.558532E+00	0.0	1.042715E+02	0.0	0.0	0.0	0.0	
		3903	0.0	-7.528904E+02	-7.162810E+01	0.0	7.585925E+01	0.0	0.0	0.0	0.0	
		3904	0.0	-9.004999E+02	-3.759332E+01	0.0	9.006535E+01	0.0	0.0	0.0	0.0	
0	3902	3903	0.0	-7.267761E+02	-7.611831E+01	0.0	7.135274E+01	0.0	0.0	0.0	0.0	
		3902	0.0	-1.843143E+01	-1.042837E+02	0.0	3.180659E+00	0.0	0.0	0.0	0.0	
		3905	0.0	-3.726037E+02	-9.020099E+01	0.0	3.726670E+01	0.0	0.0	0.0	0.0	

Figure 1-11 Forces in the CBEAM3 Elements

S T R E S S E S		I N	B E A M 3	E L E M E N T S	(C B E A M 3)							
ELEMENT-ID	GRID/ GAUSS	STRESS COMPONENT			- STRESSES IN LOCAL COORDINATE SYSTEM -			S-XF	S-MAX	S-MIN	M.S.-T	M.S.-C
		SXC	SXD	SXE	SXF	S-MAX	S-MIN					
0	2901	SX	5.889157E+04	1.087807E+03	-5.671596E+04	1.087807E+03	5.889157E+04	-5.671596E+04	-9.8E-01	-9.6E-01		
		SY	-2.348231E+02	-2.348231E+02	-2.348231E+02	-2.348231E+02	-2.348231E+02	-2.348231E+02				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	2902	SX	5.818494E+03	1.902465E+02	-5.438001E+03	1.902465E+02	5.818494E+03	-5.438001E+03				
		SY	-1.096482E+03	-1.096482E+03	-1.096482E+03	-1.096482E+03	-1.096482E+03	-1.096482E+03				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	2903	SX	3.235503E+04	6.390266E+02	-3.107698E+04	6.390266E+02	3.235503E+04	-3.107698E+04				
		SY	-6.656523E+02	-6.656523E+02	-6.656523E+02	-6.656523E+02	-6.656523E+02	-6.656523E+02				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3901	SX	5.344819E+04	1.042714E+03	-5.136276E+04	1.042714E+03	5.344819E+04	-5.136276E+04	-9.8E-01	-9.6E-01		
		SY	-3.558532E+01	-3.558532E+01	-3.558532E+01	-3.558532E+01	-3.558532E+01	-3.558532E+01				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3903	SX	3.840311E+04	7.585925E+02	-3.688593E+04	7.585925E+02	3.840311E+04	-3.688593E+04				
		SY	-7.162810E+02	-7.162810E+02	-7.162810E+02	-7.162810E+02	-7.162810E+02	-7.162810E+02				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3904	SX	4.592565E+04	9.006535E+02	-4.412434E+04	9.006535E+02	4.592565E+04	-4.412434E+04				
		SY	-3.759331E+02	-3.759331E+02	-3.759331E+02	-3.759331E+02	-3.759331E+02	-3.759331E+02				
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		

Figure 1-12 Stresses in the CBEAM3 Elements

S T R A I N S		I N	B E A M 3	E L E M E N T S	(C B E A M 3)							
ELEMENT-ID	GRID/ GAUSS	STRAIN COMPONENT			- STRAINS IN LOCAL COORDINATE SYSTEM -			S-MAX	S-MIN	M.S.-T	M.S.-C	
		SXC	SXD	SXE	SXF	S-MAX	S-MIN					
0	2901	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	-1.0E+00	
		SY	8.765757E-17	8.765757E-17	8.765757E-17	8.765757E-17	8.765757E-17	8.765757E-17	8.765757E-17	8.765757E-17		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	2902	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	
		SY	5.160202E-17	5.160202E-17	5.160202E-17	5.160202E-17	5.160202E-17	5.160202E-17	5.160202E-17	5.160202E-17		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	2903	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	
		SY	6.962979E-17	6.962979E-17	6.962979E-17	6.962979E-17	6.962979E-17	6.962979E-17	6.962979E-17	6.962979E-17		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3901	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	-1.0E+00	
		SY	-7.548301E-16	-7.548301E-16	-7.548301E-16	-7.548301E-16	-7.548301E-16	-7.548301E-16	-7.548301E-16	-7.548301E-16		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3903	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	
		SY	-4.543672E-16	-4.543672E-16	-4.543672E-16	-4.543672E-16	-4.543672E-16	-4.543672E-16	-4.543672E-16	-4.543672E-16		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
0	3904	SX	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	1.000000E-01	
		SY	-6.045987E-16	-6.045987E-16	-6.045987E-16	-6.045987E-16	-6.045987E-16	-6.045987E-16	-6.045987E-16	-6.045987E-16		
		SZ	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0		

Figure 1-13 Strains in the CBEAM3 Elements



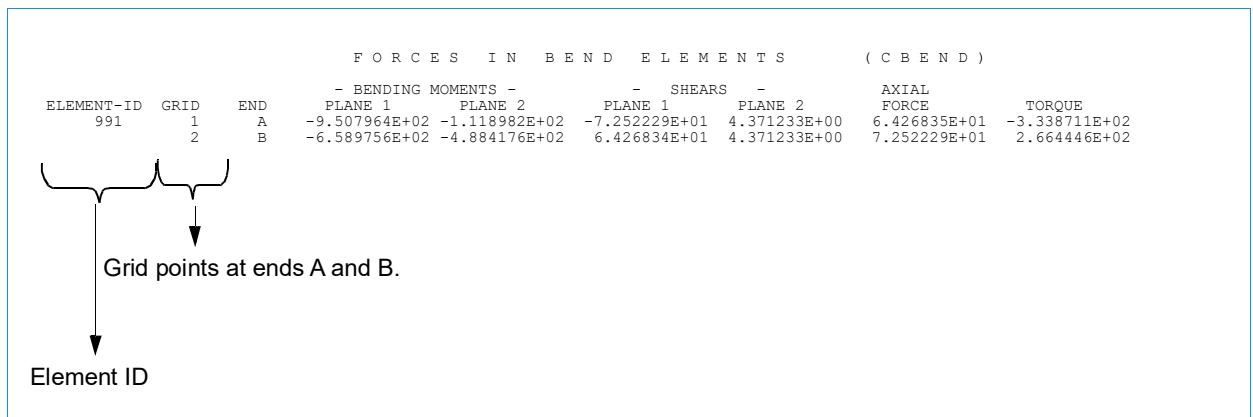


Figure 1-14 Forces in CBEND Elements

Notes: See [One-Dimensional Elements, 90](#) 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.



S T R E S S E S I N B E N D E L E M E N T S (C B E N D)												
ELEMENT-ID	GRID	END	CIRC.	ANG.	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
991	1	A	0	0	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	3.8E-01	
	2	B	0	0	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+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.

Notes: 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 1-15 Stresses in CBEND Elements



		F O R C E S I N B U S H E L E M E N T S (C B U S H)					
	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

Note: The forces are in the CBUSH element coordinate system.

Figure 1-16 Forces in CBUSH Element

		F O R C E S A C T I N G O N S H E A R P A N E L E L E M E N T S (CSHEAR)																
ELEMENT	POINT 1	POINT 2	POINT 3	POINT 4	F-FROM-4	F-FROM-2	F-FROM-1	F-FROM-3	F-FROM-4	KICK-1	SHEAR-12	KICK-2	SHEAR-23	KICK-3	SHEAR-34	KICK-4	SHEAR-41	
	===== POINT 1 =====	===== POINT 2 =====	===== POINT 3 =====	===== POINT 4 =====														
ELEMENT ID	F-FROM-4	F-FROM-2	F-FROM-3	F-FROM-4	F-FROM-4	F-FROM-2	F-FROM-1	F-FROM-3	F-FROM-4	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	1.49620E+00	KICK-1	SHEAR-12	KICK-2	SHEAR-23	KICK-3	SHEAR-34	KICK-4	SHEAR-41	
	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01	.0									
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	1.28268E+00	KICK-2	SHEAR-12	KICK-3	SHEAR-23	KICK-4	SHEAR-34	KICK-1	SHEAR-41	
	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01	.0									

Figure 1-17 Forces Acting on Shear Panel Elements

Notes: See [Two-Dimensional Elements, 154](#) for definitions of element coordinate system and force components.

Standard FORCE output, requested in Case Control Section.



S T R E S S E S I N S H E A R P A N E L S (C S H E A R)			
ELEMENT ID.	MAX SHEAR	AVG SHEAR	SAFETY MARGIN
11	2.992391E+00	-2.992391E+00	1.5E+00



Maximum and average shear stresses and margin-of-safety due to shear only.

Notes: 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 1-18 Stresses in Shear Panels

F O R C E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)									
ELEMENT ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -		
	FX	FY	FXY	MX	MY	MMX	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).

Traverse shear forces (force/length).

Figure 1-19 Forces in CQUAD4 Elements (Without Corner Option - Default)
Forces in CQUADR Elements (With Center Option)

Notes: Positive force is tension.

See Two-Dimensional Elements, 154 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.

