

# MSC Nastran 2022.1

SOL 400 Getting Started Guide



#### Americas

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

#### Worldwide Web

www.mscsoftware.com, 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 © 2022 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 <a href="http://www.mscsoftware.com/thirdpartysoftware">http://www.mscsoftware.com/thirdpartysoftware</a> (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 11, 2022

NA:V2022.1:Z:Z:DC-SOL400GS-PDF

#### Europe, Middle East, Africa

Am Moosfeld 13 81829 Munich, Germany Telephone: (49) 89 431 98 70 Email: europe@mscsoftware.com

#### Asia-Pacific

100 Beach Road #16-05 Shaw Tower Singapore 189702 Telephone: 65-6272-0082 Email: APAC.Contact@mscsoftware.com

# **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 at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

Document name

Note:

- 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 at: http://msc-documentation.questionpro.com.

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

SOL 400 Getting Started Guide

#### Preface

| Jsing this Manual                        | 2 |
|--|---|
| Prerequisites                            | 2 |
| Goals of this Manual                     | 2 |
| Contents of This Guide                   | 2 |
| ist of MSC Nastran Guides                | 3 |
| Jsing other Manuals                      | 4 |
| Patran Documentation                     | 5 |
| Fypographical Conventions                | 5 |
| Accessing MSC Nastran Manuals            | 6 |
| Downloading the PDF Documentation Files. | 6 |
| Navigating the PDF Files.                | 6 |
| Printing the PDF Files                   | 6 |
| Fraining and Internet Resources          | 7 |
| Fechnical Support                        | 8 |
| Visit SimCompanion                       | 8 |
| Help Us Help You                         | 8 |

# 1 Nonlinear Analysis

| Linear vs. Non-linear                | 10 |
|--------------------------------------|----|
| Nonlinear Analysis: Basics           | 10 |
| When to Consider Nonlinear Analysis? | 11 |
| Causes of Nonlinearity               | 11 |
| Nonlinear Characteristics            | 12 |
| General Classification               | 12 |
| General Recommendations              | 13 |
| Applications for Nonlinear Analysis  | 14 |
| Industry Uses                        | 14 |
| Limitations of Nonlinear Analysis    | 14 |
| Performing Nonlinear Static Analysis | 15 |



## 2 Overview of SOL 400

| What's New in SOL 400 since 2016 Release       | 19 |
|--|----|
| Introduction to SOL 400                        | 21 |
| History of SOL 400                             | 22 |
| Advantages of SOL400                           | 22 |
| Capabilities of SOL400                         | 23 |
| Converting Nastran Linear to Nastran Nonlinear | 24 |
| Analysis Types                                 | 25 |
| Analysis Procedures                            | 27 |
| SUBCASEs and STEPs                             | 27 |

# 3 Capabilities

| Nonlinear Analysis                              | 30 |
|---|----|
| Applications for Nonlinear Analysis             | 30 |
| Static Analysis                                 | 30 |
| Linear Analysis                                 | 31 |
| Linear Perturbation Analysis.                   | 32 |
| Linear Static                                   | 33 |
| Normal Modes                                    | 33 |
| Direct and Modal Linear Transient Dynamics      | 33 |
| Direct and Modal Frequency Response.            | 34 |
| Direct and Modal Complex Eigenvalue             | 34 |
| General Nonlinear Analysis                      | 34 |
| Nonlinear Static                                | 34 |
| Nonlinear Transient Dynamic                     | 35 |
| Creep   | 35 |
| Viscoelastic                                    | 35 |
| Heat Transfer Procedures                        | 35 |
| Nonlinear Transient Response Analysis           | 36 |
| Nonlinear Transient Response Analysis Interface | 36 |
| Time Step Definition                            | 37 |
| Coupled Thermal-Mechanical                      | 38 |
| Thermal Contact.                                | 38 |
| Benefits  | 39 |

# SOL 400 Files

| SOL 400 Input File | 41 |
|--------------------|----|
|--------------------|----|



4

| SOL 400 Example                                     |
|---|
| Running Existing Nonlinear Models in SOL 400        |
| Generating and Editing the Bulk Data File in Patran |
| SOL 400 Output File                                 |
| The .sts file                                       |
| Postprocessing with Patran                          |
| Setting Up a SOL 400 Job 47                         |
| Executive Control Statements                        |
| 47 Solution Type                                    |
| Specifying the Solution Type                        |
| Steps and Subcases                                  |
| Specifying Subcases                                 |
| Multi-step or Multi-subcase Analyses       48       |

# 5 Elements

| Introduction                                  | 51 |
|---|----|
| Element Classes                               | 51 |
| 0-D   | 52 |
| 1-D Elements – Not Numerically Integrated     | 53 |
| 1-D Elements that are Numerically Integrated. | 53 |
| Beam Element Considerations                   | 55 |
| Large Displacement/Large Strain               | 57 |
| Planar Continuum Elements                     | 57 |
| Axisymmetric Shell Elements                   | 63 |
| 3-D Membrane, Plate, and Shell Elements       | 64 |
| Shells  | 65 |
| Shear Panel                                   | 66 |
| 3-D Solid Shell Element                       | 67 |
| 3-D Volumetric Solid Elements                 | 68 |
| Composite Solid Elements                      | 72 |
| Interface Elements                            | 73 |
| Automatic Property Mapping                    | 77 |

# 6 Materials

| Material Model Overview                        | 84 |
|--|----|
| Material Property Definitions                  | 84 |
| SOL 400 Material Entries                       | 85 |
| Linear Elastic Behavior                        | 88 |
| Element Selection for Incompressible Materials | 89 |



| Linear Elastic Materials                  | 89  |
|---|-----|
| Isotropic Materials                       | 89  |
| Orthotropic Materials                     | 90  |
| Anisotropic Materials                     | 92  |
| Viscoelastic                              | 93  |
| Elasto-plastic Behavior                   | 97  |
| Elastoplastic Material Entries            | 99  |
| Strain Rate Dependent Yield               | 101 |
| Creep (MATVP, CREEP)                      | 103 |
| Viscoplasticity (Explicit Formulation)    | 104 |
| Creep (Implicit Formulation)              | 105 |
| ANAND Solder Creep Model                  | 106 |
| Specifying Creep Material Entries         | 106 |
| Composite (PCOMP or PCOMPG)               | 107 |
| Specifying Composite Material Entries     | 109 |
| Cohesive Zone Modeling (MCOHE)            | 113 |
| Progressive Composite Failure             | 113 |
| Micro-mechanics Material Models (MATDIGI) | 115 |

# 7 Contact

| Introduction to Contact                | 118 |
|--|-----|
| Contact Types                          | 118 |
| Touching Contact                       | 119 |
| Glued Contact                          | 119 |
| Cohesive Contact                       | 122 |
| Contact Definition Method              | 123 |
| Contact Bodies                         | 124 |
| Deformable Contact Bodies (3D/2D/1D)   | 125 |
| Creating a Deformable Body             | 126 |
| Rigid Contact Bodies (3D/2D)           | 128 |
| Control of Rigid Body                  | 130 |
| Contact in MSC Nastran                 | 131 |
| Defining Contact Interactions          | 131 |
| Contact Settings for a SOL400 Analysis | 132 |
| Node-to-segment (NTS) Contact          | 132 |
| Segment-to-Segment (STS) contact       | 133 |
| Contact Considerations                 | 134 |
| Contact Parameters                     | 135 |



## 8 Constraints

| Introduction  | 138 |
|---|-----|
| Constraints   | 138 |
| Single-Point Constraints (SPC and SPC1)                     | 138 |
| Enforced Motion Constraints (SPCD and SPCR)                 | 139 |
| Applying Constraints  | 139 |
| Single Point Constraint                                     | 139 |
| Multipoint Constraints (MPC)                                | 140 |
| Static Loads  | 141 |
| Using Patran to Apply Loads and Boundary Conditions         | 141 |
| CWELD/CFAST/CSEAM Element Enhancements                      | 142 |
| Benefits  | 143 |
| Description of Features                                     | 143 |
| Enhanced Search Algorithm                                   | 144 |
| Additional Information.                                     | 147 |
| Connector Stiffness   | 148 |
| Detailed Projection Algorithm for Best Possible Projection. | 150 |

# 9 Boundary Conditions

| Introduction  | 153 |
|---|-----|
| Zero and Enforced Displacements   | 153 |
| Enforced Motion Constraints (SPCD and SPCR)                             | 153 |
| Fixed Direction Grid Point Forces                                       | 153 |
| p-Element Loads and Constraints   | 154 |
| Thermal Loads (TEMP and TEMPD)  | 155 |
| Inertial and Dynamic Loads  | 156 |
| Gravity and Centrifugal Force   | 156 |
| Initial Stress and Initial Plastic Strain Mapping from Previous Results | 157 |

# 10 Iteration Control in Nonlinear Analysis

| Introduction  | 159 |
|---|-----|
| Nonlinear Characteristics and General Recommendations | 160 |
| Starting the Analysis                                 | 161 |
| Load Increments and Iterations                        | 163 |
| Load Incrementation and Iteration                     | 166 |
| Nonlinear Solution Procedure                          | 166 |



| Adaptive Solution Strategies.                              | 167 |
|--|-----|
| Load Increment Size  | 167 |
| Fixed Load Incrementation                                  | 168 |
| Adaptive Load Incrementation in SOL 400 (NLSTEP)           | 168 |
| NLSTEP Bulk Data Entry                                     | 168 |
| Convergence Controls                                       | 169 |
| Solution Parameters  | 170 |
| Requesting Output For a Step                               | 171 |
| Defining Subcases in Patran                                | 171 |
| Define History by Selecting Load Steps                     | 171 |
| Subcase Parameters   | 172 |
| Static Subcase Parameters                                  | 173 |
| Specifying Creep Subcase Parameters                        | 174 |
| Normal Modes Subcase Parameters                            | 175 |
| Specifying Transient Dynamic Subcase Parameters            | 176 |
| Specifying Body Approach Subcase Parameters                | 177 |
| Specifying Complex Eigenvalue Subcase Parameters in Patran | 178 |
| Specifying Frequency Response Subcase Parameters           | 179 |
| Thermal So lution Subcases                                 | 179 |

# 11 Trouble Shooting

| Overview  | 182 |
|---|-----|
| Review Fatal Error Message                            | 182 |
| Review the .sts File                                  | 183 |
| Review the .f06 File                                  | 184 |
| Review NLSTEP parameters                              | 185 |
| Review Nonlinear Iteration Diagnostics                | 185 |
| Request for More Diagnostics                          | 187 |
| Review the Intermediate Results                       | 188 |
| SOL 400 Analysis Messages                             | 188 |
| Reviewing Convergence                                 | 190 |
| Determining if Your Simulation has failed to Converge | 191 |
| Review Plot Results from Converged Increments         | 192 |
| Reviewing the MSC Analysis Manager                    | 193 |
| Tips for Starting with Nonlinear Analysis             | 194 |
| Output Messages                                       | 194 |



# Frequently Asked Questions

# Preface

Using this Manual 2 List of MSC Nastran Guides 3 Using other Manuals 4 Typographical Conventions 5 Accessing MSC Nastran Manuals 6 Training and Internet Resources 7 Technical Support 8



## Using this Manual

This guide is written for a new user wanting to use SOL 400. No prior experience with commercial finite element software is assumed, and no finite element-specific university coursework is required.

## **Prerequisites**

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

## **Goals of this Manual**

This manual provides a basic background to MSC Nastran Implicit Nonlinear (SOL 400) Analysis and describes using SOL 400 within the MSC Nastran environment. The theoretical aspects of nonlinear analysis methods, types, and techniques are included as well as thorough descriptions for nonlinear material models, properties, and loads and constraints.

The goal is to provide material relevant to this subject in such a manner that so that you understand MSC Nastran Implicit Nonlinear (SOL 400). For details, refer to the *Nonlinear SOL 400 User's Guide*. Where appropriate, Patran forms and menus are shown so that you can easily use SOL 400 from the Patran pre- and postprocessing software environment.

## **Contents of This Guide**

A brief description of what is in each chapter follows:

- 1. Chapter 1: Nonlinear Analysis: This chapter gives you an overview of nonlinear analysis. It discusses the capabilities, characteristics, recommendations, limitations and general considerations of nonlinear analysis. It also ties to give you a overall picture of the general procedure for nonlinear static analysis
- Chapter 2: Overview of SOL 400: This chapter introduces you at SOL 400—it is a set of application
  modules in the MSC Nastran system that pairs the full features of MSC Nastran with the nonlinear
  capabilities to analyze a wide variety of structural problems subjected to geometric and material
  nonlinearity, and contact. It discusses the capabilities, the input files, and the output files of SOL 00.
- 3. Chapter 3: Capabilities: This chapter describes the capabilities of SOL 400.
- 4. Chapter 4: SOL 400 Files: This chapter describes the input and output files associated with SOL 400.
- 5. Chapter 5: Elements: This chapter provides information regarding elements types and classes.
- 6. Chapter 6: Materials: This chapter discussing SOL 400 support for linear, non-linear and composite materials.
- 7. Chapter 7: Contact: This chapter describes the contact capabilities in MSC Nastran SOL 400 that may be used in solving nonlinear structural and thermal analysis problems. It also provides an introduction to contact (Tied and Touching). It describes the various contact types, bodies, and interactions. It also describes contact definition method and contact in MSC Nastran.



- 8. Chapter 8: Constraints: This chapter discusses the procedures for applying loads and constraints which can be used in SOL 400 models.
- 9. Chapter 9: Boundary Conditions: This chapter describes the types of loads and boundary conditions which can be applied using SOL 400
- 10. Chapter 10: Iteration Control in Nonlinear Analysis: This chapter provides a comprehensive FE solution for multi-physics problems such as structure analysis, thermal analysis, as well coupled analysis.
- 11. Chapter 11: Trouble Shooting: This chapter provides some trouble shooting tips that you can review, when you receive non-convergence error messages (when a job does not converge).
- 12. Appendix A: Frequently Asked Questions: This Appendix contains a list of frequently asked question regarding SOL 400 and the answers to those questions.

## List of MSC Nastran Guides

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





Rotordynamics
Implicit Nonlinear (SOL 400)
Explicit Nonlinear (SOL 700)
Aeroelastic Analysis
User Defined Services
Non Linear (SOL 600)
High Performance Computing
DEMATD

You may find any of these documents from Hexagon at:

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

#### Using other Manuals

If you are new to the MSC Nastran SOL400, we recommend that you first read the *Getting Started Manual*. After reading *Getting Started*, we recommend that you refer to the following:

- Nonlinear (SOL 400) User's Guide: This manual provides a complete background to MSC Nastran Implicit Nonlinear (SOL 400) and describes using SOL 400 within the MSC Nastran environment. The theoretical aspects of nonlinear analysis methods, types, and techniques are included as well as thorough descriptions for nonlinear material models, properties, and loads and constraints. 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.
- MSC Nastran Quick Reference Guide (QRG): The QRG contains a complete description of all the input entries for MSC Nastran. Within each section, entries are organized alphabetically so they are easy to find. Each entry provides a description, formats, examples, details on options, and general remarks. You will find the full descriptions for all SOL 400 input entries in the QRG.

This guide contains many excerpts from the *QRG* which contains complete descriptions of all the finite element input data and options available in MSC Nastran. Most of the excerpts have been edited-some extensively-to eliminate material that is not relevant to the topics covered in this book.

- The *MSC Nastran Linear Static Analysis User's Guide:* It provides support information on the basic use of MSC Nastran which can also be applied to SOL 400.
- MSC Nastran Dynamic Analysis User's Guide: It provides support information on the basic use of MSC Nastran which can also be applied to SOL 400.
- MSC Nastran Reference Guide: It provides supporting information that relates to the theory of MSC Nastran inputs, element libraries, and loads and boundary conditions.
- MSC Nastran Demonstrations Problems Manual: It provides example problem and includes description of the input, procedures, and results information that relates to the practical use of the MSC Nastran inputs, element libraries, and loads and boundary conditions.



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

Two ways of working with links are as follows:

- Use  $alt \leftarrow$  to return back to the window your curser is in.
- Open the other *linked to* documents in a new window from an Adobe Reader.
  - a. Cleadite  $\rightarrow$  Preferences  $\rightarrow$  Documents  $\rightarrow$ Open.
  - b. Cross-document links in the same window.
  - c. Uncheck the  $\Box$  checkbox.
  - d. Select OK.

#### **Patran Documentation**

Three key books from the Patran library may be of assistance in running MSC Nastran Nonlinear:

- Patran User's Guide: This introductory guide gives you the essential information you need to
  immediately begin using Patran for MSC Nastran Nonlinear projects. Understanding and using the
  information in this guide requires no prior experience with CAE or finite element analysis.
- Patran Reference Manual: This a counterpart to the MSC Nastran Reference Guide, this manual
  provides complete descriptions of basic functions in Patran, geometry modeling, finite element
  modeling, material models, element properties, loads and boundary conditions, analysis, and results.
- MSC Nastran Preference Guide: This gives specific information that relates to using Patran with MSC Nastran as the intended analysis code. All application forms and required input are tailored to MSC Nastran.

# Typographical Conventions

The section provides a brief overview of the typographical conventions used in the document to help the user better follow the MSC Nastran documentation.

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

| Courier New | Represents command-line options of MSC Nastran and results from f04/f06 files. |
|-------------|--|
|             | Example: nast20170 memorymax=16gb myjob.dat                                    |
| Quoted Text | Represents command-line options of MSC Nastran for in-line text.               |
|             | Example: memorymax=16gb  |
| Arial font  | To represent elements.   |
|             | Example: RBE3 and RSPLINE are interpolation elements and are not rigid.        |



| Red Text    | Represents items in the examples that we to emphasize.  |
|-------------|---|
|             | Example: smp=16   |
| Bold Text   | Represents items in the text that we want to emphasize. |
|             | Example: <b>dmp=4</b>                                   |
| Italic Text | Represents references to manuals/documents.             |
|             |   |

Note: Since there is no user interface in MSC Nastran, we shall use bold font to emphasize.

#### Accessing MSC Nastran Manuals

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

- MSC Nastran documentation installer
- SimCompanion
- Combined documentation

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

#### **Downloading the PDF Documentation Files**

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

#### **Navigating the PDF Files**

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, links to other guides, hyperlinks to all cross-references to chapters,

#### **Printing the PDF Files**

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

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

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



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

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

| File Edit View Document Comments Forms Tools                                |  |
|---|--|
| Ctrl+O<br>Organizer ►   |  |
| Create PDF Portfolio<br>Modify PDF Portfolio                                |  |
| Create PDE  |  |
| 🙍 Collaborate 🔸   |  |
| Save Ctri+S<br>Save As Shift+Ctri+S<br>Save as Certified Document<br>Deport | Printer Name Adobe PDF   |
| Attach to Email<br>Revert<br>Close Ctrl+W                                   | Status: Paused; 16 documents waiting<br>Type: Adobe PDF Converter  |
| Properties Ctrl+D   | Print Range  |
| Print Setup Shift+ Ctrl+P   | All  |
| Print Ctrl+P  | <ul> <li>Current view</li> <li>Current page</li> <li>Pages 1 - 300</li> <li>Subset: All pages in range  <ul> <li>Reverse pages</li> </ul> </li> <li>Page Handling</li> <li>Copies: 1  <ul> <li>Copies: 1</li> <li>Collate</li> </ul> </li> <li>Page Scaling: Shrink to Printable Area  <ul> <li>Image Scaling: Shrink to Printable Area</li> </ul> </li> </ul> |

#### **Training and Internet Resources**

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

#### 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.



http://www.mscsoftware.com/msc-training

The MSC-Training link above will point you to schedule and description of MSC Seminars. Following courses are recommended for beginning 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.

## **Technical Support**

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

#### **Visit SimCompanion**

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

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

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

#### **Help Us Help You**

Clients frequently call up the support engineers at Hexagon with enquiry regarding models that do not run correctly. Our technical support staff can help you much more efficiently and effectively if you are working with a small model, since debugging a small model is much easier, and the turnaround time to rerun a (hopefully) corrected test model is minutes rather than hours.

- For information on the latest events, products and services for all products, refer to www.mscsoftware.com.
- For technical support phone numbers and contact information, please visit: https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304



# Nonlinear Analysis

- Linear vs. Non-linear 10
- Nonlinear Analysis: Basics 10
- When to Consider Nonlinear Analysis? 11
- Causes of Nonlinearity 11
- General Classification 12
- General Recommendations 13
- Applications for Nonlinear Analysis 14
- Limitations of Nonlinear Analysis 14
- Performing Nonlinear Static Analysis 15



#### Linear vs. Non-linear

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

Nonlinear analysis occurs when the load to response relationship is not linearly proportional. In a nonlinear analysis, a nonlinear relation holds between applied forces and displacements. In case of nonlinear analysis stiffness matrix change is considered.

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

Nonlinear response could be caused by any of several characteristics of a system, like large deformations and strains, material behavior or the effect of contact or other boundary condition nonlinearities. In reality many structures exhibit combinations of these various nonlinearities. Hexagon provides solutions to help you simulate accurately and efficiently systems with any or all of the nonlinearities, with applications encompassing multiple industries.

## Nonlinear Analysis: Basics

In many structures the deflections and the stresses do not change proportionately with the loads. In these problems the structure's response depends upon its current state and the equilibrium equations reflect the fact that the stiffness of the structure is dependent on both u and P.

#### P = K(P, u)u

As the structure displaces due to loading, the stiffness changes, and as the stiffness changes the structure's response changes. As a result, nonlinear problems require incremental solution schemes that divide the problem up into steps calculating the displacement, then updating the stiffness. Each step uses the results from the previous step as a starting point. As a result the stiffness matrix must be generated and inverted many times during the analysis adding time and costs to the analysis.In addition, because the response is not proportional to the loads, each load case must be solved separately and the principle of superposition is not applicable.



### When to Consider Nonlinear Analysis?

The following table lists different types of analysis, when you must consider them.

| Table 1-                       | 1 Different Analysis Types   |
|--------------------------------|--|
| Type of Analysis               | Explanation  |
| Strength analysis              | How much load can the structure support before a global failure occurs   |
| Deflection analysis            | When deflection control is of primary importance   |
| Stability analysis             | Finding critical points (limit points or bifurcation points) closest to operational range  |
| Service configuration analysis | Finding the operational equilibrium form of ceratin slender<br>structures when the fabrication and service configurations are<br>different (e.g cables, inflatable structures and helicoids) |
| Reserve strength analysis      | Finding the load carrying capacity beyond critical points to asses safety under abnormal conditions  |
| Progressive failure analysis   | A variant of stability and strength analysis in which progressive deterioration (e.g cracking) is considered.  |
| Envelope analysis              | A combination of previous analyses in which multiple<br>parameters are varied and the strength information thus<br>obtained is condensed into failure envelopes.                             |

#### Causes of Nonlinearity

Nonlinear response can be caused by any of several characteristics of a structure:

- Large deformations and strains. (Geometric Nonlinearity)
- Material behavior (Material Nonlinearity)
- The effect of contact (Contact Nonlinearity)
- Boundary condition nonlinearities

In reality many structures exhibit combinations of these various nonlinearities. Most of the time either material behavior is not linear in the operating conditions, or geometry of the structure itself keeps it from responding linearly.

Due to cost or weight advantage of nonmetals (polymers, woods, composites etc.) over metals, nonmetals are replacing metals for variety of applications, which have nonlinear load to response characteristics, even under mild loading conditions. The structures are also optimized to make most of its strength, pushing the load level so close to the strength of the material, that it starts behaving nonlinearly. In order to accurately predict the strength of the structures in these circumstances, it is necessary to perform nonlinear analysis.



#### **Nonlinear Characteristics**

Modeling for nonlinear analysis is not exempted from the guidelines for good modeling practice pertaining to linear analysis, which are summarized as follows:

- The analyst should have some insight into the behavior of the structure to be modeled; otherwise, a simple model should be the starting point.
- Substructuring should be considered for the modularity of the model and/or synergism between projects and agencies involved. The structure represented by a substructure is always linear.
- The size of the model should be determined based on the purpose of the analysis, the trade-off between accuracy and efficiency, and the scheduled deadline.
- Prior contemplation of the geometric modeling will increase efficiency in the long run. Factors to be considered include selection of coordinate systems, symmetric considerations for simplification, and systematic numbering of nodal points and elements for easy classification of locality.
- Discretization should be based on the anticipated stress gradient, i.e., a finer mesh in the area of stress concentrations.
- Element types and the mesh size should be judiciously chosen. For example, avoid highly distorted and/or stretched elements (with high aspect ratio).
- The model should be verified prior to the analysis by some visual means, such as plots and graphic displays.

## **General Classification**

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

• Geometric nonlinearity: In the structures where stiffness is dependent on the displacement which they may undergo are termed geometrically nonlinear. If a continuum body undergoes large finite deformations, the strain-displacement relations become nonlinear. For structural mechanics problems, under large deformations, the stiffness changes with deformation thus making the problem nonlinear. (Buckling problems are also nonlinear).

Geometric nonlinearity accounts for phenomena such as:

- The stiffening of a loaded clamped plate,
- Buckling or snap-through behavior in slender structures or components.

Without taking these geometric effects into account, a computer simulation can fail to predict the real structural behavior.

Material nonlinearity: It refers to the ability for a material to exhibit a nonlinear stress-strain (constitutive) response. This is one of the most common forms of nonlinearity. Material nonlinearity is often, but not always, characterized by a gradual weakening of the structural response as an increasing force is applied, due to some form of internal decomposition.



Elasto-plastic, hyperelastic, crushing, and cracking are good examples, but this can also include temperature and time-dependent effects such as visco-elasticity or visco-plasticity (creep). For thermal problems, a temperature dependent thermal conductivity will produce nonlinear equations.

This is required to predict plastic strains in metallic parts, cracking or crushing of concrete, or extreme deformation of plastic or rubber materials. It is nonlinear constitutive relation.

 Boundary/Contacts nonlinearity: When considering either highly flexible components, or structural assemblies comprising multiple components, progressive displacement gives rise to the possibility of either self or component-to-component contact. This characterizes to a specific class of geometrically nonlinear effects known collectively as *boundary condition* or *contact* nonlinearity.

In boundary condition nonlinearity the stiffness of the structure or assembly may change considerably when two or more parts either contact or separate from initial contact. Examples include bolted connections, toothed gears, and different forms of sealing or closing mechanisms.

Problems involving contact mechanics normally include a boundary condition that depends on the deformation thereby producing a nonlinear formulation. Thermal problems involving melting or freezing (phase change) also include such nonlinear boundary conditions.

They are required to predict change in status and/or sliding friction between assembly parts. It is onconstant displacement BCs.

#### **General Recommendations**

With these points in mind, additional recommendations are imperative for nonlinear analysis:

- Identify the type of nonlinearity and localize the nonlinear region for computational efficiency. If unsure, perform a linear analysis to help understand the problem.
- Segregate the linear region by using superelements and/or linear elements if possible. Notice that the potentially nonlinear elements can be used as linear elements.
- The nonlinear region usually requires a finer mesh. Use a finer mesh if severe element distortions or stress concentrations are anticipated.
- The step/subcase structure should be utilized properly to divide the load or time history for conveniences in data recovery, and database storage control, not to mention changing constraints and loading paths.
- The load or time for the subcase with NLSTEP should then be further divided into increments, for the purpose of convergence control. Automatic adaptive load incrementation, such as is available via NLSTEP, is the recommended method.
- Many options are available in solution methods to be specified on the NLSTEP data entries. The
  defaults should typically be used on all options to gain experience before experimenting with other
  options.
- Normal rotation for the *drilling degree of freedom* of shell elements restrained by the default value of 100 on the K6ROT parameter when the geometric nonlinearity is involved. In rare cases it may be necessary to adjust his value. This can help with convergence, but may also affect the results.
- Understand the basic theory of plasticity, creep, or rubber elasticity before using these capabilities.



• The time step size for a transient response analysis should be carefully considered based on the highest natural frequency of interest because it has significant effects on the efficiency as well as the accuracy of the solution. The automated procedures used by NLSTEP is adequate for this purpose.

## **Applications for Nonlinear Analysis**

Early development of nonlinear finite element technology was mostly influenced by the nuclear and aerospace industries. In the nuclear industry, nonlinearity is mainly due to high-temperature behavior of materials. Nonlinearity in the aerospace industry are mainly geometric in nature and range from simple linear buckling to complicated post-bifurcation behavior. Nonlinear finite element techniques are now applied to problems as diverse as automotive, biomechanics, civil, manufacturing, ship building, and many more.

#### **Industry Uses**

- Aerospace and Defense: Landing gear, Wing structures, Fuselage, Seals and hoses, Sheet metal forming
- Automotive: Power train, Tire, Seals and gaskets, Exhaust systems, Brakes, Suspension, Gear contact, welding, joints and connectors
- Electronics: Soldering, Welding, Drop tests, Sealing, Switches and connectors
- Energy: Wind turbine blades, Composite blade failure, Gear trains, Packers, blow out preventers, Seals and gaskets, Pipes and casings, Weldments, Drill bits
- Heavy Equipment and Machinery: Gears, Steering yokes, Belts, Hoses, Metal forming, Hose crimping, Wire crimping, Curing, Welding, Extrusion
- Medical: Stents, Catheters, Pacemakers, Dental and knee implants, Prosthetics, Muscle and tissue, Hospital equipment like beds, wheel chairs
- Rail: Tip-over stability study, Structural components, Welding, Joints and connectors
- Shipbuilding: Structural analysis, Riveting, Bolts, Welding, Sealing

## Limitations of Nonlinear Analysis

For the analyst familiar with the use of LFEA, there are a number of consequences of nonlinear behavior that have to be recognized before embarking on a NFEA:

- The principle of superposition cannot be applied
- Results of several load cases cannot be scaled, factored and combined as is done with LFEA
- Only one load case can be handled at a time
- The loading history (i.e. sequence of application of loads) are important
- The structural response can be markedly non-proportional to the applied loading, even for simple loading states
- Careful thought needs to be given to what is an appropriate measure of the behavior.



• The initial state of stress (e.g. residual stresses from welding, temperature, or prestressing of reinforcement and cables) may be extremely important for the overall response

#### Performing Nonlinear Static Analysis

For a typical nonlinear static analysis project, perform the following steps:

1. **Build FE model**: FE model building is very important step in FE analysis, irrespective of what kind of analysis to be performed. Selection of appropriate element for certain application need to be done to care. The FEA group is provided with part surface data, which is required to be meshed with elements to get the component mesh.

When all the parts in the assembly are meshed, they are all connected together using appropriate fastening elements. In general,Quad and Hex elements should be preferred over Tria, Penta and Tetra. Important features like fillets, holes, cutouts should be captured in the model appropriately.

If there are fasteners or welds between two parallel surfaces, attempts should be made to create similar mesh on both surfaces, this will facilitate placement of weld or rigid elements normal to the surfaces without distorting the shell elements. However, many FEA codes support node independent welds which is based on tied contact concept, which allow the FEA user to place weld elements independent of the nodes in the parts to be welded.

It's recommended to mesh the complex portion of a part first then proceed towards the simple or plane areas to ensure good quality mesh in the model. Correct representation of fasteners, joints and welds are necessary in the model for correct load transfer within the structure. The stiffness and preloading should be defined for these elements as applicable for better accuracy. If load transfer is supposed to take place from one surface in a structure to the other, a contact set should be defined between them. Each FEA code has its own format to input contact parameters. A typical contact definition requires primary and secondary nodes or elements, coefficient of friction, offset (gap) distance between surfaces and contact algorithm.

- 2. Assign material properties: Material nonlinearity is defined in the FE model via this very important step. The response of the structure depends on the properties supplied to the FE model. The software manual should be referred to understand the input format of the material data card. as different software codes may have different format. If the software expects the true stress-strain data then the test stress-strain data should be converted into true data before feeding them to the FE model. Sufficient stress-strain data points should be included to capture the nonlinearity of the material. Analysts should request material suppliers to provide certified properties for the exact same material which is going to be used to build the partS.
- 3. Apply loads and boundary conditions: No matter how good the FE mesh is, the results will not be accurate if the FE model is not constrained appropriately or if the load applied is not representative of intended loading. The mesh size and node location sometimes puts constrains on how and where loads and boundary conditions are applied to the model. It would not hurt to re-mesh the FE model locally at the locations where to be applied to make sure loads and boundary conditions are represented best possible way.



If the FE analysis is being done to virtually validate any test done at lab, then it is a good practice to visit the testing facility and take important measurements on test fixtures and loading devices. These measurements will help apply loads and boundary conditions in FE model to in the same way as the part or test specimen is subjected to at the time of testing. For example, measurements of loading device would help placing the loads on certain nodes or elements. Similarly the fixture dimensions would dictate constraints locations (nodes) and its degrees of freedom.

- 4. **Specify nonlinear analysis control parameters:** The basic controlling parameters for nonlinear static analysis are initial increment, minimum and maximum increment, maximum numbers of iterations, the interval at which results file are to be output and convergence criteria for iterations (acceptable residual load).
- 5. **Run the analysis:** The FE model is now ready to be run. The analysis run command may have options to specify solver version, memory size, and number of CPUs to better control execution.
- 6. **Review and interpret results:** It is highly recommended that the analysis results should be carefully reviewed and checked for accuracy before making any conclusions based on simulations. There are many ways the FEA results can be checked, some of them are:
  - Observe for unexpected movements in the animation.
  - Compare the reaction forces against applied forces.
  - Check if stresses and strains are as per material properties supplied to the FE model
  - Check interacting surfaces in the contact set for any malfunction
  - Make quick hand calculations by simplifying the problem and compare it with the FE results.

While reporting the FE results, you should always share all the assumptions made while building the FE model. The FEA results should always be verified using engineering judgment and past results with similar FE model.

If there are unrealistically high or low results, you should review the model. It is recommended to maintain consistency in the model to increase accuracy of the results, especially when analysis is being performed to evaluate effect of changing certain parameter in the model.



MSC Nastran: SOL 400 Getting Started Manual Nonlinear Analysis



# 2

# Overview of SOL 400

- What's New in SOL 400 since 2016 Release 19
- Introduction to SOL 400 21
- History of SOL 400 22
- Advantages of SOL400 22
- Capabilities of SOL400 23
- Converting Nastran Linear to Nastran Nonlinear 24
- Analysis Types 25
- Analysis Procedures 27



## What's New in SOL 400 since 2016 Release

#### 2021.2

- New BDF entries for Co-Simulation
- Contact detection of PCOMP(G) with Z0

#### 2021.1

Accelerated separation check for node-to-segment contact

#### 2021

- Brake squeal enhancement
- Support specification of contact status in linear perturbation analysis with touching contact
- Segment-to-segment contact default settings
- SOL 400 algorithm improvements
- Element strain energy (ESE) and kinetic energy (EKE) with linear perturbation analysis
- External Superelement MONPNT1 and MONPNT3 Data Recovery
- Addition of Lossy Compression for Nastran HDF5 Matrix and NLOUT Outputs (NH5RDB)

#### 2020

- Geometric perturbation
- Nonlinear buckling analysis
- New MATVE format
- Check Input errors when using Advanced Nonlinear Elements
- Support for NLLOAD output in HDF5

#### 2019FP1

- Linear perturbation buckling analysis
- Surface contact
- Linear contact in segment-to-segment contact

#### 2019

- Model contact check
- Support for SEGANGL on BCPARA (and BCAUTOP)
- Support for N3DSUM on NLOPRM, NLDBG to provide the simplified debug output
- Output of advanced composites
- Monitor points with NLSTAT and NLTRAN

#### 2018

Nonlinear CFAST/PFAST

- External superelement capability
- Support for automatic contact generation (ACG)
- Contact model check
- Output Geometry Adjustment in Initial Stress Free Contact
- Output analytical Smooth Surface in Contact
- Allow Real Zero BIAS in BCONPRG and BCTABLE
- GPFORCE

#### 2017

- Solder creep material model
- Cohesive contact
- Output the resultant contact force and moment for each contact pair
- THRU capability for BCTABL1 entry

#### 2016

- Support advance elements in Superelements and NLRESTART
- Contact in small deformation simulations
- Segment-to-segment contact enhancements
- Beam contact
- Interference fit
- Clearance (initial gap)
- Contact separation improvement
- Support connector elements (CWELD/CFAST/CSEAM) in SOL 400
- RC Network heat transfer analysis



#### Introduction to SOL 400

SOL 400 is a set of application modules in the MSC Nastran system that pairs the full features of MSC Nastran with the nonlinear capabilities of the Marc solver to analyze a wide variety of structural problems subjected to geometric and material nonlinearity, and contact.



In MSC Nastrans advanced integrated nonlinear solution (SOL 400), all the MSC Nastran infrastructure is available. This is the recommended default solution for solving nonlinear problems. SOL 400 is a powerful, easy to use tool for simulating manufacturing processes and component designs. An extensive finite element library for building your simulation model, and a set of solution procedures for the nonlinear analysis, which can handle very large matrix equations, are available in both solution sequences of MSC Nastran implicit nonlinear.