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		
1	-5.00000E-02	-7.523499E+03	-1.793419E+03	-3.373840E+02	-86.6419	-1.773622E+03	-7.543295E+03	6.831404E+03	
	5.00000E-02	7.476501E+03	1.786722E+03	3.313992E+02	3.3222	7.495738E+03	1.767485E+03	6.786861E+03	
2	-5.00000E-02	-4.515933E+03	-3.544799E+02	-7.011219E+01	-89.0350	-3.532990E+02	-4.517114E+03	4.351235E+03	
	5.00000E-02	4.484066E+03	3.550410E+02	6.498148E+01	.9014	4.485089E+03	3.540186E+02	4.318975E+03	
3	-5.00000E-02	-1.508649E+03	3.605036E+02	1.940267E+02	84.1357	3.804320E+02	-1.528577E+03	1.750085E+03	
	5.00000E-02	1.491351E+03	-3.600570E+02	-2.035071E+02	-6.1993	1.513457E+03	-3.821626E+02	1.736371E+03	

Fiber distance Z1 and Z2 specified on the PSHELL property entry

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.

Figure 1-20 Stresses in CQUAD4 Elements (Without Corner Option - Default)
Stresses in CQUADR Elements (With Center Option)

Notes: This is standard STRESS output requested in Case Control Section.

This output is typical for all plate elements.

See Two-Dimensional Elements, 154 for definition of element coordinate system.



STRAINS IN QUADRILATERAL ELEMENTS (QUAD4)									
ELEMENT ID.	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			ANGLE	PRINCIPAL STRAINS (ZERO SHEAR)		MINOR VON MISES	MAJOR VON MISES
		NORMAL-X	NORMAL-Y	SHEAR-XY		ANGLE	MAJOR		
1 .0	-1.000000E+00	-2.249436E-06	3.701331E-07	-7.780218E-07	-81.7292	4.266812E-07	-2.305984E-06	1.697520E-06	9.657605E-03
2 .0	-1.000000E+00	-1.392596E-02	9.198587E-04	-1.738835E-03	-86.6598	9.706012E-04	-1.397670E-02	6.327650E-06	1.327650E-06
3 .0	-1.000000E+00	-1.601760E-06	5.060585E-07	-6.669924E-07	-81.2203	5.575652E-07	-1.653267E-06	6.625173E-03	6.625173E-03
		-8.787143E-03	1.990479E-03	-3.512435E-04	-89.0667	1.993340E-03	-8.790004E-03	9.941859E-07	9.941859E-07
		-8.715881E-07	2.817985E-07	-1.232463E-06	-66.5508	5.490937E-07	-1.138883E-06	3.270770E-03	2.904646E-03
		-3.216168E-03	1.620561E-03	1.033588E-03	83.9688	1.675163E-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.

Figure 1-21 Strains in CQUAD4 Elements (Without Corner Option - Default)
Strains in CQUADR Elements (With Center Option)

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, 154](#) for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.



FORCES IN QUADRILATERAL ELEMENTS (QUAD4)								OPTION = BILIN		
ELEMENT ID	GRID-ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -		
		FX	FY	FXY	MX	MY	MXY	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	-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.

Figure 1-22 Forces in CQUAD4 Elements (With Corner Option)
Forces in CQUADR Elements (With Corner Option - Default)

Notes:

Positive force is tension.

See [Two-Dimensional Elements, 154](#) 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.



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)							OPTION = BILIN		
ELEMENT ID	GRID-ID	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	-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	-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	-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	-5.000000E-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.

Figure 1-23 Stresses in CQUAD4 Elements (With Corner Option)

Stresses in CQUADR Elements (With Corner Option - Default)

Notes: 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, 154 for definition of element coordinate system.**

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	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	-2.253929E-06	3.851096E-07	-1.360647E-06	-76.3625	5.501681E-07	-2.418988E-06	1.823920E-06
		-1.000000E+00	-1.392596E-02	9.198587E-04	-1.738836E-03	-86.6598	9.706012E-04	-1.397670E-02	9.657605E-03
1	.0	-4.048489E-06	.0	8.190221E-07	84.2816	4.100733E-08	-4.089496E-06	2.740102E-06	
	-1.000000E+00	-1.566278E-02	.0	-1.986349E-03	-86.3862	6.272583E-05	-1.572551E-02	1.050464E-02	
2	.0	-4.048489E-06	7.702193E-07	-2.770097E-06	-75.0535	1.139956E-06	-4.418226E-06	3.389972E-06	
	-1.000000E+00	-1.055118E-02	2.096610E-03	-5.262259E-03	-78.7048	2.622131E-03	-1.107670E-02	8.396119E-03	
6	.0	-4.593699E-07	7.702193E-07	-3.540316E-06	-54.5763	2.029306E-06	-1.718457E-06	2.166252E-06	
	-1.000000E+00	-1.184074E-02	1.582825E-03	-1.582825E-03	-86.6375	1.629323E-03	-1.188724E-02	8.520021E-03	
5	.0	-4.593699E-07	.0	4.880284E-08	86.9679	1.292550E-09	-4.606624E-07	3.075400E-07	
	-1.000000E+00	-1.764913E-02	.0	1.693086E-03	87.2602	4.051156E-05	-1.768964E-02	1.180662E-02	

↓

Denotes CEN/4 for the center strain and curvatures and a grid point ID for a corner strain or curvature.



Figure 1-24 Strains in CQUAD4 Elements (With Corner Option)
Strains in CQUADR Elements (With Corner Option - Default)

Corner strain output is available for the plate elements and is requested using the Case Control command STRAIN(CORNER).

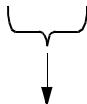
Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.

This output is typical for all plate elements.

See [Two-Dimensional Elements, 154](#) for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.

FORCES IN QUADRILATERAL ELEMENTS (QUAD8)									
ELEMENT ID	GRID-ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -	
		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	-1.014605E+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.

Figure 1-25 Forces in CQUAD8 Elements

Notes: Positive force is tension.

See [Two-Dimensional Elements, 154](#) 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.



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 8)									
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 5.000000E-02	-7.521979E+03 7.478021E+03	-2.218078E+03 2.211569E+03	-2.661170E+01 1.614634E+01	-89.7125 .1757	-2.217945E+03 7.478070E+03	-7.522112E+03 2.211519E+03	6.694611E+03 6.653905E+03
1	-5.000000E-02 5.000000E-02	-8.942510E+03 8.860332E+03	-2.682753E+03 2.658100E+03	-4.751408E+01 2.187377E+01	-89.5651 .2021	-2.682239E+03 8.860409E+03	-8.942870E+03 2.658022E+03	7.948703E+03 7.875326E+03	
2	-5.000000E-02 5.000000E-02	-6.126543E+03 6.048768E+03	-1.731518E+03 1.721540E+03	-3.533652E+01 3.490948E+01	-89.5394 .4622	-1.731234E+03 6.049049E+03	-6.126827E+03 1.721259E+03	5.470668E+03 5.398310E+03	
6	-5.000000E-02 5.000000E-02	-5.915213E+03 5.909476E+03	-1.726948E+03 1.738582E+03	-5.709310E+00 1.041892E+01	-89.9219 .1431	-1.726940E+03 5.909501E+03	-5.915221E+03 1.738556E+03	5.268484E+03 5.260303E+03	
5	-5.000000E-02 5.000000E-02	-9.103648E+03 9.093509E+03	-2.731095E+03 2.728053E+03	-1.788687E+01 -2.616797E+00	-89.8392 .0236	-2.731045E+03 9.093510E+03	-9.103699E+03 2.728052E+03	8.091559E+03 8.082488E+03	



Denotes CEN/4 for the center stress or a grid point ID for a corner stress.

Figure 1-26 Stresses in CQUAD8 Elements

Notes: This is standard STRESS output requested in Case Control Section.

This output is typical for all plate elements.

See Two-Dimensional Elements, 154 for definition of element coordinate system.

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 8)									
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	-2.100242E-06 -1.367111E-02	3.338823E-07 7.035311E-05	-1.360496E-06 -1.111709E-04	-75.3990 -89.7682	5.110861E-07 7.057795E-05	-2.277446E-06 -1.367133E-02	1.714246E+06 9.137837E-03
1	.0 -1.000000E+00	-3.739106E-06 -1.620059E-02	5.151080E-22 2.948739E-20	-3.333241E-06 -1.804084E-04	-69.1422 -89.6810	6.350135E-07 5.022378E-07	-4.374119E-06 -1.620109E-02	3.149165E-06 1.080089E-02	
2	.0 -1.000000E+00	-3.739106E-06 -1.113939E-02	6.677646E-07 1.995355E-04	-5.551511E-08 -1.826396E-04	-89.6391 -89.5386	6.679394E-07 2.002709E-04	-3.739280E-06 -1.114013E-02	2.742746E+06 7.494402E-03	
6	.0 -1.000000E+00	-4.613732E-07 -1.078503E-02	6.677646E-07 8.187693E-05	-6.122495E-07 -4.193340E-05	75.7662 -89.8895	7.454184E-07 8.191738E-05	-5.390330E-07 -1.078507E-02	7.447625E-07 7.217507E-03	
5	.0 -1.000000E+00	-4.613792E-07 -1.655941E-02	-1.419557E-22 2.215808E-22	-2.665477E-06 -3.970219E-05	-49.9101 -89.9313	1.121867E-06 2.379707E-08	-1.583246E-06 -1.655944E-02	1.569352E-06 1.103963E-02	



Denotes CEN/4 for the center strain and curvature and a grid point ID for a corner strain and curvature.