There are two forms of this combination.

- The first form is used in SOL 400 where the algorithms of Marc are embedded completely in MSC Nastran to form a completely integrated MSC Nastran solution. We recommend this as the default solution for solving MSC Nastran nonlinear problems.
- The second method is SOL 600 where MSC Nastran preprocesses the data and calls the Marc solver. In SOL 400, all MSC Nastran infrastructure is available, while in SOL 600 only selected MSC Nastran infrastructure capabilities have been integrated.

This is why *MSC recommends that SOL 400 be the default solution method for solving nonlinear problems*. An extensive finite element library for building your simulation model, and a set of solution procedures for the nonlinear analysis, which can handle very large matrix equations, are available in both solution sequences of MSC Nastran implicit nonlinear.



#### History of SOL 400

The following figure gives an idea of the history and progression of SOL 400.



#### Advantages of SOL400

The advantages of SOL 400 are as follows:

- Allows for analysis chaining which are:
  - Automatically chains together sequences of analyses with output state of one used as input state for another
  - Model complete processes in a single simulation through analysis chaining.
- Utilizes native Nastran elements, where no translation is required.
- Follower force distributed loads.
- Temperature applied to nodes. Temperature can be applied as a load in a structural analysis. The reference temperature is user definable.
- Inertial body forces, acceleration and velocity can be applied in the global coordinate system.
- Combines static and transient into one analysis
  - · Pre-stress, transient, steady-state analysis chaining
  - Thermal-structural analysis chaining
  - Multiple, independent loadcases in one run
  - Linear perturbation
- Use general contact capability: Solid-to-solid, surface-to-surface, edge-to-edge and beam-to-beam, etc.
- Not restricted to small-strain element limitations



- Large strain elements/materials (shells, solids)
- Model large displacement/rotation rigid elements
  - Kinematic RBEi elements

#### Capabilities of SOL400

It is an advanced nonlinear solution process that combines capabilities of multiple solution sequences and software components into a common solution.

- SOL 400 solves linear and nonlinear (material, contact and/or geometric) static, heat transfer, modal (vibration), buckling, and transient dynamic structural finite element problems.
- Eigenvalue solutions are available in SOL 400 for solving linear or nonlinear modal analyses and linear buckling analysis using either Lanczos or inverse power sweep methods of iteration. Through the use of parameters you can control the convergence of the eigenvalues, and the modes to retain.
- Forces can be applied to nodes in any coordinate system.
- Constrained nodal displacements (zero displacements at specified DOF). Enforced nodal displacements (nonzero displacements at specified DOF in the nodal coordinate system). Both constrained and enforced displacements can be specified as relative or absolute.
- Defines contact between two bodies by selecting the contacting bodies and defining the contact interaction properties, where gluing and un-gluing properties are provided. Enforced motion or velocity of rigid contacts surfaces is available.
- Supports the following elements types:
  - Scalar elements
  - Beams
  - Shells
  - 2D plane strain
  - 2D plane stress
  - Axisymmetric
  - 3D solids
  - Lower-order elements
  - Higher-order elements
- RBE elements and multi-point constraint equations are supported to tie specific nodes or DOFs with each other. Special MPC entities are supported, (e.g. rigid links) which can be used to tie two nodes together or equate the motion of two degrees of freedom. Both small and large rotations are supported.
- SOL 400 supports both temperature independent and dependent, isotropic, orthotropic, and anisotropic material properties. They can be defined for elastic, elastic-plastic, hyper-elastic, hypoelastic, visco-elastic, and creep constitutive models.



- SOL 400 also supports cohesive material, gasket material, and thermo-mechanical shape memory material. Nonlinear elastic-plastic materials can be defined by specifying piecewise linear stress-strain curves, which may be temperature and / or rate dependent.
- Physical properties can be associated with SOL 400 elements such as the cross-sectional properties of the beam element, the area of the beam and rod elements, the thickness of shell, plane stress, plane strain, and membrane elements, spring parameters, and point masses among others.
- Fracture mechanics capabilities include VCCT crack propagation and cohesive zone interface and closure analysis, and a large number of failure index criterion for analyzing delamination of composite elements.
- Laminated composite solid and shell elements are supported in SOL 400 through the PCOMP, PCOMPG, and PCOMPLS entries of the materials capability. Each layer has its own material, thickness, and orientation and represents linear or nonlinear material behavior, failure index calculations are also supported. Fast integration techniques are available with the PCOMPF entry. Equivalent material models may be incorporated using PSHELL.
- Analysis jobs can consist of (possibly) complicated loading histories (such as would occur in a multistep manufacturing process). A single SOL 400 analysis (sub-case) may consist of multiple steps that specify the loading sequence.
- Geometric Nonlinearity
  - Large displacement RBEs, Bush, and Connectors
  - Large displacement beam and shell offsets
- Material Nonlinearity
  - Extensive nonlinear material library
  - Large strain elements
  - User subroutines and digimat support
- Boundary Nonlinearity
  - 3D contact: touching and glued
- Adaptive load incrementing (NLSTEP)
- Multidiscipline Analysis: chained, linear perturbation, coupled, multiple independent sub-cases

## Converting Nastran Linear to Nastran Nonlinear

Converting from Nastran linear to Nastran Nonlinear solutions 400 require less effort in modifying than recreating the model from scratch. Most of the additional effort goes in preparing the model for the challenges of a nonlinear solution. These challenges include:

- Making sure the system is well conditioned (model is fully constrained, no autospc!, not force balance only!)
- Ensuring that the model behaves as desired due to nonlinear effects. Plasticity, large displacement and contact must be defined and checked carefully even for a converged solution.
- Defining the proper (efficiency but accurate) parameters for a nonlinear solution (e.g. convergence criteria)



# Analysis Types

A large class of stress analysis problems can be solved with SOL 400. A fundamental division of stress problems is whether a static, transient dynamics, or perturbation analysis is to be performed. In a dynamic response, the inertia effects are important. SOL 400 allows complete flexibility in making this distinction, so that the same analysis may contain several static, dynamic, and pertubation phases.

An important aspect of the flexibility is the manner in which MSC Nastran SOL 400 allows you to step through the loading history to be analyzed. This is accomplished by defining the job steps for the analysis.

A basic concept in MSC Nastran SOL 400 is the division of the problem history into job steps. A clear distinction is made in MSC Nastran SOL 400 between linear analysis and nonlinear analysis procedures. Loading conditions are defined differently for the two cases, time measures are different, and the results should be interpreted differently. A step is any convenient phase of the history – a thermal transient, a creep period, a dynamic transient, etc. In its simplest form, a step is just a static analysis of a load change from one magnitude to another.

An analysis step during which the response may be nonlinear is called a general analysis step. An analysis step during which the response may only be linear is called a linear perturbation analysis step. Since MSC Nastran SOL 400 treats such linear analysis as a linear perturbation about a preloaded, predeformed state, its capability for doing linear analysis is rather more general than that of a purely linear analysis program.

| Туре                           | Description   |
|--------------------------------|---|
| Linear Static                  | Static stress analysis is used when inertia effects can be neglected. During a linear static step, the model's response is defined by the linear elastic stiffness at the base state, the state of deformation and stress at the beginning of the step.   |
| Nonlinear Static               | Nonlinear static analysis requires the solution of nonlinear equilibrium equations,<br>for which SOL 400 uses Newton's method. Many problems involve history<br>dependent response, so that the solution is usually obtained as a series of<br>increments, with iteration within each increment to obtain equilibrium. For most<br>cases, the automatic incrementation provided by SOL 400 is preferred, although<br>direct user control is also provided for those cases where you have experience with<br>a particular problem. This includes creep, viscoelastic and body approach |
| Normal Modes                   | This solution type uses eigenvalue techniques to extract the frequencies of the current system,   |
| Modal Transient<br>Dynamic     | The Duhamel Integral method integrates all of the equations of motion through time. The accuracy is based upon the number of modes extracted.   |
| Nonlinear<br>Transient Dynamic | This solution type is used when nonlinear dynamic response is being studied. For<br>most cases, the automatic incrementation provided is preferred, although direct<br>user control is also provided for those cases where you have experience with a<br>particular problem. The Generalized-alpha method has been presented as an<br>unconditionally stable (for linear systems), second-order algorithm that allows<br>user-controllable numerical dissipation.   |

Analysis types for steps/subcases in SOL 400 include the following:



| Direct Frequency<br>Response  | Frequency response analysis is a method used to compute structural response to<br>steady-state oscillatory (such as rotating machinery) excitation. In frequency<br>response analysis the excitation is explicitly defined in the frequency domain. The<br>direct method solves the coupled equations of motion in terms of forcing<br>frequency  |
|-------------------------------|---|
| Modal Frequency<br>Response   | The modal method utilizes the mode shapes of the structure to reduce and<br>uncouple the equations of motion. The solution for a particular forcing frequency<br>is obtained through the summation of the individual modal responses.   |
| Direct Complex<br>Eigenvalue  | Complex eigenvalue analysis is used to compute the damped modes of structures<br>and assess the stability of systems modeled with transfer functions. Direct<br>Complex eigenvalue analysis solves the coupled equations of motion.   |
| Modal Complex<br>Eigenvalue   | The modal method utilizes the mode shapes of the structure to reduce and uncouple the equations of motion.  |
| Steady State Heat<br>Transfer | This solution type is for heat transfer problems where the temperature field can be<br>found for the current contact and deformation states of the bodies being studied.<br>(e.g., it is assumed that there are no changes in the structure during the heat<br>transfer analysis). For cases where there are mechanical changes in the structure a<br>coupled thermal-structural solution must be performed.  |
| Transient Heat<br>Transfer    | This solution type is for transient heat transfer problems where the temperature<br>field can be found based on the current state of stress and deformation of the<br>bodies being studied (e.g., it is assumed that there are no changes in the structure<br>during the heat transfer analysis). For cases where there are mechanical changes in<br>the structure a coupled thermal-structural solution must be performed. For all<br>transient heat transfer cases, the time increments may be specified directly, or will<br>be selected automatically based on a user prescribed maximum nodal temperature<br>change in a step. Automatic time incrementation is generally recommended. |

A complete description of the available analysis types is provided in Chapter 3: Nonlinearity and Analysis Types.

In each step, choose the solution type. This defines the type of analysis to be performed during the step: dynamic stress analysis, eigenvalue buckling, transient heat transfer analysis, etc. The procedure choice may be changed from step to step in any meaningful way, so that you have great flexibility in performing analyses. Since the state of the model (stresses, strains, temperatures, etc.) is updated throughout all nonlinear analysis steps, the effects of previous history are always included in the response in each new step. Thus, for example, if natural frequency extraction is performed after a geometrically nonlinear static analysis step, the preload stiffness will be included. Superposition cannot be applied in nonlinear problems. In general, a different loading sequence (reordering of the steps) requires a complete new analysis.

In a nonlinear static analysis, you first determine the total value of loading to be applied at a particular stage of the analysis. This loading value is selected with the LOAD case control command specifying a load set ID that exists in the bulk data. In this case, the step functions as a type of landmark in the loading history. It may be an expected point or a point at which the nature of the loading changes (for example, first applying an internal pressure loading and then an axial loading on a cylinder). The steps is a major partition of the loading


history. The loading history should be divided into subcases since this provides you with more control over the solution and restart strategy.

# **Analysis Procedures**

This section describes the practical steps involved in setting up and running MSC Nastran SOL 400 jobs, including the use of Patran. The Patran UI is set up to guide you through the process of setting up the SOL 400 analysis, including

- The job information for the executive section (select the solution type and solution parameters),
- The subcases and steps (create subcases and steps and use subcase parameters to specify the required step/subcase control information)
- The job submission (use Analyse --> Entire Model -> Full Run)
- Monitoring the job while it is running and once it is done (user Monitor -> Job -> View sts to monitor and Monitor -> Job -> View f06 to debug).

It covers the Patran user interface (UI) and its capabilities for setting up, submitting, and monitoring the job. It shows the user how to specify the analysis type, but does not go into any detail about the analysis types of the individual steps being set up. See Nonlinearity and Analysis Types for the details of analysis types.

The STEP is the SOL 400 mechanism for associating loads and boundary conditions, output requests, and various other parameters to be used during part of a complete run. Each step can be designated with one of the analysis types listed below. For each **Analysis Type**, define the **Solution Parameters** and **Output Requests**; these collectively constitute the **Analysis Procedures**.

In MSC Nastran, case control commands provide the loads and constraints, and load incrementation method, and controls the program after the initial elastic analysis. Case control commands also include blocks which allow changes in the initial model specifications. Case control commands can also specify print-out and postprocessing options.

Each set of load sets must begin with a SUBCASE/STEP command and be terminated by another SUBCASE/STEP or a BEGIN BULK command. If there is only one load case, the SUBCASE/STEP entry is not required. The SUBCASE option requests that the program perform another increment or series of increments. The input format for these options is described in *MSC Nastran Quick Reference Guide*.

### **SUBCASEs and STEPs**

Each set of load sets must begin with a SUBCASE/STEP command and be terminated by another SUBCASE/STEP or a BEGIN BULK command. If there is only one load case, the SUBCASE/STEP entry is



not required.



The SUBCASE option requests that the program perform another increment or series of increments. The input format for these options is described in *MSC Nastran Quick Reference Guide*.

In SOL 400, all SUBCASEs are independent from each other. Sub-cases are independent, multidiscipline load cases. Like SOL 101, many independent subcases can be performed in one single run. Each STEP can be performed with another type of analysis, i.e. statics, transient, normal modes, and frequency response, etc.



# 3 Capabilities

- Applications for Nonlinear Analysis 30
- Nonlinear Analysis 30
- Static Analysis 30
- Linear Analysis 31
- General Nonlinear Analysis 34
- Nonlinear Transient Response Analysis 36
- Coupled Thermal-Mechanical 38
- Thermal Contact 38



### **Nonlinear Analysis**

However, we know that in many structures the deflections and the stresses do not change proportionately with the loads. In these problems the structure's response depends upon its current state and the equilibrium equations reflect the fact that the stiffness of the structure is dependent on both u and P.

P = K(P, u)u

As the structure displaces due to loading, the stiffness changes, and as the stiffness changes the structure's response changes. As a result, nonlinear problems require incremental solution schemes that divide the problem up into steps calculating the displacement, then updating the stiffness. Each step uses the results from the previous step as a starting point. As a result the stiffness matrix must be generated and inverted many times during the analysis adding time and costs to the analysis.In addition, because the response is not proportional to the loads, each load case must be solved separately and the principle of superposition is not applicable.

# **Applications for Nonlinear Analysis**

Early development of nonlinear finite element technology was mostly influenced by the nuclear and aerospace industries. In the nuclear industry, nonlinearity is mainly due to high-temperature behavior of materials. Nonlinearity in the aerospace industry are mainly geometric in nature and range from simple linear buckling to complicated post-bifurcation behavior. Nonlinear finite element techniques are now applied to problems as diverse as automotive, biomechanics, civil, manufacturing, ship building, and many more. MSC Nastran SOL 700 may be used for solving high speed events such as crash and explosions.

# Static Analysis

Static stress analysis is used when inertia effects can be neglected. The problem may still have a real time scale, for example when the material has a viscoplastic response, such as rate dependent yield. The analysis may be linear or nonlinear. Nonlinearity may arise from large displacement effects, material nonlinearity and boundary nonlinearity (such as contact and friction).

Linear static analysis involves the specification of load cases and appropriate boundary conditions. Solutions may be combined in a postprocessing mode.

Nonlinear static analysis requires the solution of nonlinear equilibrium equations, for which the program uses Full Newton-Raphson or Modified Newton-Raphson. Many problems involve history dependent response, so that the solution is usually obtained as a series of increments, with iteration within each increment to obtain equilibrium. Increments must sometimes be kept small (in the sense that rotation and strain increments must be small) to assure correct modeling of history dependent effects, but most commonly the choice of increment size is a matter of computational efficiency – if the increments are too large, more iteration will be required. Each solution method has a finite radius of convergence, which means that too large an increment can prevent any solution from being obtained because the initial state is too far away from the equilibrium state that is being sought – it is outside the radius of convergence. Thus, there is an



algorithmic restriction on the increment size. For most cases, the automatic incrementation scheme is preferred, because it will select increment sizes based on these considerations. Direct user control of increment size is also provided because there are cases when you have considerable experience with his particular problem and can therefore select a more economic approach. A complete discussion of the numerical methods used to solver nonlinear static problems is included in this chapter; to get a converged solution, see Chapter 6: Setting Up, Monitoring, and Debugging the Analysis.

Geometrically nonlinear static problems frequently involve buckling or collapse behavior, where the loaddisplacement response shows a negative stiffness, and the structure must release strain energy to remain in equilibrium. Several approaches are possible in such cases. One is to treat the buckling response dynamically, thus actually modeling the kinetic response with inertia effects included as the structure snaps. This is easily accomplished by using the restart option to terminate the static procedure and switch to a dynamic procedure when the static solution goes unstable. In some simple cases, displacement control can provide a solution, even when the conjugate load (the reaction force) is decreasing as the displacement increases. More generally, static equilibrium states during the unstable phase of the response can be found by using the modified Riks method. This method is for cases where the loading is proportional – that is, where the load magnitudes are governed by a single scalar parameter. The method obtains equilibrium solutions by controlling the path length along the load-displacement curve within each increment (rather than controlling the load or displacement increment), so that the load magnitude becomes an unknown of the system. The method can provide solutions even in cases of complex, unstable response. The Riks method cannot be used in contact, heat transfer, coupled, or enforced motion.

### **Linear Analysis**

In a linear analysis, we implicitly assume that the deflections and strains are very small and the stresses are smaller than the material yield stresses. Consequently, there is assumed to be a linear relationship between the applied loads and the response of the structure. The stiffness can be considered to remain constant (i.e., independent of the displacements and forces) and the finite element equilibrium equations

#### P = Ku

are linear where the stiffness matrix K is independent of both u, the generalized displacement vector, and

P, the generalized force vector. This linearity implies that any increase or decrease in the load will produce proportional increase or decrease in displacements, strains, and stresses. Because of the linear relationship, you only need to calculate the stiffness of the structure once. From this stiffness representation, you can find the structure's response to other applied loads by multiplying the load vectors by the decomposed stiffness matrix. Linear static problems are solved in a single step. In addition, solutions can be combined using the principle of superposition.