Figure 1-27 Strains in CQUAD8 Elements

Notes: 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, 154](#) for definition of element coordinate system.

Material coordinate systems are specified on MATi entries.



Stress coordinate system definition

- 0 = Basic Coordinate System
- 1 = Element Coordinate System
- +X = Material Coordinate System

Defined on PSOLID entry

Location of Stresses

- GRID = Stresses at center and vertex points
- GAUSS = Stresses at center and Gauss points

Number of active grid points for the element

STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)										
ELEMENT-ID	CORNER GRID-ID	CENTER AND CORNER POINT STRESSES-----			PRINCIPAL	DIR. COSINES			MEAN PRESSURE	VON MISES
		NORMAL	SHEAR	-A-		-B-	-C-			
1	1GRID CS 8 GP				A	1.642750E+01	LX .01	1.00 .03	-6.818652E-02	2.776379E+01
	CENTER	X -1.45905E+01	XY -4.613456E-01	YZ -4.613456E-01	B	-1.560962E+01	LY -.03	.03-1.00		
		Y -6.143415E-01	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	YZ 6.159913E+01	A	2.073488E+02	LX -.59	.81 -.03	-8.944785E+01	1.848900E+02
		Y 6.159913E+01	Z 1.350090E+02	ZX -9.888012E+01	B	-6.360914E-01	LY -.03	-.06-1.00		
	55	X 6.940665E+01	XY 4.212266E+00	YZ 5.992439E+01	C	6.163087E+01	LZ .81	.59 -.06		
		Y 5.992439E+01	Z 1.152021E+02	ZX -8.830753E+01	A	1.836086E+02	LX -.61	.79 -.02	-8.151105E+01	1.614635E+02
					B	9.289488E-01	LY -.02	-.05-1.00		
					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.**Notes:** 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 1-28 Stresses in CHEXA Elements



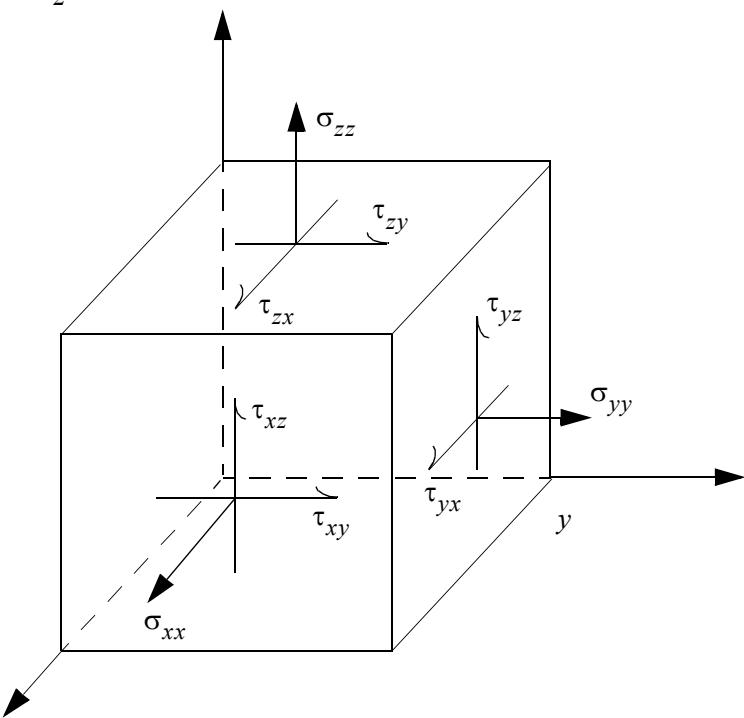
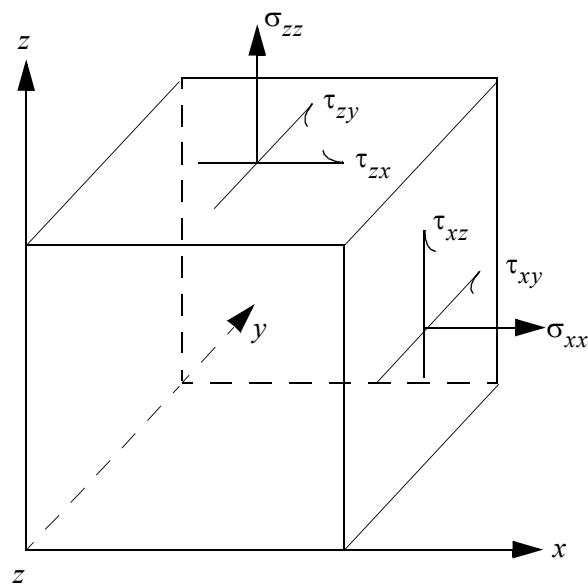


Figure A-28 Stresses in CHEXA Elements (continued)



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	GRID-ID	C O R N E R			S T R A I N S			D I R . C O S I N E S			M E A N P R E S S U R E	V O N M I S E S
		N O R M A L	C E N T E R A N D C O R N E R P O I N T	S H E A R	P R I N C I P A L	-A-	-B-	-C-				
1	1GRID CS 8 GP											
	CENTER	X -3.315102E-06	XY 4.167708E-08	YZ -9.459202E-08	A 2.637197E-06	LX .01	1.00	-.01	1.582972E-07	3.456435E-06		
		Y 2.051620E-07	ZX 1.718858E-07	ZY 2.043855E-07	B -3.316474E-06	LY -.02	-.01	1.00				
	51	X 2.388123E-06	XY -2.219673E-06	YZ 1.648163E-05	C 2.043855E-07	LZ 1.00	-.01	.02				
		Y -1.419658E-07	ZX 2.219673E-06	ZY -4.278821E-06	B -4.278821E-06	LY .09	.07	.99				
	55	X 9.691174E-06	ZX -1.925896E-05	ZY -2.654761E-07	C -2.654761E-07	LZ .82	.56	.12				
		Y 3.108307E-06	XY -2.219673E-06	ZX 1.451382E-05	A 1.451382E-05	LX -.61	.79	-.02	-3.651943E-06	1.105123E-05		
		Z 1.419658E-07	YZ 2.219673E-06	ZY -3.543073E-06	B -.3.543073E-06	LY .11	.06	.99				
		Z 7.705556E-06	ZX -1.727543E-05	ZY -1.491768E-08	C -1.491768E-08	LZ .79	.60	.12				

Notes: This output is typical for the CHEXA, CPENTA, and CTETRA elements.

Standard STRAIN output, requested in Case Control Section.

Figure 1-29 Strains in CHEXA Elements

S T R E S S E S A N D S T R A I N S F O R L A Y E R E D C O M P O S I T E E L E M E N T S														
ELEMENT ID	PLY ID	POINT ID	S T R E S S E S			S T R A I N S			S T R E S S E S			S T R A I N S		
			S11	S22	S33	S12	S23	S31	S11	S22	S33	S12	S23	S31
61	9	1	9.740E-04	2.493E-03	8.303E-04	-6.340E-03	3.720E-04	6.445E-04	-3.552E-09	4.449E-07	1.259E-08	-2.113E-06	2.583E-07	3.638E-07
		2	1.236E-03	2.943E-03	1.034E-03	-6.381E-03	-1.370E-03	6.228E-04	-3.552E-09	5.215E-07	2.540E-08	-2.127E-06	-9.510E-07	3.516E-07
		3	-1.868E-03	2.419E-03	7.749E-04	-5.880E-03	-1.605E-03	-8.073E-05	-3.146E-08	4.449E-07	1.729E-08	-1.960E-06	-1.114E-06	4.557E-08
		4	-1.638E-03	2.850E-03	9.188E-04	-5.922E-03	-3.591E-03	3.786E-06	-3.146E-08	5.215E-07	1.929E-08	-1.974E-06	-2.494E-06	2.137E-09
	8	1	7.482E-04	2.225E-03	7.455E-04	-5.625E-03	3.722E-04	6.461E-04	-4.401E-09	3.973E-07	1.259E-08	-1.875E-06	2.585E-07	3.647E-07
		2	9.925E-04	2.642E-03	9.395E-04	-5.661E-03	-1.370E-03	6.244E-04	-4.401E-09	4.681E-07	2.540E-08	-1.887E-06	-9.512E-07	3.525E-07
		3	-1.669E-03	2.164E-03	7.022E-04	-5.201E-03	-1.604E-03	-8.049E-05	-2.816E-08	3.973E-07	1.729E-08	-1.734E-06	-1.114E-06	4.543E-08
		4	-1.456E-03	2.562E-03	8.359E-04	-5.237E-03	-3.592E-03	4.031E-06	-2.816E-08	4.681E-07	1.929E-08	-1.746E-06	-2.494E-06	2.276E-09

Figure 1-30 Stresses and Strains for Layered HEXA Elements (PCOMPLS - SOL 400)

I N T E R L A M I N A R S T R E S S E S F O R L A Y E R E D C O M P O S I T E E L E M E N T S													BOND INDEX	
ELEMENT ID	PLY ID	POINT ID	N O R M A L			S T R E S S			S H E A R			S T R E S S		
			1	2	3	1	2	3	1	2	3	1	2	3
61	9	1	0.000E+00	-4.914E-24	8.294E-04	6.014E-06	2.862E-07	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		2	0.000E+00	0.000E+00	1.034E-03	5.817E-06	-1.049E-06	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		3	0.000E+00	-4.587E-24	7.742E-04	-7.131E-07	-1.225E-06	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		4	0.000E+00	0.000E+00	9.180E-04	5.684E-08	-2.748E-06	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
	8	1	0.000E+00	2.688E-21	8.446E-04	5.418E-04	2.578E-05	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		2	0.000E+00	0.000E+00	1.063E-03	5.240E-04	-9.454E-05	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		3	0.000E+00	2.618E-21	8.226E-04	-6.424E-05	-1.104E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	
		4	0.000E+00	0.000E+00	9.812E-04	5.121E-06	-2.476E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	



Figure 1-31 Interlaminar Stresses for Layered HEXA Elements (PCOMPLS - SOL 400)

SUBCASE 1									
FORCES IN WELD ELEMENTS (CWELD)									
ELEMENT ID	BEND-MOMENT END-A			BEND-MOMENT END-B			- SHEAR -		AXIAL
109	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX	
110	-2.135744E+02	1.586358E+02	7.728133E+01	1.390482E+01	-2.908656E+02	-1.447359E+02	-8.345229E+01	0.0	
111	2.135769E+02	1.586297E+02	-7.728750E+01	1.390158E+01	2.908643E+02	-1.447281E+02	-8.346558E+01	0.0	
112	-2.363556E+02	-3.536731E+02	-2.396099E+02	-2.361487E+02	3.254667E+00	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	

Figure 1-32 Forces in CWELD Elements

FORCES IN FASTENER ELEMENTS (CFAST)						
ELEMENT ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
777	0.0	2.000000E+03	-5.170960E-10	6.467096E-10	1.722580E-07	1.000000E+01
STRESS IN FASTENER ELEMENTS (CFAST)						
ELEMENT ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
777	0.0	2.000000E+03	-5.170960E-10	6.467096E-10	1.722580E-07	1.000000E+01

NOTE: The Forces and Stress or Strain for the CFAST Elements are the same.
They are all the relative displacement between ends A and B

Figure 1-33 Forces and Stresses in CFAST Element



SUBCASE 1

			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	3	F-OF-SPC	5.000000E+03	4.990313E+02	.0	.0	.0	.0	.0
1	4	ROD	-2.495156E+03	-4.990313E+02	.0	.0	.0	.0	.0
1		*TOTALS*	-2.504844E+03	.0	.0	.0	.0	.0	.0
2		F-OF-SPC	.0	.0	.0	.0	.0	.0	.0
2	1	ROD	-5.000000E+03	5.009687E+02	.0	.0	.0	.0	.0
2	2	ROD	2.495156E+03	-5.009687E+02	.0	.0	.0	.0	.0
2		*TOTALS*	2.504844E+03	.0	.0	.0	.0	.0	.0
3	1	ROD	.0	.0	.0	.0	.0	.0	.0
3	3	ROD	-2.495156E+03	4.990313E+02	.0	.0	.0	.0	.0
3	5	ROD	2.495156E+03	-4.990313E+02	.0	.0	.0	.0	.0
3		*TOTALS*	.0	-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 1-34 Grid Point Force Balance



Surface or volume ID
referenced on the Case
Control SURFACE or
VOLUME command

Grid point ID where stress
averaging is performed

Referenced
coordinate
system ID

S T R E S S E S A T G R I D P O I N T S - - S U R F A C E 91										
GRID ID	ELEMENT ID	SURFACE X-AXIS X NORMAL(Z-AXIS) Z			REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID			MAX	VON MISES	
		FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR			
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
21	0	MID	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		Z1	-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	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
61	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
61	0	MID	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		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

Fiber location as
specified on the
PSHELL entry

Average component stresses at
grid points in reference coordinate
system

Major and
minor principal
stresses

Maximum
shear and von
Mises stresses

Angle of principal stress in
referenced coordinate system

Notes: 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 1-35 Grid Point Stress Output



Surface or volume ID referenced
on the Case Control SURFACE or
VOLUME command

Grid point ID where stress
discontinuities are calculated

Referenced
coordinate
system ID

GRID POINT STRESS DISCONTINUITIES -- SURFACE 91										0
GRID ID	SURFACE X-AXIS X NORMAL(Z-AXIS) Z			REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID						ERROR EST.
	FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	MAJOR	MINOR	MAX SHEAR	VON MISES		
20	Z1	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02		5.345E+02
	Z2	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02		5.345E+02
	MID	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02		5.345E+02
21	Z1	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02		5.092E+02
	Z2	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02		5.092E+02
	MID	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02		5.092E+02
61	Z1	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02		3.775E+02
	Z2	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02		3.775E+02
	MID	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02		3.775E+02
62	Z1	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02		3.596E+02
	Z2	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02		3.596E+02
	MID	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02		3.596E+02

Fiber location as
specified on the
PSHELL entry

Probable error of
component
stresses δ_g

Probable error of major principal,
minor principal, maximum shear
and von Mises stresses

Stress error
measured at
each grid point

Notes:

This output is obtained by using the STRFIELD, STRESS, GPSDCON, and SURFACE (or VOLUME) Case Control command.

Grid point stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Grid point stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

Figure 1-36 Grid Point Stress Discontinuities Output



ELEMENT STRESS DISCONTINUITIES -- SURFACE 91										
ELEMENT ID	ELEMENT TYPE	SURFACE X-AXIS X NORMAL(Z-AXIS) Z			REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0					
		NORMAL STRESS	DISCONTINUITY IN SURFACE	SHEAR-XY	PRINCIPAL STRESS	DISCONTINUITY	MAJOR	MINOR	MAX SHEAR	VON MISES
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	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+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 element

Notes:

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 1-37 Element Stress Discontinuities Output



```

1 BEAM LIBRARY TEST CASE                      BLO      MAY 29, 2005 MSC.NASTRAN 5/27/05 PAGE 10
TRANSVERSE TIP LOAD
0

S T R E S S E S   I N   B E A M   E L E M E N T S   ( S C R E E N E D )
AXIAL           SHEAR-XY          SHEAR-XZ          VON MISES
ELEMENT-ID STATION    MAX      MIN      MAX      MIN      MAX      MIN      MAX
2   0.000  2.206054E+00 -2.254727E+00  2.075935E-01 -2.175731E-02  1.687584E-01 -1.681947E-01  2.258905E+00
2   1.000  6.117611E-17 -6.315223E-17  2.075935E-01 -2.175731E-02  1.687584E-01 -1.681947E-01  3.694130E-01

```

Figure 1-38 Screened Stresses in CBAR/CBEAM Elements.

```

SUBCASE 1

S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( B E A M 3 )
ELEMENT GRID     PLY      D I R E C T   S T R E S S E S   FAILURE   MAXIMUM   STRENGTH
ID     ID      ID      NORMAL-1  NORMAL-2  NORMAL-3  SHEAR-12  SHEAR-23  SHEAR-13 THEORY FAIL. INDEX  RATIO  FLAG
2     302      2  1.277E+02  1.189E+02 -4.705E+00  1.490E+02 -1.166E+00 -7.300E+00 TSAI-WU  5.521E-03 6.005E+01
      102      2  1.242E+02  1.185E+02 -4.690E+00  1.473E+02 -1.119E+00 -7.235E+00 TSAI-WU  5.500E-03 6.059E+01
      1301     2  1.259E+02  1.187E+02 -4.698E+00  1.481E+02 -1.143E+00 -7.267E+00 TSAI-WU  5.511E-03 6.032E+01

SUBCASE 2

S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( B E A M 3 )
ELEMENT GRID     PLY      D I R E C T   S T R E S S E S   FAILURE   MAXIMUM   STRENGTH
ID     ID      ID      NORMAL-1  NORMAL-2  NORMAL-3  SHEAR-12  SHEAR-23  SHEAR-13 THEORY FAIL. INDEX  RATIO  FLAG
2     302      2  2.060E+01  1.647E+01 -5.759E-01  2.148E+01 -2.284E-01 -8.606E-01 TSAI-WU  7.399E-04 4.222E+02
      102      2  1.685E+01  1.610E+01 -7.230E-01  1.993E+01 -1.377E-01 -5.572E-01 TSAI-WU  7.258E-04 4.470E+02
      1301     2  1.812E+01  1.626E+01 -7.292E-01  2.048E+01 -1.593E-01 -5.863E-01 TSAI-WU  7.323E-04 4.376E+02

```

Figure 1-39 Stresses in Composite CBEAM3 Elements.