A linear analysis is the simplest and most cost effective type of analysis to perform. Because linear analysis is simple and inexpensive to perform and often gives satisfactory results, it is the most commonly used structural analysis. Nonlinearities due to material, geometry, or boundary conditions are not included in this type of analysis. The behavior of an isotropic, linear, elastic material can be defined by two material constants: Young's modulus, and Poisson's ratio.



In actuality, linear analysis is merely an approximation to the true behavior of a structure. In some cases the approximation is very close to the true behavior, in other cases linear analysis may provide highly inaccurate results.

Linear analysis is obtained by considering the response in the step as the linear perturbation response about the base state. The base state is the current state of the model at the end of the last general, nonlinear analysis step prior to the linear perturbation step. Thus, the concept of linear analysis in MSC Nastran SOL 400 is rather general. A simple example of the value of this generalization is the natural frequencies of a violin string under increasing tension. In this case, geometrically nonlinear analysis of the string can be done in several steps, in each of which the tension is increased. At the end of each such step, the frequencies can be extracted in a linear perturbation analysis step.

Load magnitudes (including the magnitudes of prescribed boundary conditions), during a linear perturbation analysis step, are defined as the magnitudes of the load perturbations only.

Likewise, the value of any solution variable is output as the perturbation value only – the value of the variable in the base state is not included.

During a linear perturbation analysis step, the model's response is defined by its linear elastic stiffness at the base state. Plasticity and other inelastic effects are ignored. For hyperelastic materials, the tangent elastic moduli in the base state are used. Contact conditions cannot change during a perturbation analysis – they remain as they are defined in the base state. Frictional slipping is not allowed during perturbation analyses – all points in contact are assumed to be sticking if friction is present. If geometric nonlinearity is included in the general, nonlinear analysis upon which the linear perturbation study is being based, stress stiffening or softening effects and (pressure and other follower force) load stiffness effects are included in the linear perturbation analysis. In this case, perturbation stresses and strains are defined relative to the base state configuration. The effects of temperature and field variable perturbations are ignored for materials that are dependent on temperature and field variables. However, temperature perturbations will produce perturbations of thermal strain.

Some procedures are purely linear perturbation procedures. These are:

- Linear Statics: ANALYSIS = STATICS
- Bifurcation Buckling: ANALYSIS = BUCK
- Natural Frequency: ANALYSIS = MODES
- Modal Linear Transient: ANALYSIS = MTRAN
- Modal Complex Eigenvalue: ANALYSIS = MCEIG

Modal linear transient analysis and linear static analysis are done in the time domain. The step time of linear perturbations is never accumulated into the total time. For linear static perturbations the step time always begins at zero for each new step.

#### **Linear Perturbation Analysis**

Linear perturbation analysis may be performed from time to time during a fully nonlinear analysis. This is done by continuing the nonlinear response steps between the linear perturbation steps. The linear perturbation response has no effect as the nonlinear analysis is continued. Generally, dynamic analyses may



not be interrupted to perform perturbation analyses: before performing the perturbation analysis, MSC Nastran SOL 400 requires that the structure be brought into static equilibrium.

In SOL 400, the ANALYSIS case control command may be used to define a linear perturbation analysis STEP. Pertubation analysis implies a linearized solution about a nonlinear, preloaded state. The preloaded state may be either a nonlinear static, nonlinear transient analysis, or a thermo-mechanically coupled analysis. When performing a perturbation analysis in SOL 400 with large displacement activated:

- The preload will be performed including large displacements and other material nonlinearities.
- The stress obtained will be included in the initial stress stiffness of the perturbation step.

# **Linear Static**

Static stress analysis is used when inertia effects can be neglected. The problem may still have a real time scale, for example when the material has a viscoplastic response, such as rate dependent yield. The analysis may be linear or nonlinear.

In linear static analysis, ANALYSIS = STATICS

Linear static analysis involves the specification of load cases and appropriate boundary conditions. Traditionally, linear static analysis is performed using SOL 101.

#### **Normal Modes**

This solution type uses eigenvalue techniques to extract the frequencies of the current system. The stiffness determined at the end of the previous step is used as the basis for the extraction, so that small vibrations of a preloaded structure or nonlinearly deformed structure can be modeled.

# **Direct and Modal Linear Transient Dynamics**

Depending upon the structure and the nature of the loading, two different numerical methods can be used for a transient response analysis: direct and modal.

• The direct method (DTRAN): It performs a numerical integration on the complete coupled equations of motion.

```
ANALYSIS = DTRAN
```

• The modal method (MTRAN): It utilizes the mode shapes of the structure to reduce and uncouple the equations of motion (when modal or no damping is used); the solution is then obtained through the summation of the individual modal responses.

```
ANALYSIS = MTRAN
```

It is used when the transient dynamic response of a linear system, which includes inertial effects, is being studied. Since the use of modal transient analysis is covered completely in the MSC Nastran Dynamic Analysis User's Guide, it will not be covered in this manual.



#### **Direct and Modal Frequency Response**

Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. Examples of oscillatory excitation include rotating machinery, unbalanced tires, and helicopter blades. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and/or enforced motions (displacements, velocities, or accelerations).

Two different numerical methods can be used in frequency response analysis.

- The direct method (ANALYSIS = DFREQ) solves the coupled equations of motion in terms of forcing frequency.
- The modal method (ANALYSIS = MFREQ) utilizes the mode shapes of the structure to reduce and uncouple the equations of motion.

### **Direct and Modal Complex Eigenvalue**

Complex eigenvalue analysis is used to compute the damped modes of structures and assess the stability of systems modeled with transfer functions (including servomechanisms and rotating systems).

ANALYSIS = DCEIG, MCEIG

Complex eigenvalue analysis solves for the eigenvalues and mode shapes similar to normal modes analysis except that damping is added and the eigenvalue is now complex. In addition, the mass, damping, and stiffness matrices may be unsymmetric, and they may contain complex coefficients.

# General Nonlinear Analysis

A general analysis step is one in which nonlinear effects are included (although this is not necessary – it is possible to define a problem using general analysis procedures so that the response is entirely linear). The starting condition for each general step is generally the ending condition from the last general step, with the state of the model evolving throughout the history of general, nonlinear analysis steps as it responds to the history of loading.

In a general, nonlinear analysis step the loads must be defined as total values. MSC Nastran SOL 400 always considers total time to increase throughout the general, nonlinear analysis. Each step also has its own step time, which begins at zero in each step. If the analysis procedure for the step has a physical time scale, as in a dynamic analysis, step time corresponds to that physical time. Otherwise, step time is any convenient time scale, typically 0.0-1.0, for the step. The step times of all general nonlinear analysis steps accumulate into total time.

# **Nonlinear Static**

Nonlinearity may arise from large displacement effects, material nonlinearity and boundary nonlinearity (such as contact and friction).



ANALYSIS = NLSTATICS

This behavior requires the solution by a series of increments, with iteration within each increment to obtain equilibrium. For most cases, the automatic incrementation provided by SOL 400 is preferred, although direct user control is also provided for those cases where you have experience with a particular problem.

For static analysis, which involves post buckling behavior, where the load-displacement response shows a negative stiffness, and the structure must release strain energy to remain in equilibrium, an automatic load incrementation procedure must be used.

For local buckling, a quasi-static damping procedure via the ADAPT option on the NLSTEP bulk data entry may be used. However, this option only works in conjunction with advanced nonlinear elements. For global buckling, an arc length based procedure via the ARCLN option on the NLSTEP bulk date entry may be used. This option only works for non-contact scenarios.

### **Nonlinear Transient Dynamic**

This solution type is used when the transient dynamic response, which includes inertial effects, is being studied. When nonlinear behaviors are included in the problem, the direct integration method, ANALYSIS = NLTRAN, must be used.

For most cases, the automatic load incrementation method provided by NLSTEP is preferred, although direct user control is also provided for those cases where you have experience with a particular problem. For linear transient dynamic analysis, the MTRAN perturbation method should be used.

### Creep

This analysis procedure performs a transient, static, stress/displacement analysis.

ANALYSIS = NLSTATICS

It is especially provided for the analysis of materials which are described by the MATVP material form. The time integration method is controlled by (and described under) the NLSTEP bulk data entry.

### **Viscoelastic**

This is especially provided for the time domain analysis of materials which are described by the MATVE material options.

ANALYSIS = NLSTATICS

The dissipative part of the material behavior is defined through a Prony series representation of the normalized shear and bulk relaxation moduli. The time integration method is controlled the same as a creep analysis using (and described under) the NLSTEP bulk data entry.

### **Heat Transfer Procedures**

Heat transfer problems including conduction, forced convection, and boundary radiation and convection can be solved with MSC Nastran SOL 400.



ANALYSIS = HSTAT or HTRAN

The problems can be transient or steady-state, linear or nonlinear. The heat transfer elements allow for heat storage (specific heat) and heat conduction, and also allow for forced convection caused by fluid flowing through the mesh. Heat interface elements are also provided, to model the heat transfer across the boundary layer between a solid and a fluid, or between two closely adjacent solids. Shell-type heat transfer elements are included, since so many structures are of this type. The second order elements usually give more accurate results for the same number of nodes in the mesh. Analyses that involve both thermal and mechanical solutions are referred to in SOL 400 as multi-physics solutions.

# Nonlinear Transient Response Analysis

Nonlinear analysis requires iterative solution methods, thereby making it far more computationally intensive than a corresponding linear analysis. Nonlinear transient response analysis is available in MSC Nastran in SOL 129 and 400, ANALYSIS = NLTRAN. Nonlinear problems are classified into three broad categories: geometric nonlinearity, material nonlinearity, and contact.

The primary solution operations are load and time steps, iterations with convergence tests for acceptable equilibrium error, and stiffness matrix updates. The iterative process is based on the Newton-Raphson method. The tangent matrix updates are performed automatically to improve the computational efficiency, and may be overridden at your discretion.

The adaptive method is implemented using the two-point recurrence (or one-step) formula as its foundation. The optimum time step size, which is required for accuracy and efficiency, changes continuously in the transient dynamic environment. The primary concept of automatic time step adjustment is that the proper size of the time step can be predicted based on the dominant frequency in the incremental deformation pattern at the previous time step. This concept presents a deficiency of time lag involved in the prediction process. Furthermore, changes in nonlinearity cannot be predicted from the deformation pattern at the previous time step.

# Nonlinear Transient Response Analysis Interface

#### **User Interface**

The nonlinear properties and/or effects are defined by nonlinear material data (MATS1, MATEP, and TABLES1), and PARAM,LGDISP for geometric nonlinearity. The transient effects are produced by time-dependent loading functions (TLOADi, DAREA, etc.), damping (parameters, elements and material data), and mass properties.

The unique data required for SOL 400 is supplied on the NLSTEP, an all-encompassing time/load incrementation control entry for nonlinear analysis. See Chapter 4: Solution Strategies for Nonlinear Analysis: NLSTEP Bulk Data Entry for a detailed description of the NLSTEP entry and using it for nonlinear static and transient analyses.

#### **Case Control**

Each subcase and step defines a time interval starting from the last time step of the previous subcase or subcase, subdivided into smaller time steps using the NLSTEP entry. The output time is labeled by the cumulative time, including all previous subcases. The data blocks containing solutions are generated at the



end of each subcase for storage in the database for output processing and restarts. As such, converged solutions are apt to be saved at many intermediate steps in case of divergence and more flexible control becomes possible with multiple subcases. Results from converged increments can be output to \*.op2 files using the NLOPRM case control entry.

The input loading functions may be changed for each subcase or continued by repeating the same DLOAD request. However, it is recommended that one use the same TLOAD bulk data for all the subcases in order to maintain the continuity between subcases, because TLOAD data defines the loading history as a function of cumulative time. Static loads (PLOADi, FORCEi, MOMENTi) may be associated with time-dependent functions by matching the EXCITEID on the TLOADi entries. Nonlinear forces as functions of displacements or velocities (NOLINi) may be selected and printed by the case control commands NONLINEAR and NLLOAD, respectively. Each subcase may have a different time step size, time interval, and iteration control selected by the NLSTEP request. Case control requests that may not be changed after the first subcase are SPC, MPC, DMIG, and TF.

Output requests for each subcase are processed independently. Requested output quantities for all the subcases are appended after the computational process for actual output operation. See Chapter 8 for a discussion on output requests and see the *MSC Nastran Quick Reference Guide* for a complete list of output requests.

Initial conditions (displacement or velocity) can be specified by the bulk data input, TIC, selectable by the case control command IC. If initial conditions are given, all of the nonlinear element forces and stresses must be computed to satisfy equilibrium with the prescribed initial displacements. On the other hand, initial conditions can be generated by applying static analysis for the preload using PARAM,TSTATIC in the first subcase. Then the transient analysis can be performed in the ensuing subcases. Associated with the adaptive time stepping method, the PARAM,NDAMP is used to control the stability in the ADAPT method. The NDAMP parameter represents the numerical damping (a recommended value for usual cases is 0.01), which is often required to improve the stability and convergence in contact problems.

### **Time Step Definition**

In a transient dynamic analysis, time step parameters are required for integration in time. These parameters are specified in SOL 400 through the NLSTEP entry. These can be used for the Newmark beta operator to invoke the adaptive time control. Enter parameters to specify the time step size and period of time for this set of boundary conditions.

When using the Newmark-beta operator, decide which frequencies are important to the response. The time step in this method should not exceed 10 percent of the period of the highest relevant frequency in the structure. Otherwise, large phase errors will occur. The phenomenon usually associated with too large a time step is strong oscillatory accelerations. With even larger time steps, the velocities start oscillating. With still larger steps, the displacement eventually oscillates. In nonlinear problems, numerical instability usually follows oscillation. When using adaptive dynamics, you should prescribe a maximum time step.

As in the Newmark-beta operator, the time step in Houbolt integration should not exceed 10 percent of the period of the highest frequency of interest. However, the Houbolt method not only causes phase errors, it also causes strong artificial damping. Therefore, high frequencies are damped out quickly and no obvious oscillations occur. It is, therefore, completely up to the engineer to determine whether the time step was adequate. The damping problem is alleviated to a large extent with the Single Step Houbolt operator.



In nonlinear problems, the mode shapes and frequencies are strong functions of time because of large displacement effects, so that the above guidelines can be only a coarse approximation. To obtain a more accurate estimate, repeat the analysis with a significantly different time step (1/5 to 1/10 of the original) and compare responses.

# **Coupled Thermal-Mechanical**

Traditionally, structural simulations and thermal simulations have been performed independently of one another in separate analysis codes specialized in solving each physical discipline. In large companies there are different structural and thermal specialists and departments. These groups interact with each other as the thermal analysts provide the structural analysts with temperature data for thermal distortion and thermal stress analysis. For sophisticated systems, the thermal analysis is dependent on the structural deflection, so an iterative loop is set up to capture all the effects properly. MSC Nastran has had nonlinear structural and heat transfer capabilities since its inception.

One of the keys to multi-physics simulations is to capture the structural and thermal load path changes caused by contact. MD Nastran has had structural contact capabilities since its inception. General structural contact allows mechanical meshes to come in to contact and change the load path. Glued structural contact will weld mechanical meshes together. Thermal contact is a similar concept but for thermal analysis rather than structural analysis. In addition, the multi-physics aspect of MSC Nastran allows for coupling thermal and mechanical contact in the same run. In thermal analysis, thermal contact is available for both steady state and transient.

For coupled thermal-mechanical analysis the mechanical analysis can be static or dynamic and the thermal analysis can be steady state or transient. This allows four combinations of coupled thermo mechanical analysis:

- HSTAT-NLSTAT
- HSTAT-NLTRAN
- HTRAN-NLSTAT
- HTRAN-NLTRAN,

where HSTAT, HTRAN, NLSTAT, and NLTRAN stand for steady state heat transfer, transient heat transfer, structural nonlinear statics, and structural nonlinear transient respectively.

The coupled analysis can account for plasticity and frictional heat coupling and proper updating of interface conditions when there is relative motion between bodies. The bi-directional coupling is a weakly coupled approach between thermal and mechanical passes. The available bi-directional coupling schemes are identified in the figure below and they allow simulation of thermo-mechanical effects associated with large deformation problems and frictional contact.

# **Thermal Contact**

As described in the multi-physics introduction, one of the keys to multi-physics simulations is to capture the structural and thermal load path changes caused by contact. MSC Nastran has had structural contact capabilities since its inception. General structural contact allows mechanical meshes to come in to contact and change the load path. Glued structural contact will weld mechanical meshes together.



Thermal contact is a similar concept but for thermal analysis rather than structural analysis. You will be able to analyze thermal interactions between different contact bodies for the body areas that are in contact and thermal interactions between contact bodies and the environment for the body areas that are not in contact. In addition to *contact* and *no contact* from the pure mechanical case, there is *near contact* which allows thermal interactions between bodies that are getting near to each other, but not yet in real (mechanical) contact. Thermal contact is available for both steady state and transient analysis.

#### **Benefits**

This functionality provides a user-friendly interface to defining thermal contact conditions. The user defines contact bodies with their thermal properties and contact tables defining the possible contact pairings with the thermal properties for each pairing. The program automatically identifies the body areas involved in contact and the body areas exposed to the environment and for each situation, it sets up the appropriate interface conditions for the thermal interactions.

In a coupled thermal-mechanical analysis, bodies may have relative motions, causing the contact conditions to change over time. Due to friction forces in the contact interface, heat may be generated adding an extra heat flux load in the thermal phase of the analysis. All changes in the interface conditions and resulting loads are updated automatically.









# SOL 400 Input File

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 an ASCII text file which can be created using any text editor or one of the many preprocessors that interface with MSC Nastran.

After the generation of the input file is complete, it is submitted for execution as a batch process. Once the input file has been submitted, you have no additional interaction with MSC Nastran until the job is complete.

The MSC Nastran Input File, often referred to as the Bulk Data File (.bdf), (or .dat in the MSC Nastran manuals), is made up of three distinct sections:

- Executive Control: It describes the problem or solution type and optional file management.
- Case Control: It defines the load history and output requests.
- Bulk Data: It defines a detailed model, load and constraint description.

For details, refer to the MSC Nastran Getting Started Guide or the MSC Nastran Quick Reference Guide.

#### **SOL 400 Example**

The following text illustrates a simple example of a SOL 400 input file. It includes the required Executive Control, Case Control, and Bulk Data Sections that are required for any MSC Nastran analysis.