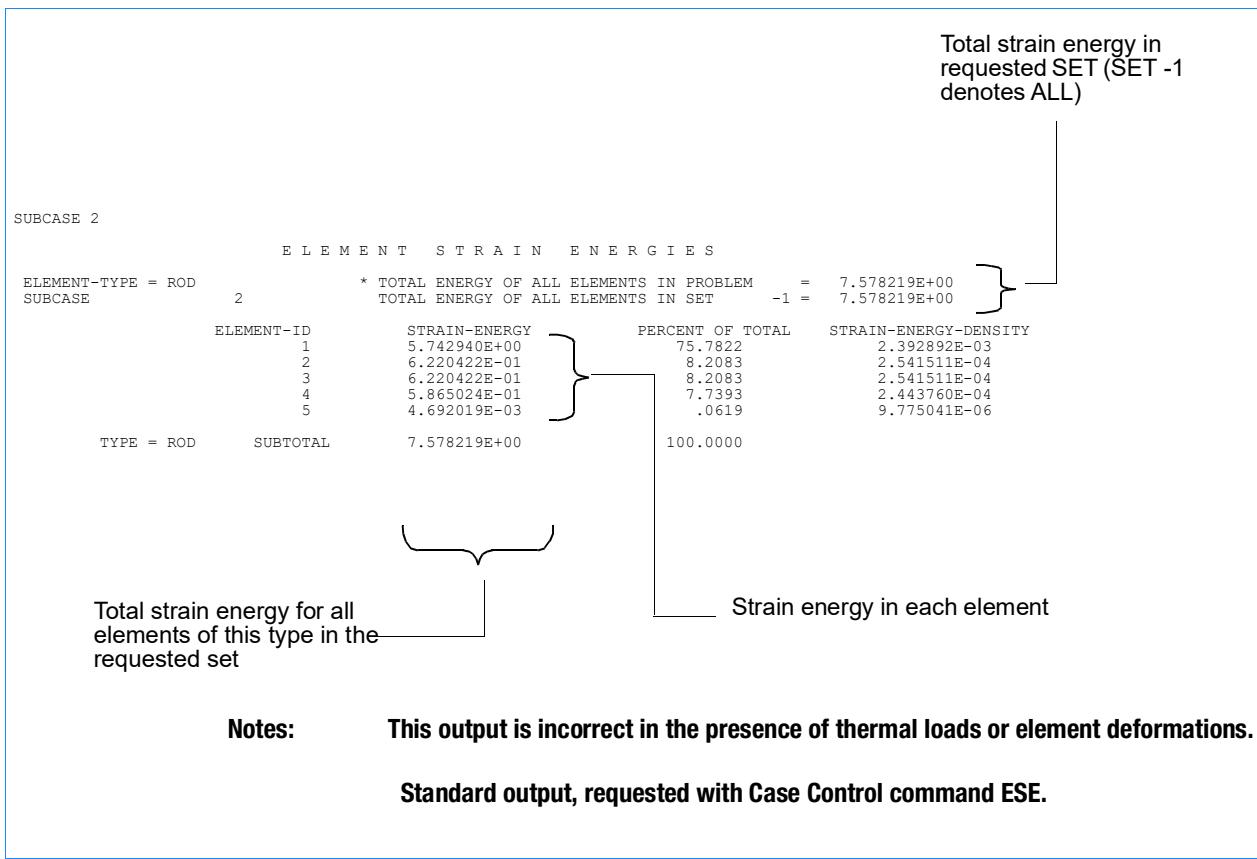


Figure 1-40 Element Strain Energy



G R I D P O I N T S I N G U L A R I T Y T A B L E							
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	UNION	NEW USET 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

G R I D P O I N T S I N G U L A R I T Y T A B L E							
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	UNION	NEW USET 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

Figure 1-41 Grid Point Singularity Table

Notes:

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 \(AUTOSPC\), 238](#) for a discussion of automatic single point constraints.



```
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
LOAD SEQ. NO.          EPSILON          EXTERNAL WORK
  1      -4.5888836E-15    6.4518387E+01
  2      -2.8991481E-15    7.5782189E+00
EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERisks
```

$$\varepsilon = \frac{\text{Residual vector error}}{\{u_l\}^T \{P_l\}}$$

where:

Work performed by the applied loads = $\frac{1}{2} P_l u_l$

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
$\{\delta P_l\}$	$= [K_{ll}] \{u_l\} - \{P_l\}$

Notes: 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 1-42 User Information Message 5293



		OLOAD			RESULTANT		
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	-----	-----	0.000000E+00	0.000000E+00	0.000000E+00
	FY	-----	0.000000E+00	-----	0.000000E+00	-----	0.000000E+00
	FZ	-----	-----	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
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.

Figure 1-43 OLOAD Resultant

Notes: Seven lines are printed for each subcase.

One line, the total, is printed for each subcase if PARAM,RESLOPT,1 is added to the bulk data file.

See [Overview of the MSC Nastran Output Files, 38](#) for an example.

This output can be removed by adding PARAM,PRTRESLT,NO in the bulk data file.

		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	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

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.



Figure 1-44 SPCFORCE Resultant

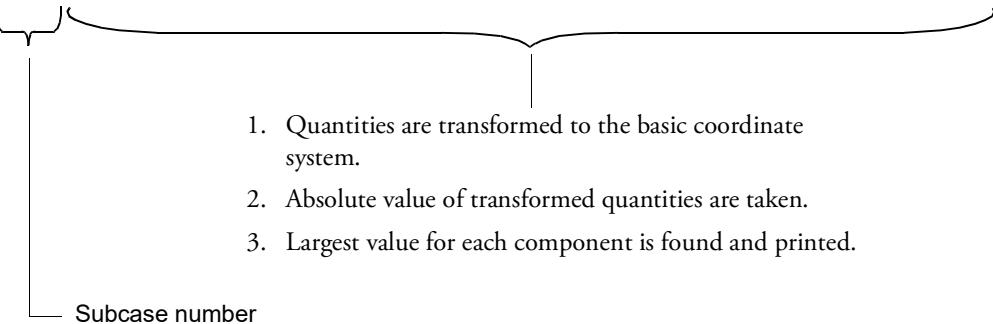
Notes: Seven lines are printed for each subcase.

One line, the total, is printed for each subcase if PARAM,RESLOPT,1 is added to the bulk data file.

See [Overview of the MSC Nastran Output Files, 38](#) for an example.

This output can be removed by adding PARAM,PRTRESLT,NO in the bulk data file.

MAXIMUM SPCFORCES						
SUBCASE/ DAREA ID	T1	T2	T3	R1	R2	R3
1000	1.1823553E+03	3.8440362E-11	1.3217409E-09	9.0155258E+00	7.1536668E-02	2.6507227E+02
2000	1.1823553E+03	3.8440362E-11	1.3217409E-09	9.0155258E+00	7.1536668E-02	2.6507227E+02
MAXIMUM DISPLACEMENTS						
SUBCASE/ DAREA ID	T1	T2	T3	R1	R2	R3
1000	4.4799998E-02	5.7587172E-03	2.0933952E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2000	4.4799998E-02	5.7587172E-03	2.0933952E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
MAXIMUM APPLIED LOADS						
SUBCASE/ DAREA ID	T1	T2	T3	R1	R2	R3
1000	1.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2000	1.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
MAXIMUM MPCFORCES						
SUBCASE/ DAREA ID	T1	T2	T3	R1	R2	R3
1000	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2000	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00



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 1-45 Maximum SPCFORCES, DISPLACEMENT, APPLIED LOADS, and MPCFORCES

```

*** 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  -1.046443E-05  -6.331082E-08  0.000000E+00  0.000000E+00  0.000000E+00
  2   G  100      -6.331082E-08  1.046443E-05  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
  51  G  BASIC     0.000000E+00  1.350188E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

*** 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    1000
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

*** 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     -1.182355E+03  0.000000E+00  0.000000E+00  9.015526E+00  0.000000E+00  2.650723E+02
  3   G  100      0.000000E+00  0.000000E+00  -1.182355E+03  2.650723E+02  0.000000E+00  9.015526E+00
  101 G  BASIC     -1.093257E+03  -5.627498E-12  -1.321741E-09  0.000000E+00  0.000000E+00  9.527259E-02

```

Figure 1-46 MAXMIN (old form) DISPLACEMENT, VELOCITY, ACCELERATION, SPCFORCE and MPCFORCE

M A X / M I N S U M M A R Y (DISPLACEMENTS)					
POINT	CID	OPERATION	COMPONENT	SUBCASE	VALUE
59	0	MAX	1	T1	1000 1.749636E-02
2	0		2		1000 0.000000E+00
51	0		3		1000 0.000000E+00
2	0	MIN	1	T1	1000 0.000000E+00
51	0		2		1000 0.000000E+00
59	0		3		1000 1.749636E-02
59	0	ABS	1	T1	1000 1.749636E-02
2	0		2		1000 0.000000E+00
51	0		3		1000 0.000000E+00
51	0	MAX	1	T2	1000 1.350188E-04
2	0		2		1000 -1.046443E-05
59	0		3		1000 -3.285893E-03
59	0	MIN	1	T2	1000 -3.285893E-03
2	0		2		1000 -1.046443E-05
51	0		3		1000 1.350188E-04
59	0	ABS	1	T2	1000 3.285893E-03
51	0		2		1000 1.350188E-04
2	0		3		1000 1.046443E-05

Note: The MAXMIN(DEF) plus MAXMIN (new form) can produce a wide variety of output. This selection was for Displacements T1 and T2 for a set of grids.

Figure 1-47 Sample of MAXMIN(DEF) Grid Point Output



MAX / MIN SUMMARY (QUAD4)						
ELEMENT	CID	OPERATION	COMPONENT	SUBCASE	VALUE	
37	-1	MAX	1	SMAX1	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	SMAX1	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03
37	-1	MAX	1	SMAX2	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	SMAX2	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03

Note: The MAXMIN(DEF) plus MAXMIN (new form) can produce a wide variety of output. This selection was for max and min CQUAD4 Stress Components smax1, smax2.

Figure 1-48 Sample of MAXMIN(DEF) Element Output

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	.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

Figure 1-49 Displacement Vectors

Note: The first three lines of each type describe the MAXMIN output options requested.

Standard DISPLACEMENT output for a static analysis, requested in Case Control Section.



See Coordinate Systems, 64 for definition of the global coordinate system.

**T1, T2 and T3; translations are in units of length of the model.
R1, R2, and R3; rotations are in units of radians.**

See Scalar Points, 63 and Coordinate Systems, 64 for an explanation of the displacement components T1, T2, T3, R1, R2, R3.

PRINT, PUNCH AND COORDINATE SYSTEMS WITH SETS										SUBCASE 1000
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	OCS ID		
2	G	-6.331082E-08	1.046443E-05	0.0	0.0	0.0	0.0	100		
51	G	0.0	1.350188E-04	0.0	0.0	0.0	0.0	0		
59	G	1.749636E-02	-3.285893E-03	0.0	0.0	0.0	0.0	0		

Output Coordinate System

Note: The CID keyword in the output request(s) need(s) only appear once in the Case Control Section. It will affect ALL grid point related output, such as DISplacement, VELOCITY, ACCEleration, OLOAD, SPCForce and MPCForce.

Figure 1-50 Displacement Vectors with DISP(CID)=all

SUBCASE 1									
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 1-51 Load Vectors



*** TOTAL MEMORY AND DISK USAGE STATISTICS ***

SPARSE SOLUTION MODULES				MAXIMUM DISK USAGE				Amount of I/O transferred between each DBset and memory
HIWATER (WORDS)	DAY TIME	SUB DMAP NAME	DMAP MODULE	HIWATER (MB)	DAY TIME	SUB DMAP NAME	DMAP 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 (BLOCKS)	BLOCKSIZE (WORDS)	USED (BLOCKS)	FILE	ALLOCATED (BLOCKS)	HIWATER (BLOCKS)	HIWATER (MB)	I/O TRANSFERRED (GB)
MASTER	5000	2048	149	MASTER	5000	149	1.164	.010
DBALL	250000	2048	9	DBALL	250000	9	.070	.000
OBJSCR	5000	2048	130	OBJSCR	5000	130	1.016	.003
SCRATCH	500100	2048	20	(MEMFILE	100	62	.484	.000
				SCRATCH	250000	1	.008	.000
				SCR300	250000	1	.008	.000
							TOTAL:	.013

*** BUFFER POOL AND SCRATCH 300 USAGE STATISTICS ***

BUFFER POOL				SCRATCH 300					
OPTION SELECTED	BLOCKS ALLOCATED	BLOCKS REUSED	BLOCKS RELEASED	OPTION SELECTED	HIWATER (BLOCKS)	DAY TIME	SUB DMAP NAME	DMAP MODULE	OPN/CLS COUNTER
GINO, EXEC	37	2729	768	2	1	9:56:40	PREFACE	0 PREFACE	0

Figure 1-52 Database Summary Table



	PROJECT_ID	ASSIGNED_INT.	VERSION_ID	CREATION_TIME
0	"BLANK"	1	1 ** ** ** ** ** **	3/1/93 10:44.42 3/1/93 10:45.37 3/1/93 10:46.5 3/1/93 10:46.38 3/1/93 10:47.2 3/1/93 10:47.25 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 1-53 Restartable Versions



Glossary



Main Index

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). If material behavior and boundary conditions have no variation in circumferential behavior then axisymmetric elements may be used. If there is a variation in the circumferential direction, a Fourier series can be used to represent this and a Harmonic analysis is performed.

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](#) 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 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 Guide](#) and the [MSC Nastran 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](#) 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

III-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 [Executing MSC Nastran](#) (p. 1) in 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. A complete listing of parameter functions is included in [Parameters](#) (p. 793) in 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, RBAR1, RJOINT, RBE1, RBE2, RROD, RTRPLT and RTRPLT1; 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 tetrahedral (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 sub-structure 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 solution sequences called cyclic symmetric solution sequences that perform this task automatically (see [Cyclic Symmetry, 557](#)).

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 Guide](#) for a complete listing of all of the UFs.

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 Guide](#) 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 Guide](#) for a complete listing of all of the UWMS.

.usrobj 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.

.usrssou File

A file created by running MSC Nastran that contains the source file of the user-written DMAPs. By default, the extension .USRSSOU 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.





References



Main Index

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 Guide*, The MacNeal-Schwendler Corporation, Los Angeles, California.,
5. *MSC/NASTRAN Application Manual*, Version 67, The MacNeal-Schwendler Corporation, Los Angeles, California, June 1991.
6. *MSC Nastran Quick Reference Guide*, The MSC Software Corporation, Newport Beach, California.
7. K. Ranger, *MSC Nastran Common Questions and Answers* , The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.
8. S. P. Timoshenko and J. M. Gere, *Theory of Elastic Stability*, Engineering Societies Monograph Series, Second Edition, 1961.
9. L. Komzsik, *MSC Nastran Numerical Methods User's Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California, 2001.
10. M. A. Reymond, *MSC Nastran DMAP Programmer's Guide*, The MacNeal-Schwendler Corporation, Santa Ana, California.
11. R. J. Roark and W. C. Young, *Formulas for Stress and Strain*, McGraw-Hill, Fifth Edition, 1975.
12. D. T. Greenwood, *Principles of Dynamics*, Prentice-Hall, 1965.
13. "A Proposed Standard Set of Problems to Test Finite Element Approach," *MSC/NASTRAN Application Notes*, 1984.
14. Grant G. Sitton, *MSC Nastran Dynamic Analysis User's Guide*, Version 69, The MacNeal-Schwendler Corporation, Los Angeles, California.
15. R. H. MacNeal (ed.), *The NASTRAN Theoretical Manual*, December 1972.
16. *MSC NASTRAN Nonlinear User's Guide (SOL 400)*, 2014, The MSC Software Corporation, Newport Beach, California.
17. R. T. Jones, *Mechanics of Composite Materials*, McGraw-Hill, 1975.
18. *Getting Started with MSC Nastran User's Guide*, The MSC Software Corporation, Newport Beach, California.
19. M. S. Chainyk, *MSC Nastran Thermal Analysis User's Guide*, The MacNeal-Schwendler Corporation, Los Angeles, California, 2001.
20. R. H. MacNeal, *Finite Elements: Their Design and Performance*.



Index

MSC Nastran Linear Static Analysis User's Guide

A

acceleration load, 277
ADUM8, 172
ADUM9, 180
Anisotropic material, 204, 205, 211, 214, 223
Antisymmetric, 380, 382
arbitrary beam cross section, 134
ASCII, 39, 549
ASET, 476
a-set, 471, 472, 475, 477, 478, 480, 481
ASET1, 476, 477, 480, 482
ASSIGN, 534, 548
Automatic restart, 535, 542
AUTOSPC, 161, 176, 238, 239
Axisymmetric, 591
Axisymmetric analysis, 204, 206, 558, 560, 563, 567, 702

B

Backup structure, 483
BAR, 402, 406
Basic coordinate system, 64, 374
BCD, 51
BEAM, 402, 406
beam cross section, 134
Beam cross section library, 105
BEGIN BULK, 36
Bending, 219
Block, 544, 546, 555
Boundary conditions, 236, 242, 558, 567
Brick element, see also CHEXA
Buckling, 22, 496, 498, 499, 501, 506, 515, 516, 518, 521, 522, 523, 525, 527
BUFSIZE, 35, 534, 544, 556

Bulk Data Entries

ACCEL, 277
ACCEL1, 277
PBRSECT, 134

Bulk Data entries, 482, 502

Bulk Data Parameters

COUPMASS, 274

Bulk Data Section, 51, 294, 398, 484, 502

Byte, 545

C

Cantilever beam, 396
Case Control Section, 35, 38, 44, 294, 313, 398, 412, 419, 421, 486, 491, 500, 587
CBAR, 19, 76, 94, 188, 259, 288, 334, 335, 344, 386, 391, 413, 419, 432, 434, 505, 507, 511
element coordinate system, 100
orientation vector, 98