See Install.dir/Doc/pdf\_nastran/user/implicit\_nonlinear\_examples/example\_input\_files

#### Listing 4-1 Sample Implicit Nonlinear Solution 400 Input

```
$ NASTRAN input file created by Patran 64-Bit
input file
                                                       Executive Control Section
NASTRAN SYSTEM(316)=19
SOL 400
CEND
$ Direct Text Input for Global Case Control
Data
TITLE = MSC.Nastran job
SUBCASE 1
 STEP 1
   TITLE=This is a default subcase.
   ANALYSIS = NLSTATIC
   NLSTEP = 1
                                                        Case Control Section
   BCONTACT = ALLBODY
   SPC = 2
   LOAD = 1
   DISPLACEMENT (SORT1, REAL) = ALL
   SPCFORCES (SORT1, REAL) = ALL
   STRESS (SORT1, REAL, VONMISES, BILIN) = ALL
   NLSTRESS (SORT1) =ALL
   BOUTPUT (SORT1, REAL) = ALL
$ Direct Text Input for this Subcase
```



```
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM POST 1
PARAM PRTMAXIM YES
PARAM MRNOECHO 123
BCPARA 0 NLGLUE 1
      LGDISP 1
PARAM
NLSTEP 1 1.
       GENERAL 10
                   1
                          10
                         .5
                               4 1.2
                   1.-5
                                             0
       ADAPT .01
            6
                   2.-4
                                                  Bulk Data Section
       MECH PV
                                       PFNT
                                .2
$ Elements and Element Properties for region : solids
PSOLID 1 1
                    0
PSLDN1
      1
            1
      C8 SOLID
                   T.
$ Pset: "solids" will be imported as: "psolid.1"
      1 1 1
                    2 8
                               7
                                        37
CHEXA
38
      44 43
2 1
            1 2 3 9 8
                                        38
CHEXA
39
      45 44
```

#### **Running Existing Nonlinear Models in SOL 400**

Some users may have existing models that have been developed and analyzed using MSC Nastran Nonlinear Solution Sequences 106 or 129 (or others). These models may be run through SOL 400 by changing the SOLUTION procedure input to SOL 400 with the proper Analysis, such as NLSTATIC or NLTRAN, case control command in each SUBCASE/STEP as described in the following chapters. It is advantageous to read the model into Patran and change the Solution Type on the Analysis menu. In this case, Patran then modifies the input it writes to match that required by the specific SOL 400 solution sequence being written out. This method is especially useful to activate advanced nonlinear capabilities.

#### Generating and Editing the Bulk Data File in Patran

Patran offers a MSC Nastran interface that provides a communication link between Patran and MSC Nastran. The interface is a fully integrated part of the Patran system.

#### **Generating the BDF**

Selecting MSC Nastran as the analysis code preference in Patran, activates the customization process. These customizations ensure that sufficient and appropriate data is generated for the MSC Nastran interface. Specifically, the Patran forms in these main areas are modified:

- Materials
- Element Properties
- Finite Elements/MPCs and Meshing
- Loads and Boundary Conditions



Analysis Forms

Using Patran, you can run a MSC Nastran analysis or you may generate the MSC Nastran Input File to run externally. For information on generating the MSC Nastran Input file from within Patran, see Chapter 3: Running an Analysis, Analysis Form in the *Patran Interface to MSC Nastran Preference Guide*.

#### **Editing the BDF**

Once the bulk data file has been generated, you can edit the file directly from Patran.

- 1. Click the Analysis Application button to bring up Analysis Application form.
- 2. On the Analysis form, set the Action > Object > Method combination to Analyze > Existing Deck > Full Run and click Edit Input File...

Patran finds the bulk data file with the current job name and displays the file for editing in a text editing window.

#### Input File Processing (IPF) Checking

Checking of MSC Nastran bulk data entries are done during IFP. When one of these entries has erroneous data entered, the IFP will flag the entry and issue a FATAL ERROR. One needs to examine the output (.fo6) to observe the field and continuation line where the erroneous data occurs.

SOL 400 uses the Nastran input format which are same as elements, materials, properties, loadings and formats.

| SOL 101   |  |    |
|---|--|----|
| CEND  |  |    |
| ECHO = NONE   |  |    |
| SUBCASE 1   |  |    |
| \$ Subcase name : Default                           |  |    |
| SUBTITLE=Default                                    |  |    |
| NLPARM = 1  |  |    |
| BCONTACT = ALLBODY                                  |  |    |
| SPC = 2 SOL 106                                     |  |    |
| LOAD = 1 CEND                                       |  |    |
| DISPLACEMENT(SORT] ECHO = NONE                      |  |    |
| SPCFORCES(SURI1, R SUBLASE 1                        |  |    |
| STRESS(SORT1, REAL) & Subcase name : Default        |  |    |
| BEGIN BULK SUBITILE=Default                         |  |    |
| S Direct Text Input 1 NLPARM = 1                    | SOL 400  |    |
| PARAM POST U SPC = 2                                | CEND   |    |
| PARAM PRIMAXIM YES LOAD - 1                         | , SUBCASE 1  |    |
| NLPARM 1 DISFLACEMENT(SORTI, REAL)-                 | STEP 1   |    |
| S Elements and Element SFCFORCES(SORTI, REAL) - ALL | SUBTITLE=Default   |    |
| PSHELL 8 2 SIRESS SORII, REAL, TOWNISE              | ANALYSIS = NLSIAIIC  |    |
| S Pset: "Skin-U.USin BLOIN BULK                     | BCONTACT = ATTRODY   |    |
| CQUAD4 1 8 PIPECT TEXT INPUT TOF BUIK               | SPC = 2  |    |
| COULDA 2 8 PARAM AUTOSPC NO                         | LOAD = 1   |    |
| CQUAD4 3 8 PARAM LODISP 1                           | DISPLACEMENT(PLOT, SORT1, REAL)=ALL                        |    |
| PARAM PRTMAYIM VES                                  | SPCFORCES(PLOT, SORT1, REAL)=ALL                           |    |
| NTPARM 1 10   | STRESS(PLOT, SORT1, REAL, VONMISES, BILIN)=ALL             |    |
| \$ Elements and Element Proper                      | S Direct lext input for this Subcase                       |    |
| PSHFIT 8 2 08                                       | * Direct Text Input for Bulk Data                          |    |
| \$ Pset: "Skin-0 OSin" will be                      | PARAM PRIMAXIM VES   |    |
| COUAD4 1 8 1  | PARAM LGDISP 1   |    |
| COUAD4 2 8 2  | NLSTEP 1 1.  |    |
| COUAD4 3 8 4  | GENERAL 10 1 10  |    |
| COUAD4 4 8 5  | FIXED 40   |    |
|   | A Elevent Elevent Desertion (en encient Chino Opie         |    |
|   | S Elements and Element Properties for region : Skin-U.USin |    |
|   | PSHINI 8 2 0   |    |
|   | C4 DCTN LDK  |    |
|   | \$ Pset: "Skin-0.08in" will be imported as: "pshell.8"     |    |
|   | CQUAD4 1 8 1 2 5 4   | 0. |
|   | CQUAD4 2 8 2 3 6 5   | 0. |
|   | CQUAD4 3 8 4 5 8 7   | 0. |



#### SOL 400 for Nonlinear Analysis

- Nonlinear statics and transient Analysis
- Perturbation STEPs

#### Nonlinear Iteration Strategy

- NLPARM: Nonlinear Parameters for Statics
- TSTEPNL: Nonlinear Parameters for Transient
- NLSTEP: Replaces NLPARM, TSTEPNL, NLPCI, NLADAPT

#### Geometric Nonlinear Analysis

• PARAM, LGDISP, 1: For large displacement effects

#### Material Nonlinear Analysis

• MATS1, MATEP: For elasto plastic materials.

```
SOL 400
DTAG 8
CEND
TITLE = THIS IS A DEMO INPUT EXAMPLE
SUBCASE 10
 STEP 1
      LOAD = 1
     NLPARM = 110
 STEP 2
      ANALYSIS = NLTRAN
      DLOAD = 3
      TSTEPNL = 130
BEGIN BULK
PARAM, LGDISP, 1
NLPARM 110 25
                                15
                                     Р
                     ITER
                          1
                                          NO
+
          0.05
                     -3
MAT1 1 210000.
                    0.3 7.85-9 1.2-6
MATS1 1
        PLASTIC 1000. 1 1 240.
.
```

#### SOL 400 Output File

As a part of the input, you can request which results quantities you want to be written to the output from MSC Nastran, and also what formats (MASTER/DBALL, OP2, HDF5) of the results files to use. These output requests are placed in the case control section of the input (refer to the example below).



While the DISPLACEMENT, SPCFORCES, and STRESS requests are common with most MSC Nastran solution sequences, the NLSTRESS and BOUTPUT requests are unique to SOL 400 and are required to get the output unique to nonlinear analyses such as failure indices and contact interaction status, forces and stresses. Control of these options is available through the Analysis menu job setup interface in Patran (see Chapter 8 or the *Patran MSC Nastran Preference Guide* for more information on this topic).

```
SUBCASE 1
STEP 1
TITLE=This is a default subcase.
ANALYSIS = NLSTATIC
NLSTEP = 1
BCONTACT = ALLBODY
SPC = 2
LOAD = 1
DISPLACEMENT(SORT1,REAL)=ALL
SPCFORCES(SORT1,REAL)=ALL
STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
NLSTRESS(SORT1)=ALL
BOUTPUT(SORT1,REAL)=ALL
```

MSC recommends to us either the MASTER/DBALL or new OP2 (PARAM, POST, 1) output file formats for SOL 400 analysis. The advantage of using the MASTER/DBALL format is that it contains all of the database information from the MSC Nastran run and so can be used to retrieve the attributes of the model at a later time. The disadvantages to this format is that it is platform specific (e.g. not portable across platforms) and takes up more disk space. The advantages of the new OP2 format is that it takes less disk space than the MASTER/DBALL and has limited cross-platform portability (see Chapter 2: File Management Statements, The File Management Section (FMS) of the *QRG* for more details on portability).

#### The .sts file

SOL 400 provides a status file that can be queried periodically to see how the analysis is progressing and if the job is completed. If not, how much of it was completed before the analysis was terminated. The easiest way to have a real-time monitor of your SOL 400 job is to open the .sts file with a text editor that automatically updates when the file is changed. The name of an STS file consists of the root name of the job and the extension of STS, for instance, jobname.sts.

While an STS file provides MSC Nastran users a convenient and succinct means to monitor the incremental solution process and examine the relevant information of the overall iteration procedure, a sophisticated user is encouraged to look into the F06 file for the nonlinear iteration module output, which is led by the percent sign % in each entry. The .f06 file provides all nonlinear solution information.

### Postprocessing with Patran

The results application in Patran provides the capabilities for creating, modifying, deleting, posting, unposting and manipulating results visualization plots as well as viewing the finite element model. In addition, results can be derived, interpolated, extrapolated, transformed, and averaged in a variety of ways, which can be controlled by you.



Control is provided for manipulating the color/range assignment and other attributes for plot tools, and for controlling, and creating animations of static, and transient results. Results are selected from the database, and assigned to plot tools using simple forms. Results transformations are provided to derive scalars from vectors, and tensors as well as to derive vectors from tensors. This allows for a wide variety of visualization tools to be used with all of the available results.

If the job was created within Patran where a Patran jobname has the same name as the MSC Nastran jobname, you are only required to use the Results tools and Patran will import or attach the jobname.xxx file without you having to select it. If you did not create the job in Patran, you can still import the model and results and postprocess.

#### **Results**

Like the enormous amount of data needed to define the simulation model to an analysis code, there is a large volume of data returned from the simulation analysis. And just as it is virtually impossible to construct a model with a text editor alone, it is equally difficult to read and interpret the results by hand. Hence, the use of a postprocessor with a graphical user interface such as Patran, or SimXpert is highly recommended.

The Patran **Results** application gives you control of powerful graphical capabilities to display results quantities in a variety of ways:

- Deformed structural plots
- Color banded fringe plots
- Marker plots (vectors, tensors)
- Freebody diagrams
- Graph (XY) plots
- Animations of most of these plot types.

The **Results** application treats all results quantities in a very flexible and general manner. In addition, for maximum flexibility results can be:

- Sorted
- Reported
- Filtered
- Derived
- Deleted

All of the above features help give meaningful insight into results interpretation of engineering problems that would otherwise be difficult.

The **Results** application is object oriented, providing postprocessing plots which are created, displayed, and manipulated to obtain rapid insight into the nature of results data. The imaging is intended to provide graphics performance sufficient for real time manipulation. Performance will vary depending on hardware, but consistency of functionality is maintained as much as possible across all supported display devices.

Capabilities for interactive results postprocessing also exist. Advanced visualization capabilities allow creation of many plot types which can be saved, simultaneously plotted, and interactively manipulated with results quantities reported at the click of the mouse button to better understand mechanical behavior. Once defined,



the visualization plots remain in the database for immediate access and provide the means for results manipulation and review in a consistent and easy to use manner.

For more information, see Chapter 1: Introduction to Results Postprocessing in the Patran Reference Manual.

# Setting Up a SOL 400 Job

#### **Executive Control Statements**

The SOL 400 executive control statement is as follows:

SOL 400

See Chapter 2: MSC Nastran Files, The Executive Control Section Overview for a description on some of the options.

#### **Solution Type**

MSC Nastran SOL 400 can simulate different types of structural and thermal responses. In general, an structural analysis can be either static or dynamic. Both static, and dynamic analysis may simulate linear response, or nonlinear response. SOL 400 incorporates the formulations and functionality to simulate nonlinear static and dynamic structural responses.

### **Specifying the Solution Type**

The specific procedure MSC Nastran will run is specified on the ANALYSIS case control entry. SOL 400 represents multiple types of analysis procedures including structural, thermal, or multi-physics; any of which can be specified using the case control entries (see Analysis Procedures on how to use case control to specify the analysis procedures, including multi-physics/multistep).

 Entry
 Description

 ANALYSIS
 Specifies the analysis procedure to be used for the step or subcase being set up.

#### **Steps and Subcases**

Creating multiple steps allows you to simulate complex loading histories and even mix, and match different analysis procedures as required to get an accurate and efficient solution. Each step is a collection of loads and boundary conditions that define one phase of the behavior being modeled. For SOL 400, nonlinear analysis runs the starting point of each step is the ending point of the previous step.

Creating multiple subcases allows you to stack simulations and efficiently analyze multiple stand-alone jobs in one run. Each subcase should represent a complete stand-alone loading history, while gaining the efficiency of not having to re-form any matrices that can be re-used. In SOL 400, multi-subcase runs the starting point of each subcase is an unstressed, un-deformed state that is completely independent of the previous subcase.



#### **Specifying Subcases**

Each subcase is designated with the following case control commands:

| Entry   | Description  |
|---------|--|
| SUBCASE | Delimits and identifies a subcase.   |
| STEP    | Delimits and identifies a section of a subcase, typically delineating one load step, or<br>perturbation step in the analysis (see Chapter 4: Solution Strategies for Nonlinear<br>Analysis, on multi-stepping for a more detailed explanation of how STEPs and<br>SUBCASEs work together). |

#### Multi-step or Multi-subcase Analyses

SOL 400 analysis allows for six analysis type combinations:

- Nonlinear single physics
- Nonlinear chained physics
- Nonlinear coupled physics
- Linear perturbation analysis
- Nonlinear chained analysis with mesh/time change physics, and
- Standard linear physics.

If there are multiple subcases, the linear subcases will be solved first.

The general rule for this is: The solutions of all SUBCASES are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP in the same SUBCASE. The solutions of the SUBSTEPs occur sequentially within a STEP (coupled analysis).



#### CHAPTER 4 49 SOL 400 Files









# Introduction

An element is the basic building block of finite element analysis (FEA). There are several basic types of elements. Which type of element is used for the analysis depends on two factors:

- The type of analysis that is going to be performed
- The type of object that is to be modeled

An element is a mathematical relation that defines how the degrees of freedom of a node relate to the next. It also relates how the deflections create stresses. Elements can be:

- Lines (trusses or beams)
- Areas (2-D or 3-D plates and membranes)
- Solids (bricks or tetrahedrals)

The element is used as a mechanism to integrate a physical quantity over a volume, a surface, a curve, or a point. For the simplest simulation, it provides a transfer function (or impedance) between the degrees of freedom of one grid and the degrees of freedom of another grid. This is a very general definition which may be applied to any type of physics including structural analysis, thermal analysis, acoustic analysis, and fluid dynamics among others.

The selection of the element type and the design of the finite element mesh are important to obtain an accurate solution. The design of a finite element mesh is done either fully automatically by a mesh generator available from MSC (Patran, SimXpert) or any other mesh generator available in the market.

In MSC Nastran, there are two aspects of element definition:

- 1. The definition of the location of the element, by identifying grid points comprising the element. This is often called the element topology.
- 2. The definition of the characteristics of the element.

For nonlinear analysis, the topology of the elements, is generally the same for linear analysis (SOL 101) and for nonlinear analysis (SOL 400), though there are some restrictions. Additionally, there are elements available in the implicit nonlinear procedure that are not available in the linear solution sequences. For a detailed description of MSC Nastran element technology, see the *MSC Nastran Linear Static Analysis User's Guide*, MSC Nastran Elements (Ch. 4), *MSC Nastran Reference Guide*, Structural Elements (Ch. 3) and *QRG*, Bulk Data Entries.

#### **Element Classes**

In MSC Nastran, the elements are divided into classes based upon their dimensionality and function. In structural applications such as automotive body, aerospace, and civil engineering, the use of shells, beams, and rods is prevalent. In applications such as generators, rockets, and pressure vessels, axisymmetric elements may be used advantageously; while in automotive engines, housing, etc., 3-D solid elements dominate.

MSC Nastran uses Standard and Advanced elements. Standard elements are the native Nastran elements which are popular for linear static and dynamic analyses. The elements CQUAD, CTRIA, CHEXA, CPENTA, CTETRA, CBEAM and CBAR are available in both standard and advanced element versions and



these elements provide identical solutions for all linear simulations. The standard elements are defined using conventional property cards PSHELL, PLPLANE, PSOLID, PBEAM, PBAR, etc.

Advanced elements are activated explicitly by using additional property cards (in addition to conventional property card) PSHLN1, PSHLN2, PSLDN1, PBEMN1 and PBARN1. While Nastran uses its standard elements mostly for the linear simulations and also for the limited nonlinear simulations in SOL 106 and SOL 129, it uses its advanced elements for more advanced nonlinear simulations (large strain material and contact nonlinear solutions) in SOL 400. Few advanced elements (new type which have no equivalent in standard element type) are also directly defined using only the conventional property cards (PLCOMP, PCOMPLS, PCOHE, PAXISYM, etc.).