CBARAO, 262

CBEAM, 76, 115, 259, 288, 334, 335, 391, 505, 507, 511

CBEND, 76, 146, 259, 505

CBUSH, 76, 88, 188

CBUSH elements, 82

CD, 62, 63, 65, 241

CD, see also Coordinate system

CELAS1, 76

CELAS2, 76

CELAS3, 76

CELAS4, 76

CELASI, 79

CEND, 35, 43

Centrifugal force, 271

CHEXA, 78, 175, 211, 214, 340, 345, 367, 372, 378, 387, 389, 408, 505

element coordinate system, 177

Circumferential stress, 341

CMASS1, 76



- CMASS2**, 76
CMASS3, 76
CMASS4, 76
CMASSi, 279
Coldstart, 533, 535
Column, 470, 489
Comments (\$), 43
Composite element, 220, 231
Compressive load, 507, 519, 520, 523
Computer graphics, 386
Computer speed, 532
Concatenate, 549
Concentrated load, 261
Concentrated masses, see also **CMASS**
CONMi, 279, 413
CONROD, 93, 288, 505
Consistent, 392, 412, 447, 471
Consistent load, 376, 378
Constraint equation, 307
Continuation identifier, 52
Continuation lines, 52
Contour plot, 365
Coordinate system, 64, 373, 566, 572
 - CORD1C**, 70
 - CORD1R**, 66, 68
 - CORD1S**, 71
 - CORD2C**, 70, 74
 - CORD2R**, 66
 - CORD2S**, 71**coordinate system**, 424
Corner, 368, 370
Corner output, 159, 343, 368, 371, 372
 - BILIN**, 343
 - CORNER**, 343
 - CUBIC**, 343
 - SGAGE**, 343**CP**, 63, 65
CP, see also **Coordinate system**
CPENTA, 78, 175, 211, 340, 367, 372, 378, 408, 505
CPU time, 555
CPYRAM, 78, 175, 408
CQUAD4, 21, 23, 37, 77, 155, 167, 254, 266, 288, 335, 336, 341, 342, 343, 345, 367, 369, 370, 371, 372, 376, 386, 392, 393, 406, 432, 434, 442, 443, 445, 446, 505
 - element coordinate system, 157**CQUAD8**, 77, 155, 166, 167, 288, 335, 336, 341, 342, 343, 345, 367, 369, 370, 371, 372, 376, 505
 - element coordinate system, 167**CQUADR**, 77, 155, 169, 337, 367, 376, 406
CRAC2D, 77, 172
CRAC3D, 78, 180
CROD, 37, 76, 90, 288, 334, 505
 - element coordinate system, 91**Cross-sectional properties**, 105
CSHEAR, 77, 169, 266, 505
CTETRA, 175, 367, 372, 408, 505
CTRIA3, 77, 155, 167, 254, 266, 288, 335, 345, 367, 369, 370, 372, 376, 408
CTRIA6, 77, 155, 166, 167, 335, 367, 369, 370, 371, 372, 376, 505
CTRIAR, 77, 155, 169, 337, 367, 376, 408
CTRIAX6, 78, 179, 206
CTUBE, 93, 288, 505
Curvatures, 163
CWELD, 190, 191
CYAX, 579, 581
Cyclic symmetry, 379, 383, 579, 587
CYJOIN, 579
Cylinder, 523
Cylindrical coordinate system, 64, 74
Cylindrical coordinate system, see also **CORD1C**
Cylindrical coordinate system, see also **CORD2C**
CYSUP, 579
CYSYM, 579

D

- Data block**, 544
Database, 533, 534, 538, 539, 541, 550, 553
DATAREC, 431
DBALL, 544, 545, 546, 549, 553
DBCLEAN, 539, 550, 551
DBset, 544, 545, 546, 549, 553
Debugging, 398
DECOMP, 415
DECOMP, see also **Decomposition**
Decomposition, 532



DEFORM, 254, 283, 292, 294, 501, 587
 Degrees of freedom, 24, 477
 Degrees of freedom, see also DOFs
 DELETE, 547
 Density, 413
 Dependent degree of freedom, 307
 Determinant, 498
 DIAG, 43
 Diagnostic tools, 386, 398
 Differential stiffness, 496, 497, 500, 505
 Dihedral symmetry, 559, 563, 575
 Direct matrix input, 470
 Disk space, 532
 Displacement, 48, 442
 Dissimilar element types, 356
 Distributed loads, 266
 DMAP, 547
 DMIG, 483, 489, 491, 494
 DOFs, 343, 356, 358, 484
 DSYM, 587, 591
 Dynamic analysis, 18

E

ECHO, 48
 BOTH, 48
 NONE, 48
 PUNCH, 51
 SORT, 48
 UNSORT, 48
 ECHOOFF, 43
 ECHOON, 43
 Edge, 368, 388
 Edge loads, 256
 EIGB, 502, 520
 Eigenvalue, 238, 499, 501, 502, 507, 521, 525
 Eigenvalue extraction method, 501
 enhanced inverse power, 501
 Givens, 501
 Householder, 501
 inverse power, 501
 Lanczos, 501
 modified Givens, 501
 modified Householder, 501

Eigenvector, 501
 EIGRL, 502, 508, 512, 520
 Elapsed, 542
 Elastic, 505
 Elastic stability, 496
 Element coordinate system, 74
 element geometry checks, 401
 Element stiffness matrix, 66
 Element strain energy, 394, 398
 ELSTRESS, 224
 ELSTRESS, see also STRESS
 ENDDATA, 36
 ENDJOB, 539
 Enforced displacements, 242
 Enhanced inverse power, 501
 Epsilon, 42, 416, 460
 EPZERO, 239
 Equilibrium, 421, 470, 496, 498
 Equivalent section, 231
 Error estimate, 441
 Estimate, 502
 Executive Control Section, 35, 42, 398
 EXPAND, 549
 External programs, 494
 External work, 42

F

Face, 389
 Failure index table, 224
 Fastener Element, 191
 File Management Section (FMS), 35, 544, 545
 Files
 dat, 38, 39
 DBALL, 38, 39, 544, 545, 546, 549, 553
 f04, 38, 39
 f06, 38, 39
 log, 38, 39
 MASTER, 39, 544, 545, 553, 554
 op2, 39
 pch, 39
 plt, 39
 Finite element, 23
 Finite element method, 18



Finite element model, 62

Flexibility, 483

FMS, 536, 545, 547, 550, 552, 553

FMS, see also File Management Section

FORCE, 48, 254, 294, 584

FORCE1, 254, 294

FORCE2, 254, 294

FORTRAN, 548

Fourier series, 568

Free, 388, 389

 edge, 388

 face, 389

f-set, 471, 472

Fundamental region, 560, 564

G

GAP, 598

GENEL, 184, 483

GEOMCHECK, 402, 404

BAR_OFF, 404

BEAM_OFF, 404

HEX_AR, 403

HEX_DETJ, 403

HEX_EPLR, 403

HEX_WARP, 403

PEN_AR, 404

PEN_DETJ, 404

PEN_EPLR, 404

PEN_WARP, 404

PYR_AR, 404

PYR_DETJ, 404

PYR_EPLR, 404

PYR_EPIA, 404

PYR_WARP, 404

Q4_IAMAX, 403

Q4_IAMIN, 403

Q4_SKEW, 403

Q4_TAPER, 403

Q4_WARP, 403

T3_IAMAX, 403

T3_SKEW, 403

TET_AR, 403

TET_DETG, 403

TET_DETJ, 403

TET_EPLR, 403

Geometric, 368

Geometry, 496, 497

GINO blocks, 554

Givens, 501

Global coordinate system, 432

Global stiffness matrix, 24, 65

GPFORCE, 325, 432

GPSP, see also Grid Point Singularity Table

GPSTRESS, 367, 374

GPWG, 412, 413, 464

GRAV, 254, 294, 295, 584, 586

Gravity loads, 271, 419

GRDSET, 238

GRID, 51, 238

Grid point, 24, 62

Grid point loads, 376

Grid Point Singularity Table, 239



Grid point stress, 367, 369, 373
 Grid point weight generator, 642
 GPWG, 642
 GROUNDCHECK, 403, 429, 430
 g-set, 471, 472
 g-size, 563
 Guyan reduction, 471, 474, 480, 481

H

HARMONICS, 587, 591
 Hencky-von Mises, 350, 367, 368
 Hencky-von Mises, see also von Mises stress
 HEX_EPIA, 403
 HEXA, 402
 Hidden line, 387
 Honeycomb, 231, 232
 Honeycomb plate, 225
 Householder, 501
 HOUTPUT, 587

I

I/O time, 555
 ID, 43
 Include, 482
 Independent degrees of freedom, 307, 320
 Inertia properties, 391
 Inertia relief, 458
 INIT, 547
 Initial strains, 283
 Input file, 32
 Input file structure, 533
 INPUTT4, 483
 Instability, 496
 Interior, 368, 371
 Inverse iteration, 501
 Inverse power, 501
 Isotropic material, 204, 223

K

K2GG, 488, 491
 K6ROT, 161
 KEEP, 539, 550

L

Lagrange rigid element, 306
 Lamina, 223
 Lanczos, 501, 502, 507, 511, 525
 Lateral buckling, 511
 Linear buckling, 496
 Linear stiffness, 496, 498
 Linearly varying forces, 259
 Linux, 534
 LOAD, 37, 242, 292, 294, 483, 584, 587
 Load paths, 19
 LOADCYH, 579, 583
 LOADCYN, 579, 583
 LOADCYT, 579, 583, 586
 LOADSET, 586
 Logical name, 544
 Log-key, 548
 l-set, 471, 472, 474
 lumped mass, 274