Nastran also allows the capability of automatic activation of advanced elements when the model uses certain nonlinear materials or analysis procedures. For more information, refer to See "Automatic Property Mapping" on page 77.

### **0-D**

0-D elements are a single grid; hence, they do not really have a geometry associated with them. Because of this, no numerical integration is required.

These elements include:

CELAS1, CELAS2, CELAS3, and CELAS4 (if only one grid identified) - provides a stiffness matrix

CBUSH (if only one grid identified) - provides a stiffness, damping and mass matrix

CDAMP1, CDAMP2, CDAMP3, and CDAMP4 (if only one grid identified) – provides a damping matrix

CMASS1, CMASS2, CMASS3, and CMASS4 (if only one grid identified) - provides a mass matrix

CONM1 and CONM2 – provides a mass/inertia matrix

Note that only CBUSH allows the ability to change the stiffness due to the deformation, and hence, is more powerful for nonlinear analysis.

The property options used with these elements are

| Element | Conventional Property | Auxiliary Property for Nonlinear |
|---------|-----------------------|----------------------------------|
| CELAS1  | PELAS                 | Not Applicable                   |
| CELAS2  | Not Required          | Not Applicable                   |
| CELAS3  | PELAS                 | Not Applicable                   |
| CELAS4  | Not Required          | Not Applicable                   |
| CBUSH   | PBUSH                 | Not Required                     |
| CDAMP1  | PDAMP                 | Not Applicable                   |
| CDAMP2  | Not Required          | Not Applicable                   |
| CDAMP3  | PDAMP                 | Not Applicable                   |



| Element | Conventional Property | Auxiliary Property for Nonlinear |
|---------|-----------------------|----------------------------------|
| CDAMP4  | Not Required          | Not Applicable                   |
| CMASS1  | PMASS                 | Not Applicable                   |
| CMASS2  | Not Required          | Not Applicable                   |
| CMASS3  | PMASS                 | Not Applicable                   |
| CMASS4  | Not Required          | Not Applicable                   |
| CONM1   | Not Required          | Not Applicable                   |
| CONM2   | Not Required          | Not Applicable                   |

### 1-D Elements – Not Numerically Integrated

1-D elements that have two grid (or scalar) points, but have no geometry, do not need to be numerally integrated. They effectively represent a spring or damper between two points. These elements include:

CELAS1, CELAS2, CELAS3, and CELAS4 - provides a stiffness matrix

CBUSH – provides a stiffness, damping and mass matrix.

CDAMP1, CDAMP2, CDAMP3, and CDAMP4 – provides a damping matrix.

Note that only CBUSH allows the ability to change the stiffness due to the deformation and, hence, is more powerful for nonlinear analysis.

The property options used with these elements are:

| Element | Conventional Property | Auxiliary Property for Nonlinear              |
|---------|-----------------------|---|
| CELAS1  | PELAS                 | Not Applicable                                |
| CELAS2  | Not Required          | Not Applicable                                |
| CELAS3  | PELAS                 | Not Applicable                                |
| CELAS4  | Not Required          | Not Applicable                                |
| CBUSH   | PBUSH                 | Not Required                                  |
| CBUSH1D | PBUSH1D               | If CID=0 this element supports large rotation |
| CDAMP1  | PDAMP                 | Not Applicable                                |
| CDAMP2  | Not Required          | Not Applicable                                |
| CDAMP3  | PDAMP                 | Not Applicable                                |
| CDAMP4  | Not Required          | Not Applicable                                |

# 1-D Elements that are Numerically Integrated



Physically, these elements represent slender structures, where the behavior may be considered uniaxial and the stresses in the other directions are negligible.

These elements include:

| CROD   | provides a membrane behavior plus twist, but no bending stiffness. The CROD element<br>is a 2-node element with linear interpolation. One needs to only define the rod cross-<br>section area.  |
|--------|---|
| CBAR   | provides membrane, bending and torsion behavior. The CBAR is a 2-grid element that supports linear or nonlinear material behavior. The type of numerical integration along the length and the numerical integration across the cross section is determined on the PBARN1. |
| CBEAM  | provides general beam behavior. The CEAM is a 2-grid element that will support linear or nonlinear material behavior. The type of numerical integration along the length and the numerical integration across the cross section is determined on the PBEMN1.              |
| CBEAM3 | provides general beam behavior. The CEAM3 is a 3-grid element with quadratic interpolation along the length, but only supports linear elastic material.   |
| CBEND  | is a tube element that may also be used as a curved element. The CBEND is a 2-grid element that only supports linear elastic material.  |
|        |   |



Connectivity of Typical 1-D Element

#### **Degrees of Freedom for Rods**

Global displacement degrees of freedom:

- 1 = u displacement
- 2 = v displacement
- 3 = w displacement

#### **Output of Strains**

Uniaxial in the truss member.

#### **Output of Stresses**

Uniaxial in the truss member.



#### **Beam Element Considerations**

When beam elements are used in an engineering simulation, the following considerations need to be made:

Beam cross-section definition - this is the same for linear solutions and nonlinear solutions.

Beam section orientation – the beam cross section may be either a solid section, an open section, or a hollow closed section. The definition of the beam section cross section is the same for classical and advanced elements, but the numerical procedure is different. When using the advanced elements and closed section cross sections, there is an additional restriction in that the cross section may have only one cavity.





Acceptable Cross Section

Illegal Cross section for Nonlinear Material Behavior

A detailed description of beam cross section and beam orientations is given in the QRG.

Beam cross-section integration – this is substantially different for nonlinear analysis when the additional property option. In the case of linear elastic material behavior, the cross section of the beam is integrated to obtain the area, moments of inertia, and the torsional moment. When nonlinear material models are present, the behavior has to be integrated through the cross section.

Beam offsets - MSC Nastran has two different methods to apply Beam and Shell offsets:

- a. Using rigid elements which is the default.
- a. Large rotation method activated by the MDLPRM, OFFDEF, LROFF this is the recommended approach for a nonlinear analysis.

Beam pin codes – this is the same for linear and nonlinear solutions.

#### **Degrees of Freedom for Beams**

- $1 = u_x =$ global Cartesian x-direction displacement
- 2 = u<sub>v</sub> = global Cartesian y-direction displacement
- 3 = u<sub>z</sub> = global Cartesian z-direction displacement
- 4 =  $\phi_x$  = rotation about global x-direction
- 5 =  $\phi_v$  = rotation about global y-direction
- $6 = \phi_z$  = rotation about global z-direction



Layer Stresses for Fully Nonlinear Solid Section Beam

$$1 = \sigma_{zz}$$
$$2 = \tau_{zx}$$
$$3 = \tau_{zy}$$

The Property options used with these elements are:

|         | 1-D                |                                    |                         |      |     |                          |   |                                       |      |                                  |  |
|---------|--------------------|------------------------------------|-------------------------|------|-----|--------------------------|---|---------------------------------------|------|----------------------------------|--|
| Element | Number<br>of Grids | Number of<br>Integration<br>Points | Interpolation<br>Scheme | BEH  | INT | Conventional<br>Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Sect | Permits<br>Nonlinear<br>Material |  |
| CROD    | 2                  | 1                                  | L                       | ROD  | L   | PROD                     | PRODN1<br>*                               | Yes/Yes                               | S    | Yes                              |  |
| CBAR    | 2                  | 3                                  | LC                      | PROD | LC  | PBAR(L)                  | PBARN1                                    | Yes/No                                | S    | No                               |  |
| CBAR    | 2                  | 3                                  | LC                      | PROD | LC  | PBAR(L)                  | PBARN1                                    | Yes/No                                | Ν    | Yes                              |  |
| CBAR    | 2                  | 1                                  | LC                      | PROD | LS  | PBAR(L)                  | PBARN1                                    | Yes/No                                | S    | No                               |  |
| CBAR    | 2                  | 1                                  | LC                      | PROD | LS  | PBAR(L)                  | PBARN1                                    | Yes/No                                | Ν    | Yes                              |  |
| CBEAM   | 2                  | 3                                  | LC                      | BEAM | LC  | PBEAM(L)                 | PBEMN1                                    | Yes/No                                | S    | No                               |  |
| CBEAM   | 2                  | 3                                  | LC                      | BEAM | LC  | PBEAM(L)                 | PBEMN1                                    | Yes/No                                | Ν    | Yes                              |  |
| CBEAM   | 2                  | 1                                  | LC                      | BEAM | LS  | PBEAM(L)                 | PBEMN1                                    | Yes/No                                | S    | No                               |  |
| CBEAM   | 2                  | 1                                  | LC                      | BEAM | LS  | PBEAM(L)                 | PBEMN1                                    | Yes/No                                | Ν    | Yes                              |  |
| CBEAM   | 2                  | 2                                  | LC                      | BEAM | LCC | PBEAML                   | PBEMN1                                    | Yes/No                                | Ν    | Yes                              |  |
| CBEAM   | 2                  | 2                                  | LC                      | BEAM | LCO | PBEAML                   | PBEMN1                                    | Yes/No                                | Ν    | Yes                              |  |

\* When a PRODN1 is used with a CROD to permit nonlinear material behavior, the element behavior changes such that it no longer supports torsion.

These elements cannot be used with the Hill, Barlat, Linear Mohr-Coulomb, Parabolic Mohr-Coulomb, or the IMPLICIT CREEP model specified on the MATEP option.

The user can change the Integration along the length when using PBARN1 or PBEMN1. The choices are:

| INT Code | Integration Type            |
|----------|-----------------------------|
| LC       | Linear/Cubic                |
| LCC      | Linear/Cubic Closed Section |
| LCO      | Linear/Cubic Open Section   |
| LS       | Linear-shear                |



Linear/cubic means that linear interpolation of the displacement is used along the axis and cubic displacement variation normal to the beam axis. This results in linear variation of curvature.

The CBAR, PBAR, and PBARN1 does not support LCO which requires element warping.

The cross-section behavior can either be specified by entering an S or N on the SECT option, where N means numerically integrated and S means smeared. When a nonlinear material behavior needs to be captured, you should use the N option.

# Large Displacement/Large Strain

These 1-D elements support large displacement and large rotation. The beam elements do not support large strain in the sense that the cross-section geometry, whether solid section, open section, or closed section, does not change. The CROD element does support large strain, but it is assumed to be incompressible such that the volume remains constant, hence:

 $A^*L = A_0 L_0 / L$ 

The resultant quantities are always given with respect to an element axis attached to the 1-D element. When large displacement is included, this is co-rotated with the element.

# **Planar Continuum Elements**

This is a group of elements that include plane stress, plane strain, and axisymmetric elements. The advanced elements include lower-order and higher-order triangular and quadrilateral elements with a variety of integration schemes that have the following geometry as shown in the following figure. For all these elements, the output is given with respect to the basic system.



3-node triangular element, 1-point integration point



6-node element, 3-point integration point





#### 2-D Planar Plane Stress Elements

MSC Nastran has a set of elements that can be used for 2-D plane stress simulations. Plane stress can be characterized by no variation through the thickness, and zero stress through the thickness. The coordinates must be aligned with one of the planes of the basic coordinate system. Besides the topology, one needs to define the thickness; the default is 1.0.

These elements are summarized.

Set BEH=PSTRS on  $\ensuremath{\mathsf{PSHLN2}}$ 



| Plane Stress |                    |  |                          |       |      |                           |   |                                       |                                       |  |
|--------------|--------------------|--|--------------------------|-------|------|---------------------------|---|---------------------------------------|---------------------------------------|--|
| Element      | Number<br>of Grids | Number<br>of<br>Integratio<br>n Points | Interpolatio<br>n Scheme | BEH   | INT  | Conventiona<br>I Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinea<br>r<br>Material* |  |
| CTRIA3       | 3                  | 1                                      | L                        | PSTRS | L    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |
| CQUAD4       | 4                  | 4                                      | L                        | PSTRS | L    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |
| CQUAD4       | 4                  | 1                                      | L                        | PSTRS | LRIH | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |
| CTRIA6       | 6                  | 7                                      | Q                        | PSTRS | Q    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |
| CQUAD8       | 8                  | 9                                      | Q                        | PSTRS | Q    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |
| CQUAD8       | 8                  | 4                                      | Q                        | PSTRS | QRI  | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                                   |  |

\* With exceptions

| INT Code | Integration Type              |
|----------|-------------------------------|
| L        | Linear                        |
| LRIH     | Linear Reduced Integration    |
| Q        | Quadratic                     |
| QRI      | Quadratic Reduced Integration |
| LT       | Linear with Twist             |

#### **Output of Strains**

 $1 = \varepsilon_{xx}$  $2 = \varepsilon_{yy}$  $3 = \varepsilon_{zz}$  $4 = \gamma_{xy}$ 

#### **Output of Stresses**

 $1 = \sigma_{xx}$   $2 = \sigma_{yy}$   $3 = \sigma_{zz} = 0$  $4 = \tau_{xy}$ 

These elements cannot be used with the IMPLICIT CREEP model specified on the MATEP option.



#### 2-D Plane Strain Elements

MSC Nastran has a set of elements that can be used for 2-D plane strain simulations. Plane strain can be characterized by no variation through the thickness, and zero strain through the thickness. The coordinates must be aligned with one of the planes of the basic coordinate system. Besides the topology, one needs to define the thickness; the default is 1.0.

These elements are summarized.

Set BEH=PLSTRN on PSHLN2.

| Plane Strain |                    |  |                          |        |      |                           |   |                                       |                                  |
|--------------|--------------------|--|--------------------------|--------|------|---------------------------|---|---------------------------------------|----------------------------------|
| Element      | Number<br>of Grids | Number<br>of<br>Integratio<br>n Points | Interpolatio<br>n Scheme | BEH    | INT  | Convention<br>al Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |
| CTRIA3       | 3                  | 1                                      | L                        | PLSTRN | L    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CTRIA3       | 4                  | 3                                      | L & Cubic<br>Bubble      | IPS    | L    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUAD4       | 4                  | 4                                      | L                        | PLSTRN | L    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUAD4       | 4                  | 1                                      | L                        | PLSTRN | LRIH | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CTRIA6       | 6                  | 7                                      | Q                        | PLSTRN | Q    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUAD8       | 8                  | 9                                      | Q                        | PLSTRN | Q    | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUAD8       | 8                  | 4                                      | Q                        | PLSTRN | QRI  | PLPLANE                   | PSHLN2                                    | Yes/Yes                               | Yes                              |

#### **Output of Strains**

 $1 = \varepsilon_{xx}$   $2 = \varepsilon_{yy}$   $3 = \varepsilon_{zz} = 0$  $4 = \gamma_{xy}$ 

#### **Output of Stresses**

 $1 = \sigma_{xx}$   $2 = \sigma_{yy}$   $3 = \sigma_{zz}$  $4 = \tau_{xy}$ 



Caution: The conventional CTRIA element with BEH=PLSTRN is known to give very poor results when used with incompressible or nearly incompressible behavior (including rubber materials, elastic-plastic or creep), the BEH=IPS should be used.

#### 2-D Axisymmetric Elements

MSC Nastran has a set of elements that can be used for 2-D axisymmetric simulations. Axisymmetric can be characterized as having no variation in the circumferential direction. The coordinates must be aligned with the basic X-Y system; which is interpreted as the R-Z system. Nonlinear analysis does not support superposition, so the use of Harmonic (Fourier) analysis to describe a load variation in the circumferential direction is not supported. Do not use the PAXSYMH property option. These elements are summarized.

| Axisymmetric |                    |                                    |                         |          |      |                        |   |                                       |                                  |
|--------------|--------------------|------------------------------------|-------------------------|----------|------|------------------------|---|---------------------------------------|----------------------------------|
| Element      | Number<br>of Grids | Number of<br>Integration<br>Points | Interpolation<br>Scheme | вен      | INT  | Convention al Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |
| CTRIAX       | 3                  | 1                                  | L                       | AXISOLID | L    | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CTRIAX       | 4                  | 4                                  | L & Cubic<br>Bubble     | IAX      | L    | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUADX       | 4                  | 4                                  | L                       | AXISOLID | L    | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUADX       | 4                  | 1                                  | L                       | AXISOLID | LRIH | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CTRIAX       | 6                  | 7                                  | Q                       | AXISOLID | Q    | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUADX       | 8                  | 9                                  | Q                       | AXISOLID | Q    | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUADX       | 8                  | 4                                  | Q                       | AXISOLID | QRI  | PLPLANE                | PSHLN2                                    | Yes/Yes                               | Yes                              |

Set BEH=AXISOLID on PSHLN2

#### **Output of Strains**

 $1 = \varepsilon_{rr}$  $2 = \varepsilon_{zz}$  $3 = \varepsilon_{\theta\theta}$  $4 = \gamma_{rz}$ 

#### **Output of Stresses**

 $1 = \sigma_{rr}$  $2 = \sigma_{zz}$  $3 = \sigma_{\theta\theta}$  $4 = \tau_{rz}$ 

To apply a distributed load on this element one needs to apply the pressure using PLOADX1. Note that the distributed loads are integrated over one radian. Hence, any point force applied should also be prescribed over one radian.



Caution: The conventional CTRIA element with BEH=AXISOLID is known to give very poor results when used with incompressible or nearly incompressible behavior (including rubber materials, elastic-plastic or creep), the BEH=IAX should be used.

#### 2-D Axisymmetric Elements with Twist

MSC Nastran has a set of elements that can be used for 2-D axisymmetric simulations with twist. Axisymmetric can be characterized as having no variation in the circumferential direction; that is, these elements uniformly twist in the circumferential direction. The coordinates must be aligned with the basic X-Y system; which is interpreted as the R-Z system. Nonlinear analysis does not support superposition, so the use of Harmonic (Fourier) analysis to describe a load variation in the circumferential direction is not supported. Do not use the PAXSYMH property option. These elements are summarized.

Set BEH=AXISOLID on PSHLN2.

| Axisymmetric with Twist |                    |                                    |                         |          |     |                          |   |                                       |                                  |
|-------------------------|--------------------|------------------------------------|-------------------------|----------|-----|--------------------------|---|---------------------------------------|----------------------------------|
| Element                 | Number<br>of Grids | Number of<br>Integration<br>Points | Interpolation<br>Scheme | ВЕН      | INT | Conventional<br>Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |
| CQUADX                  | 4                  | 4                                  | L                       | AXISOLID | LT  | PLPLANE                  | PSHLN2                                    | Yes/Yes                               | Yes                              |
| CQUADX                  | 8                  | 9                                  | Q                       | AXISOLID | QT  | PLPLANE                  | PSHLN2                                    | Yes/Yes                               | Yes                              |

#### **Output of Strains**

 $1 = \varepsilon_{rr}$   $2 = \varepsilon_{zz}$   $3 = \varepsilon_{\theta\theta}$   $4 = \gamma_{rz}$   $5 = \gamma_{z\theta}$  $6 = \gamma_{\theta z}$ 

#### **Output of Stresses**

- $1 = \sigma_{rr}$  $2 = \sigma_{zz}$  $3 = \sigma_{\theta\theta}$
- $4 = \tau_{rz}$
- 1 v<sub>rz</sub>
- $5 = \tau_{z\theta}$
- $6 = \tau_{\theta z}$


Note the displacements for this element are  $U_r$ ,  $U_z$ , and  $U_{\theta}$  which represents the angular displacement about the symmetry axis measured in radians. This is given as the third degree of freedom, and all torques or single points constraints on this quantity should be treated as the third degree of freedom.

## Large Displacement/Large Strain

All of the 2-D plane stress, plane strain, axisymmetric, and axisymmetric with twist support large displacements and large rotations. For plane stress elements, the thickness of the element is updated due to the in-plane membrane distortion.

The output of the results is always in the basic coordinate system.

# **Axisymmetric Shell Elements**

MSC Nastran does support the use of axisymmetric shell elements that is entered as the CAXISYM in SOL 400. This element may be used as either a 2-node linear element or a 3-node quadratic element. It is not available for the classical linear solution sequences. The CCONEAX may be used for linear solution sequences, but it should not be used in nonlinear solution sequences.

To apply a distributed load on this element, one needs to apply the pressure using PLOADX1. Note that the distributed loads are integrated over one radian. Hence, any point force applied should also be prescribed over one radian.

| Axisymmetric Shell |                    |                                    |                           |                                    |                                  |  |  |  |  |
|--------------------|--------------------|------------------------------------|---------------------------|------------------------------------|----------------------------------|--|--|--|--|
| Element            | Number of<br>Grids | Number of<br>Integration<br>Points | Property for<br>Nonlinear | Large<br>Rotation/ Large<br>Strain | Permits<br>Nonlinear<br>Material |  |  |  |  |
| CAXISYM            | 2                  | 2                                  | PAXISYM                   | Yes/Yes                            | Yes                              |  |  |  |  |
| CAXISYM            | 3                  | 3                                  | PAXISYM                   | Yes/Yes                            | Yes                              |  |  |  |  |

# **Output of Strains**

- $1 = \varepsilon_s$  = meridional membrane
- 2 =  $\varepsilon_{\theta}$  = circumferential membrane
- $3 = \gamma_t$  = transverse shear strain

## **Output Of Stresses**

- $1 = \sigma_s$  = meridional stress
- 2 =  $\sigma_{\theta}$  = circumferential stress
- $3 = \tau_t = \text{transverse shear stress}$

The degrees of freedom for this element are:



$$U_r, U_z, U_t$$

# **3-D Membrane, Plate, and Shell Elements**

In MSC Nastran terminology, these are considered to be 3-D plane stress type elements. In Patran terminology, because topologically they are not volumes they are labeled as 2-D elements. In MSC Nastran, the definition of membranes, plates, and shells are done through the same topological classes. These element satisfy plane-stress conditions as the normal stresses through the thickness are zero.

These elements exhibit significant differences between the classic and the advanced formulation. MSC Nastran has multiple flavors of these elements:

Homogeneous linear behavior for combined membrane, bending and transverse shear.

Homogeneous linear behavior for pure membrane or bending or transverse shear.

Non-homogeneous linear behavior for combined membrane, bending and transverse shear.

Homogeneous nonlinear behavior for combined membrane, bending and transverse shear.

Homogeneous nonlinear behavior for membranes.

Layered (composite) behavior where material behavior is linear.

Layered (composite) behavior where material behavior is nonlinear.

Here, the word Homogeneous refers to the thickness direction. One can use user subroutines to vary the material behavior over the surface of a membrane or shell element.

## **Material Nonlinear Behavior**

If nonlinear material occurs in the model, the recommended solution is to use PSHLN1 and when using composites, LAM on the PCOMP or PCOMG should not be set to SMEAR or SMCORE.

Additionally, the NOCOMPS parameter should be set to 1 to insure that the ply stresses, strains and failure indices are calculated and available for output.

## **Pure Membrane Behavior**

If pure membrane behavior is required than set BEH=MB on the PSHLN1 option. This results in a membrane element which utilizes only the translational degrees of freedom. This formulation has no resistance to bending and caution may be required to insure non-singular behavior. When Large Displacement is set, the differential stiffness matrix is created and for tension based structures; such as balloons, a stable system will occur.

|         | 3-D Membrane Elements |                                    |                         |     |     |                          |   |                                       |                                  |  |  |
|---------|-----------------------|------------------------------------|-------------------------|-----|-----|--------------------------|---|---------------------------------------|----------------------------------|--|--|
| Element | Number<br>of Grids    | Number of<br>Integration<br>Points | Interpolation<br>Scheme | BEH | INT | Conventional<br>Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |  |  |
| CTRIA3  | 3                     | 1                                  | L                       | MB  | L   | PSHELL                   | PSHLN1                                    | Yes/Yes                               | Yes                              |  |  |
| CQUAD4  | 4                     | 4                                  | L                       | MB  | L   | PSHELL                   | PSHLN1                                    | Yes/Yes                               | Yes                              |  |  |



| CTRIA6       | 6      | 7 | Q | MB | Q | PSHELL | PSHLN1 | Yes/Yes | Yes |
|--------------|--------|---|---|----|---|--------|--------|---------|-----|
| CQUAD8       | 8      | 9 | Q | MB | Q | PSHELL | PSHLN1 | Yes/Yes | Yes |
| * With excep | otions |   |   |    |   |        |        |         |     |

**Output of Strains** 

 $1 = \varepsilon_{xx}$  $2 = \varepsilon_{yy}$  $3 = \varepsilon_{xy}$ 

**Output of Stresses** 

$$1 = \sigma_{xx}$$
$$2 = \sigma_{yy}$$
$$3 = \tau_{xy}$$

# **Shells**

There are two classes of advanced shell elements in MSC Nastran SOL 400.

The first group (marked LDK) is thin shell elements based upon Kirchhoff theory. The second group (marked L, LRIH, and QRI) is thick shell elements that support transverse shears based upon Mindlin theory. There are the preferred elements for composite simulation.

|              |                    |                                    |                         | 3-D Sh | ell      |                                 |   |                                       |                                   |
|--------------|--------------------|------------------------------------|-------------------------|--------|----------|---------------------------------|---|---------------------------------------|-----------------------------------|
| Element      | Number<br>of Grids | Number of<br>Integration<br>Points | Interpolation<br>Scheme | BEH    | INT      | Convention al Property          | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material* |
| CTRIA3       | 3                  | 1                                  | L                       | DCTN   | LDK      | PSHELL or<br>PCOMP or<br>PCOMPG | PSHLN1                                    | Yes/Yes                               | Yes                               |
| CQUAD4       | 4                  | 4                                  | L                       | DCT    | L        | PSHELL or<br>PCOMP or<br>PCOMPG | PSHLN1                                    | Yes/Yes                               | Yes                               |
| CQUAD4       | 4                  | 1                                  | L                       | DCT    | LRI<br>H | PSHELL or<br>PCOMP or<br>PCOMPG | PSHLN1                                    | Yes/Yes                               | Yes                               |
| CQUAD4       | 4                  | 4                                  | L                       | DCTN   | LDK      | PSHELL or<br>PCOMP or<br>PCOMPG | PSHLN1                                    | Yes/Yes                               | Yes                               |
| CQUAD8       | 8                  | 4                                  | Q                       | DCT    | QRI      | PSHELL or<br>PCOMP or<br>PCOMPG | PSHLN1                                    | Yes/Yes                               | Yes                               |
| * With excep | tions              |                                    |                         |        |          |                                 |   |                                       |                                   |



| INT Code | Integration Type  |
|----------|---|
| LDK      | Linear Displacement and Rotation, Kirchhoff theory (thin shell) |
| L        | Linear Displacement and Rotation                                |
| LRIH     | Linear Displacement and Rotation, Reduced Integration           |
| QRI      | Quadratic Displacement and Rotation, Reduced Ingetration        |

The output of strains and stresses are given with respect to a coordinate system attached to the element. This coordinate system is updated with the deformation if the LGDISP parameter is activated.

### **Output of Strains**

 $1 = \varepsilon_{xx}$   $2 = \varepsilon_{yy}$   $3 = \varepsilon_{zz}$   $4 = \gamma_{xy}$   $5 = \gamma_{yz}$  for thick shell only  $6 = \gamma_{zx}$  for thick shell only

# **Output of Stresses**

$$1 = \sigma_{xx}$$

$$2 = \sigma_{yy}$$

$$3 = \sigma_{zz} = 0$$

$$4 = \tau_{xy}$$

$$5 = \tau_{yz} \text{ for thick shell only}$$

$$6 = \tau_{zx} \text{ for thick shell only}$$

These elements cannot be used with the IMPLICIT CREEP model specified on the MATEP option.

## Large Displacement/Large Strain

Whether one uses the classical MSC Nastran formulation or the advanced element formulation, these elements support large displacement and large rotations. When using the advanced element formulation, the membrane strains may become large, but the curvature strains remain small. The bending versions of these elements always satisfy the Kirchhoff assumptions that the normal to the shell remains normal and plane through the thickness remains a plane. If either of these assumptions are violated, one effectively has a three dimensional stress distribution, and one should use solid elements.

# **Shear Panel**

MSC Nastran supports a 4-node shear panel element for linear analysis. Here, linear analysis means small deformation and linear isotropic elastic material.



While SOL 400 has an alternative formulation when PSHEARN is entered, it provides a membrane formulation to the element. It is not recommended that one uses the PSHEARN option in SOL 400.

# **3-D Solid Shell Element**

The solid shell element is an element that may be used for structural problems that transition between shell behavior and solid behavior. The element appears like a CHEXA element and is input using this option, but it is an oriented element such that it has very good bending characteristics.

The solid shell element uses different integration schemes in the plane of element and in the thickness direction of element:

- In the element plane, it uses a reduced integration scheme with single integration point. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.
- In the thickness direction, the element is integrated numerically using different points through the element thickness based upon if it is COMPOSITE or NON-COMPOSITE material. The numerical integration for non-composite material is done using Simpson's rule with 5 points, the first and last points are located on the top and bottom surface. For composite material, every layer will have three integration points with the first and third on the surface of each layer.

The following Figure 5-1 demonstrates the arrangement of the integration schemes for non-composite and composite.

The element relaxes the Kirchhoff shell assumptions that normal remain normal, and it results in a full (6component) stress state. The element may be used either as a homogenous material or as a composite (layered) material. It is not necessary to define the thickness because this is obtained directly from the coordinates.

The element system has a local coordinate system which is updated with large displacements. In versions prior to the MSC Nastran 2014 release, the results were given with respect to the basic coordinate system. Currently, the results are given with respect to the local system.







# **Output of Strains**

 $1 = \varepsilon_{xx}$   $2 = \varepsilon_{yy}$   $3 = \varepsilon_{zz}$   $4 = \gamma_{xy}$   $5 = \gamma_{yz}$  $6 = \gamma_{zx}$ 

# **Output of Stresses**

 $1 = \sigma_{xx}$   $2 = \sigma_{yy}$   $3 = \sigma_{zz}$   $4 = \tau_{xy}$   $5 = \tau_{yz}$  $6 = \tau_{zx}$ 

For a homogeneous material, use PSLDN1.

Set BEH=SOLID on PSLDN1

| Element | Number of<br>Grids | Integration –<br>INT                  | Conventional<br>Property | Auxiliary<br>Property for<br>Nonlinear | Large<br>Rotation/<br>Large Strain | Permits<br>Nonlinear<br>Material |
|---------|--------------------|---------------------------------------|--------------------------|--|------------------------------------|----------------------------------|
| CHEXA   | 8                  | ASTN<br>1 per layer<br>Assumed Strain | PSOLID                   | PSLDN1                                 | Yes /Yes                           | Yes                              |

For a layered composite material, use PCOMPLS. Note in the Figure 5-1 that for each layer, there are three layer points, which allows an accurate calculation of the inter laminar shear.

| Element | Number of<br>Grids | Integration – INT                     | Property for<br>Nonlinear | Large<br>Rotation/<br>Large Strain | Permits<br>Nonlinear<br>Material |
|---------|--------------------|---------------------------------------|---------------------------|------------------------------------|----------------------------------|
| CHEXA   | 8                  | ASTN<br>1 per layer<br>Assumed Strain | PCOMPLS                   | Yes /Yes                           | Yes                              |

# **3-D Volumetric Solid Elements**



MSC Nastran has a set of elements that can be used for 3-D volumetric/solid simulations. These simulation are characterized by having no dominate geometric direction and a complete (6-component) stress state. The

advanced elements include lower- and higher-order tetrahedral, pentahedral, and hexahedral elements with a variety of integration schemes that have the following geometry.

# **Output of Strains**

$$\begin{split} 1 &= \epsilon_{xx} \\ 2 &= \epsilon_{yy} \\ 3 &= \epsilon_{zz} \\ 4 &= \gamma_{xy} \\ 5 &= \gamma_{yz} \\ 6 &= \gamma_{zx} \end{split}$$

# **Output of Stresses**

$$1 = \sigma_{xx}$$

$$2 = \sigma_{yy}$$

$$3 = \sigma_{zz}$$

$$4 = \tau_{xy}$$

$$5 = \tau_{yz}$$

$$6 = \tau_{zx}$$









These elements are summarized.

Set  ${\tt BEH=SOLID}$  on PSLDN1 unless indicated otherwise



|         |                    |                                    |                         | 3-D So | olid |                           |   |                                       |                                  |
|---------|--------------------|------------------------------------|-------------------------|--------|------|---------------------------|---|---------------------------------------|----------------------------------|
| Element | Number<br>of Grids | Number of<br>Integration<br>Points | Interpolation<br>Scheme | BEH    | INT  | Convention<br>al Property | Auxiliary<br>Property<br>for<br>Nonlinear | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |
| CTETRA  | 4                  | 1                                  | L                       | SOLID  | L    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CTETRA  | 4                  | 4                                  | L & CUBIC               | ISOL   | L    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | MATEP                            |
| CPENTA  | 6                  | 6                                  | L                       | SOLID  | L    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CHEXA   | 8                  | 8                                  | L                       | SOLID  | L    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CHEXA   | 8                  | 1                                  | L                       | SOLID  | LRIH | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CTETRA  | 10                 | 9                                  | Q                       | SOLID  | Q    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CTETRA  | 10                 | 4                                  | Q                       | SOLID  | LRIH | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CPENTA  | 15                 | 21                                 | Q                       | SOLID  | Q    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CHEXA   | 20                 | 27                                 | Q                       | SOLID  | Q    | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |
| CHEXA   | 20                 | 8                                  | Q                       | SOLID  | QRI  | PSOLID                    | PSLDN1                                    | Yes/Yes                               | Yes                              |

Caution: The conventional CTETRA element with BEH=SOLID is known to give very poor results when used with incompressible or nearly incompressible behavior (including rubber materials, elastic-plastic or creep), the BEH=ISO should be used.

# **Composite Solid Elements**

There are a series of continuum elements that may also be used to model composite materials and are also used for gasket simulations. These elements are available for plane strain, axisymmetric, and threedimensional behavior. These elements are conventional from the degrees of freedom perspective and the interpolation functions. What makes them different from conventional elements is that in one direction, there are multiple layers that support multiple materials. The advantage of these elements are that they are relatively easy to use, but it should be recognized that they are not tuned for bending behavior and multiple elements through the thickness may be required.



8-node or 20-node solid continuum composite elements

To activate the characteristics of these elements for composites, one needs to specify additional Property data.



| Element      | Number of<br>Grids | Integration –<br>INT | Property for<br>Nonlinear | BEH    | Large<br>Rotation/<br>Large Strain | Permits<br>Nonlinear<br>Material |  |  |  |
|--------------|--------------------|----------------------|---------------------------|--------|------------------------------------|----------------------------------|--|--|--|
| Plane Strain |                    |                      |                           |        |                                    |                                  |  |  |  |
| CQUAD4       | 4                  | L 2 per layer        | PLCOMP                    | COMPS  | Yes/Yes                            | Yes                              |  |  |  |
| CQUAD8       | 8                  | Q 2 per layer        | PLCOMP                    | COMPS  | Yes/Yes                            | Yes                              |  |  |  |
| Axisymmetri  | Axisymmetric       |                      |                           |        |                                    |                                  |  |  |  |
| CQUADX       | 4                  | L 2 per layer        | PLCOMP                    | AXCOMP | Yes/Yes                            | Yes                              |  |  |  |
| CQUADX       | 8                  | Q 2 per layer        | PLCOMP                    | AXCOMP | Yes/Yes                            | Yes                              |  |  |  |
| Hexahedral   |                    |                      |                           |        |                                    |                                  |  |  |  |
| CHEXA        | 8                  | L 4 per layer        | PCOMPLS                   | SLCOMP | Yes/Yes                            | Yes                              |  |  |  |
| CHEXA        | 20                 | Q 4 per layer        | PCOMPLS                   | SLCOMP | Yes/Yes                            | Yes                              |  |  |  |

## **Gasket Elements**

The lower-order continuum composite elements are also used to model gasket materials. When used in this manner, the number of layers is one and the material is defined through the MATG option. Note that these elements may be collapsed to a pentahedral so they can model the gasket between two regions modeled with tetrahedral elements. Care should be exercised to make sure the collapsing occurs in the plane of the gasket material. It is not necessary that the elements match the mesh in the surrounding material. One may utilize the contact capability to overcome the mesh incompatibility.

| Element      | Number<br>of Grids | Integration – INT | Conventiona<br>I Property | Auxiliary<br>Property for<br>Nonlinear | BEH    | Large<br>Rotation/<br>Large<br>Strain | Permits<br>Nonlinear<br>Material |
|--------------|--------------------|-------------------|---------------------------|--|--------|---------------------------------------|----------------------------------|
| Plane Strain | n                  |                   |                           |  |        |                                       |                                  |
| CQUAD4       | 4                  | L 2 per layer     | PLPLANE                   | PSHNL2                                 | COMPS  | Yes/No                                | MATG                             |
| Axisymmet    | ric                |                   |                           |  |        |                                       |                                  |
| CQUADX       | 4                  | L 2 per layer     | PLPLANE                   | PSHNL2                                 | AXCOMP | Yes/No                                | MATG                             |
| Hexahedral   | l                  |                   |                           |  |        |                                       |                                  |
| CHEXA        | 8                  | L 4 per layer     | PSOLID                    | PSLDN1                                 | SLCOMP | Yes/No                                | MATG                             |

# **Interface Elements**

There are a series of elements that are used to model the onset and progression of delamination of the bonding materials using the Cohesive Zone Method. These elements are available for plane strain, axisymmetric, and three-dimensional behavior. From a meshing perspective, these elements are unique

because one can enter a zero thickness. The interface elements provide two integration schemes. The first uses the conventional Gaussian integration scheme while the other uses a nodal lumping scheme (Lobatto-Cotes). The latter scheme may be advantageous when the interface material is relatively stiff compared to the surrounding material.