M

M2GG, 413, 486, 491
 Mass, 279, 483
 grid point weight generator, 642
 MASTER, 544, 545, 553, 554
 MAT1, 38, 204, 219, 413
 MAT2, 205, 220, 231
 MAT3, 206
 MAT8, 207, 231
 MAT9, 211, 214
 Material coordinate system, 74
 MATi, 413
 Matrix, 470
 Matrix operation, 470
 Matrix partitioning, 470
 maximum, 423
 MAXMIN, 424
 MAXMIN(DEF), 425
 MCID, 167, 230
 Mechanism, 414
 Megabytes, 553
 Megabytes, see also Bytes
 Membrane, 219



Membrane stiffness, 520

Memory, 502

Mesh, 345

Mesh density, 21, 340

METHOD, 501, 503

MID1, 219

MID2, 219

MID3, 219

MID4, 219

Mini-database restarts, 540

minimum, 423

Modified Givens, 501

Modified Householder, 501

MOMENT, 254, 258, 294, 584

MOMENT1, 254, 258, 294

MOMENT2, 254, 258, 294

MPC, 188, 236, 238, 243, 307, 312, 340, 477, 587, 601

MSC.Patran, 525

m-set, 244, 245, 471, 472, 474

MSGLIMIT, 403

MSGTYPE, 403

Multiple-point constraints, see also MPC

Multi-point constraint, 471

N

NASTRAN statement, 35

Negative eigenvalue, 512

Neutral axis, 115

NOKEEP, 539, 550

NONE, 48

Nonlinear, 19

Nonlinear analysis, 166

Nonlinear material, 505

Normal, 392, 416

Normal loads, 256

Normal pressure, 270

NOUTPUT, 587, 591

n-set, 471, 472

NSM, 421

O

Offset, 99

Offset beams, 416

OLOAD, 257, 418, 420, 464

OLOAD RESULTANT, 42, 464

OMID, 168, 169

OMIT, 476, 478, 582

OMIT1, 476, 478, 481, 482

One-dimensional elements, 76, 90, 335

op2, 168

Orientation vectors, 391

Orthotropic material, 204, 206, 207, 223

o-set, 471, 472, 475, 477, 478, 480

Output coordinate system, 65

OUTPUT(POST), 437

P

P2G, 486, 491

PARAM, 362, 364, 456, 459, 602

AUTOSPC, 415, 471

CDITER, 599, 602

GRDPNT, 418, 420, 456, 459

INREL, 456

K6ROT, 505

SEP1XOVR, 362

TOLRSC, 362, 364

WTMASS, 413, 644

patch-to-patch, 190, 191

PBAR, 421

PBTRL, 105

PBEAM, 118

PBEAML, 124

Cross sectional dimensions, 124

PCOMP, 206, 220, 228, 231

PCOMPG, 220

PEN_EPIA, 404

PENTA, 402

Pin flags, 315

Planar, 497

Planar frame buckling, 516

Plane of symmetry, 379

Plate, 365, 368

PLOAD, 254, 266, 294

PLOAD1, 146, 254, 259

PLOAD2, 254, 266, 294

PLOAD4, 254, 266, 294, 296, 378



PLOADX, 294
 Ply coordinate system, 231
 point-to-patch, 190
 point-to-point, 190, 191
 Postprocessor, 421, 442
 Postprocessor checks, 386
 Preprocessor checks, 386
 Preprocessors, 386, 388
 Pressure load, 378, 392, 393
 Principal mass axes, 644
 Principal stiffness, 414
 Principal stress, 158, 447
 PROD, 38, 90
 Project, 539
 Projected loads, 261, 263
 project-ID, 551
 proj-ID, 550, 551
 PS, 63
 PSHELL, 219, 220, 231
 PSOLID, 178
 PUNCH, 51
 PYRAM, 402
 Pyramid element, see also CTETRA

Q

QUAD4, 402
 QUADR, 402

R

RBAR, 188, 307, 308, 310, 340, 472
 RBAR1, 331
 RBE1, 307, 308
 RBE2, 188, 307, 308, 316, 340, 358
 RBE3, 188, 307, 308, 320, 357, 358, 520, 525
 Reaction, 421
 Reaction loads, 265, 420
 Rectangular matrices, 486
 REFC, 323
 Reference degree of freedom, 459
 reference frame, 424
 REFGRID, 323
 Reflective symmetry, 558, 559, 563, 564
 Releases, 113

REPCASE, 46
 Restart, 532, 534, 536
 KEEP, 539, 550
 NOKEEP, 539, 550
 Restartable versions, 539
 RFORCE, 254, 294, 584, 586
 RIGID, 330
 Rigid body modes, 472
 Roots, 503, 512
 Rotational, 356, 358
 Rotational symmetry, 558, 559, 563, 569
 Row, 470, 489
 RROD, 307, 308
 r-set, 459, 471, 472, 474
 RSPLINE, 307, 308, 345
 RSSCON, 358, 359, 363
 RTHRESH, 431
 RTRPLT, 307, 308
 R-type elements, 243, 340

S

sb-set, 473
 Scalar elements, 76, 79
 Scalar point, 62, 63
 SCR300, 553, 554
 SCRATCH, 545, 553, 554
 SCRATCH DBset, 547
 Sequencer, 416
 SET, 45, 62, 419, 421, 470
 a-set, 472, 475, 477, 478, 480, 481
 f-set, 471, 472
 g-set, 472
 l-set, 471, 472, 474
 m-set, 244, 245, 471, 472, 474
 n-set, 471, 472
 o-set, 472, 475, 477, 478, 480
 r-set, 459, 471, 472, 474
 sb-set, 473
 sg-set, 471
 s-set, 471, 472, 474
 Shaded plots, 387
 Shear center, 115
 Shear stress, 158



- Shell, 368
 Shell normals, 338
 Shrink, 386, 387
 Sine wave, 505, 525
 Single-point constraints, see also SPC
 Singularity, 414
 SINV, 502, 520
 SLOAD, 258, 294
 SOL, 43
 105, 505
 106, 505, 523
 400, 505
 Solid, 358, 365
 Solid elements, 378, 387
 SORT, 48
 SPC, 37, 63, 238, 242, 243, 325, 432, 472, 480, 501, 587
 SPCD, 238, 242, 294, 584
 SPCFORCE, 420
 SPCi, 477
 SPCi, see also SPC
 Spherical coordinate system, see also CORD1S
 Spherical coordinate system, see also CORD2S
 SPPOINT, 470
 Spring, 394, 395
 s-set, 471, 472, 474
 Static analysis, 18, 24
 Static condensation, 470, 471, 474
 Stiffened panel, 519
 Stiffness, 356, 358, 414, 471, 483
 Stiffness matrix, 532
 strain, 163, 164
 Strain energy, 460
 Strain energy output, 386
 Strain free, 431
 STRESS, 48, 158
 Stress contour plots, 434, 443
 Stress discontinuity, 437
 Stress error estimates, 386
 Stress gradients, 341
 Stress invariants, 436
 Stress recovery location, 104
 Sturm sequence, 502
 SUBCASE, 382, 423
 SUBCOM, 47, 48, 288, 382, 423, 588
 SUBSEQ, 48, 288, 382, 588
 SUBSEQ1, 48
 Superelement, 383, 538
 SUPORT, 460, 472, 599, 602
 SURFACE, 440, 441
 Symmetric matrices, 486
 Symmetry, 103, 236, 379, 380, 558
 System Cell
 QRMETH, 337

T

- TABLEDi, 587
 TEMP, 294, 547
 TEMP(LOAD), 292, 294, 501, 587
 TEMPD, 294
 TEMPP1, 294
 TEMPP3, 294
 TEMPRB, 288, 294
 TET_EPIA, 403
 TETRA, 402
 Thermal coefficients of expansion, 431
 Thermal loads, 287
 THETA, 167
 Three-dimensional, 204, 206
 Three-dimensional elements, 78, 340
 THRESH, 430
 TIME, 43
 Topological, 368, 369
 Tracking method, 501
 Transition, 344, 345, 350, 357
 Translational symmetry, 560, 563
 Transverse shear, 208, 219, 340, 522
 Transverse shear stiffness, 94, 116
 TRIA3, 402
 TRIAR, 402
 Two-dimensional, 204
 Two-dimensional elements, 77, 154, 335

U

- UACCEL, 466
 Units, 412
 UNIX, 534



UNSORT, 48
URACCEL, 466

V

VEC PLOT, 422
Version, 550
Version number, 534
version-ID, 550, 551
VOLUME, 440
von Mises stress, 23, 158, 446, 447
von Mises stress, see also Hencky-von Mises stress

W

Warping, 94, 133
Wedge element, see also CPENTA
Weighting factors, 321
Wireframe, 387
Words, 555

X

xdb, 168

Y

Young's modulus (E), 413

Z

Zero-dimensional elements, 335
Zipper effect, 390