Note that the orientation of the element dictates the direction of the interface/delamination. It is not necessary that the elements match the mesh in the surrounding material. One may utilize the contact capability to overcome the mesh incompatibility. The material properties are defined using the MCOHE material model Cohesive Zone Modeling (MCOHE). Note that delamination simulations are highly nonlinear and one must exercise caution in applying the boundary conditions.

The element is written with respect to a local coordinate system, relating the relative displacement and the normal and shear traction. The output is given with respect to the local coordinate system. This system is updated (rotated) when large displacement is used. The element does not include a mass matrixor a geometric or initial stress stiffness matrix. It also does not support application of distribute loads.

The higher-order elements are not fully quadratic; they are quadratic in the plane of the interface but linear through the thickness.

The elements are shown below.





Integration point schemes for linear interface element



Integration point schemes for linear 3-D interface element





# Integration point schemes for quadratic 3-D interface element

| Element      | Number of<br>Grids | Integration – INT | Property for<br>Nonlinear | Large<br>Rotation/<br>Large Strain | Permits<br>Nonlinear<br>Material |
|--------------|--------------------|-------------------|---------------------------|------------------------------------|----------------------------------|
| Plane Strain | l                  |                   |                           |                                    |                                  |
| CIFQUAD      | 4                  | 2<br>L            | PCOHE                     | Yes/No                             | MCHOE                            |
| CIFQUAD      | 8                  | 2<br>Q/L          | PCOHE                     | Yes/No                             | MCHOE                            |
| Axisymmetr   | ric                |                   |                           |                                    |                                  |
| CIFQDX       | 4                  | 2<br>L            | PCOHE                     | Yes/No                             | MCHOE                            |
| CIFQDX       | 8                  | 2<br>Q/L          | PCOHE                     | Yes/No                             | MCHOE                            |
| Solid        |                    |                   |                           |                                    |                                  |
| CIFPENT      | 8                  | 3<br>L            | PCOHE                     | Yes/No                             | MCHOE                            |



| Element | Number of<br>Grids | Integration – INT | Property for<br>Nonlinear | Large<br>Rotation/<br>Large Strain | Permits<br>Nonlinear<br>Material |
|---------|--------------------|-------------------|---------------------------|------------------------------------|----------------------------------|
| CIFPENT | 15                 | 6<br>Q/L          | PCOHE                     | Yes/No                             | MCHOE                            |
| CIFHEX  | 8                  | 4<br>L            | PCOHE                     | No                                 | MCHOE                            |
| CIFHEX  | 20                 | 8<br>Q/L          | PCOHE                     | No                                 | MCHOE                            |

### **Output of Strain**

The three strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom face and are given in the local element system:

 $1 = \mathcal{U}_{top} - \mathcal{U}_{bottom}$  $2 = \mathcal{V}_{top} - \mathcal{V}_{bottom}$  $3 = \mathcal{W}_{top} - \mathcal{W}_{bottom} \text{ for 3-D element only}$ 

## **Output of Stress**

$$\begin{split} &1=\sigma_n\\ &2=\tau_{s1}\\ &3=\tau_{s2} \text{ for 3-D element only} \end{split}$$

## Large Displacement/Large Rotation

These elements support large displacements and large rotations. It should be noted that there is no differential (initial) stress stiffness associated with this element.

# Automatic Property Mapping

The "SPROPMAP" keyword in the NLMOPTS bulk data entry provides a convenient option to automatically flag secondary properties like PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2, and PSLDN1. Note that these secondary property entries expose the user to a set of sophisticated 2-D continuum, 3-D beam, shell, and continuum elements in SOL 400. The rules governing the flagging of the additional properties are many and are a function of the problem dimension, material type and procedure. All these rules have been incorporated into the automatic flagging option. These are briefly summarized in the following table:



| Auxiliary<br>Property | Conventional<br>Property | Dimension | Material | Notes  | Unsupported<br>Features of<br>Primary Entry |
|-----------------------|--------------------------|-----------|----------|--------|---|
| PBARN1                | PBARL                    | 1-D       | MAT4     | Note 4 |   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |
| PBEMN1                | PBEAML                   | 1-D       | MAT4     | Note 4 | Tapered Sections                            |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |
| PRODN1                | PROD                     | 1-D       | MAT4     | Note 4 | J,C   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |
| PSHEARN               | PSHEAR                   | 3-D       | MAT4     | Note 4 | F1,F2                                       |
|                       |                          |           | MAT8     |        |   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATS8    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATORT   |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |



| Auxiliary<br>Property | Conventional<br>Property | Dimension | Material | Notes  | Unsupported<br>Features of<br>Primary Entry |
|-----------------------|--------------------------|-----------|----------|--------|---|
| PSHLN1                | PSHELL                   | 3-D       | MAT4     | Note 4 | TS/T, nondefault Z1                         |
|                       |                          |           | MAT5     | Note 4 | and $Z2$ , $121/1^{\circ}$                  |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATS8    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATORT   |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |
| PSHLN1                | PCOMP/<br>PCOMPG         | 3-D       | MAT4     | Note 4 | FT, GE, LAM options                         |
|                       |                          |           | MAT5     | Note 4 | other than BLANK<br>and SYM, SOUTi          |
|                       |                          |           | MAT8     |        |   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATS2    | Note 1 |   |
|                       |                          |           | MATS8    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATORT   |        |   |
|                       |                          |           | MATSMA   | Note 2 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 3 |   |



| Auxiliary<br>Property | Conventional<br>Property | Dimension | Material | Notes  | Unsupported<br>Features of<br>Primary Entry |
|-----------------------|--------------------------|-----------|----------|--------|---|
| PSHLN2                | PLPLANE                  | 2-D       | MAT4     | Note 4 |   |
|                       |                          |           | MAT5     | Note 4 |   |
|                       |                          |           | MATG     | Note 5 |   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATS3    | Note 1 |   |
|                       |                          |           | MATS8    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATORT   |        |   |
|                       |                          |           | MATSMA   | Note 6 |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    | Note 7 |   |
| PSLDN1                | PSOLID                   | 3-D       | MAT4     | Note 4 | IN, ISOP, FCTN                              |
|                       |                          |           | MAT5     | Note 4 |   |
|                       |                          |           | MATS1    | Note 1 |   |
|                       |                          |           | MATEP    |        |   |
|                       |                          |           | MATF     |        |   |
|                       |                          |           | MATORT   |        |   |
|                       |                          |           | MATSMA   |        |   |
|                       |                          |           | MATVE    |        |   |
|                       |                          |           | MATVP    |        |   |





The "SPROPMAP" does not support MATDIGI

Please refer to the Remark 8, related to SPROPMAP, in NLMOPTS (p. 2565) in the Quick Reference Guide. The NLMOPTS entry has more information on property mapping.



82 MSC Nastran: SOL 400 Getting Started Manual Automatic Property Mapping



# Materials

- Material Model Overview 84
- Material Property Definitions 84
- Linear Elastic Behavior 88
- Linear Elastic Materials 89
- Viscoelastic 93
- Elasto-plastic Behavior 97
- Elastoplastic Material Entries 99
- Strain Rate Dependent Yield 101
- Creep (MATVP, CREEP) 103
- Composite (PCOMP or PCOMPG) 107
- Cohesive Zone Modeling (MCOHE) 113
- Progressive Composite Failure 113



# Material Model Overview

A wide variety of materials are encountered in structural analysis problems, and for any one of these materials a range of constitutive models is available to describe the material's behavior. We can broadly classify the materials of interest as those which exhibit the following behavior:

- Elastic which fully recovers when the load is removed
- Rate dependent where the behavior is dependent upon the rate of the deformations. This includes
  viscoelastic materials like rubber and glass
- Inelastic which do not recover when loads are removed and demonstrate permanent deformation. These materials include metals represented by elastic-plastic materials, ice, and material that exhibits damage.

The following sections describe how to model material behavior in SOL 400. Modeling material behavior consists of both specifying the constitutive models used to describe the material behavior and defining the actual material data necessary to represent the material. Directional dependency can be included for materials other than isotropic materials. Each section discusses the constitutive (stress strain) relation, provides graphic representation of the models, and includes recommendations and cautions concerning the use of the models.

# Material Property Definitions

It is assumed that the material is a continuum and the material properties remain constant. This means that the material does not contain gaps or voids and the temperature of the structure is constant.

| Linear                                       | Deformations are directly proportional to the applied load (i.e., strain is directly proportional to stress).  |
|--|--|
| Elastic                                      | An elastic structure returns to its original, undeformed shape when the load is removed.   |
| Homogeneous                                  | The material is the same throughout — material properties are independent of location within the material.   |
| Isotropic                                    | Material properties do not change with the direction of the material.  |
| Modulus of elasticity<br>(Young's modulus) E | E is the constant of proportionality relating stress-to-strain for uniaxial behavior in the linear region. The greater the value of E, the stiffer the material. |
| Shear modulus (Modulus of rigidity) G        | G is the constant of proportionality relating shear stress to shear strain in the linear region.   |
| Poisson's ratio                              | It is the absolute value of the ratio of lateral linear strain to axial linear strain.   |

Definition and properties of materials are listed in the Table 6-1.

#### Table 6-1 Definitions and Properties of Materials

If the loading on a structure is sufficient to exceed the linear elastic limit of the material, then nonlinear methods are required to predict the nature of the plastically (permanently) deformed state.





A typical stress-strain curve of structural steel is shown in the Figure 6-1.

Figure 6-1 Stress–Strain Curve of Structural Steel

Modeling of nonlinear material behavior is a critical component to obtain the structural response of structures; especially, when either the loads are large, the temperatures are high, and/or the non-metallic material is used. MSC Nastran SOL 400 provides a variety of models and, in some cases, multiple numerical implementation of these models. This latter is due to the historical incorporation of solution sequences 106 and 129. The newer implementations of material models is based upon incorporating technology from Marc and is the preferred approach. The newer material formulation also requires the use of the newer element methodology which is often labeled the "advanced" elements. Aspects of these elements are described in Chapter 11: Element Library. Hence, to use the advanced material models, it is necessary to supplement the traditional property entries with new property entries (PBARN1, PBEMN1, PRODN1, PSHLN1, PSHLN2, and PSLDN1). An alternate approach is to use the NLMOPTS bulk data option with the PROPMAP keyword.

# **SOL 400 Material Entries**

The following material bulk data entries are available in SOL 400. An overview of each of these options is presented in the sections of this chapter and detailed in the *QRG*, Chapter 8: Bulk Data Entries. All standard MSC Nastran materials are also available in SOL 400.



A summary of material models is given in Table 6-2.

| Table 6-2 Material Model  | Summary           |              |
|---|-------------------|--------------|
| Physics   | Bulk Data         | Constraint   |
| Isotropic elasticity  | MAT1              |              |
| Anisotropic elasticity  | MAT2              | Shells only  |
| Orthotropic elasticity  | MAT3              | CTRIA6X only |
| Orthotropic elasticity  | MAT8              | Shells only  |
| 3-D orthotropic elasticity  | MATORT            |              |
| Conventional plasticity (small strain)  | MATEP or<br>MATS1 |              |
| Plasticity with Chaboche model, Barlat, Viscoplastic, Power<br>Law, Johnson-Cook, Kumar | MATEP             |              |
| Large strain plasticity   | MATEP             |              |
| Advanced failure model  | MATF              |              |
| Gasket Material   | MATG              |              |
| Hyperelastic (Mooney, Ogden, Aruda-Boyce, Gent, Foam)                                   | MATHE             |              |
| Generalized Mooney  | MATHP             |              |
| Shape memory material   | MATSMA            |              |
| Small strain nonlinear elasticity   | MATS1             |              |
| Orthotropic nonlinear elastic   | MATSORT           |              |
| Digimat composite model   | MATDIGI           |              |
| Small strain isotropic viscoelastic   | MATVE             |              |
| Large strain viscoelastic   | MATVE             |              |
| Viscoplastic  | MATVP'            |              |
| Creep (model)   | CREEP             | NLPARM       |
| Creep   | MATVP             |              |
| Cohesive zone model   | MCOHE             |              |

 Table 6-3
 Material Characteristics and Application



| Material                             | Characteristics  | Examples  | Models  |
|--------------------------------------|--|---|---|
| Failure Criteria<br>(MATF)           | Determines failure initiation and progressive failure.   | Aircraft panels   | Maximum stress<br>Tasi-Wu<br>Puck             |
| Failure Criteria<br>(MATDIGI)        | Determines failure initiation and progressive failure.   | Fiber reinforced plastic  | Unit cell model                               |
| Hyperelastic<br>(MATHE)              | Stress function of instantaneous<br>strain. Nonlinear load-<br>displacement relation. Unloading<br>path same as loading.                                     | Rubber  | Mooney<br>Ogden<br>Arruda-Boyce<br>Gent       |
| Hypoelastic<br>(MATUSR)              | Rate form of stress-strain law   | Concrete  | User-defined                                  |
| Nonlinear Elastic<br>( <b>MATG</b> ) | Loading with mutiple unloading curves due to damage.   | Automotive gaskets  | Gasket model                                  |
| Nonlinear Elastic<br>(MATSORT)       | Simlified nonlinear orthotropic elasticity   | Wood  | Simple  |
| Creep<br>(MATVP)                     | Strains increasing with time under<br>constant load. Stresses decreasing<br>with time under constant<br>deformations. Creep strains are<br>noninstantaneous. | Metals at high<br>temperatures, polymide<br>films, semiconductor<br>materials | Norton<br>Maxwell                             |
| Elastoplasticity<br>(MATEP)          | Yield condition flow rule and<br>hardening rule necessary to<br>calculate stress, plastic strain.<br>Permanent deformation<br>upon unloading.                | Metals<br>Soils   | von Mises<br>Isotropic<br>Hill's Anisotropic  |
| Viscoelastic<br>(MATVE)              | Time dependence of stresses in<br>elastic material under loads. Full<br>recovery after unloading.  | Rubber,<br>Glass, industrial<br>plastics                                      | Simo Model<br>Narayanaswamy                   |
| Viscoplastic<br>(MATVP)              | Combined plasticity and creep phenomenon   | Metals<br>Powder  | Power law                                     |
| Shape Memory<br>( <b>MATSMA</b> )    | Thermal - Mechanical   |   | Aruchhio's model<br>Asaro-Sayeedvafa<br>model |
| Cohesive Zone<br>Method<br>(MCOHE)   | Cohesive material model  | Glue  | Linear and<br>quadratic                       |



The following sections describe how to model material behavior in SOL 400. Modeling material behavior consists of both specifying the constitutive models used to describe the material behavior and defining the actual material data necessary to represent the material. Directional dependency can be included for materials other than isotropic materials. Data for the materials can be entered either directly through the input file or by user subroutines, or material models may be defined in the Patran Materials Application. Each section of this chapter discusses various options for organizing material data for input. Each section also discusses the constitutive (stress-strain) relation and graphic representation of the models and includes recommendations and cautions concerning the use of the models.

# Linear Elastic Behavior

SOL 400 is capable of handling problems with any combination of isotropic, orthrotropic, or anisotropic linear elastic material behavior.

The isotropic linear elastic model is the model most commonly used to represent engineering materials. This model, which has a linear relationship between stresses and strains, is represented by Hooke's Law. Figure 6-2 shows that stress is proportional to strain in a uniaxial tension test. The ratio of stress to strain is the familiar definition of modulus of elasticity (Young's modulus) of the material.

*E* (modulus of elasticity) = (axial stress)/(axial strain)

(6-1)

(6-2)



Experiments show that axial elongation is always accompanied by lateral contraction of the bar. The ratio for a linear elastic material is:

v = (lateral contraction)/(axial elongation)

This is known as Poisson's ratio. Similarly, the shear modulus (modulus of rigidity) is defined as:

G (shear modulus) = (shear stress)/(shear strain)



# **Element Selection for Incompressible Materials**

A Poisson's ratio of 0.5, which would be appropriate for an incompressible material, can be used for the following elements: plane stress, shell, truss, or beam. A Poisson's ratio which is close (but not equal) to 0.5 can be used for constant dilation elements and reduced integration elements in situations which do not include other severe kinematic constraints. Using a Poisson's ratio close to 0.5 for all other elements usually leads to behavior that is too stiff. A Poisson's ratio of 0.5 can also be used with the updated Lagrangian formulation in the multiplicative decomposition framework using the standard displacement elements. In these elements, the treatment for incompressibility is transparent.

# Linear Elastic Materials

A wide variety of materials are encountered in stress analysis problems, and for any one of these materials a range of constitutive models are available to describe the material's behavior, including directional dependencies. We can broadly classify the materials of interest as those which exhibit almost no directional dependence (isotropic materials), versus those that exhibit three mutually orthogonal planes of symmetry (orthogonal materials), versus those that exhibit different elastic properties in different directions (anisotropic materials).

| Material   | Characteristics  |
|--|--|
| Isotropic (MAT1)                                   | Isotropic material – No Directional Dependency – most commonly used material property                                    |
| Two Dimensional Anisotropic (MAT2)                 | General anisotropic two-dimensional material used with plate and shell elements  |
| Axi-symmetric / Solid Orthotropic<br>(MAT3)        | Orthotropic three-dimensional material for use with CTRIAX6, PSHLN2, or PLCOMP   |
| Two Dimensional Orthotropic (MAT8)                 | Two-dimensional orthotropic stress-strain used with the plate and shell elements   |
| Three-Dimensional Anisotropic<br>Material (MATORT) | Orthrotropic material property for the CHEXA, CPENTA, and CTETRA solid elements and solid shell and plane strain element |

# **Isotropic Materials**

Most linear elastic materials are assumed to be isotropic (their elastic properties are the same in all directions). For an isotropic material, every plane is a plane of symmetry and every direction is an axis of symmetry. It can be shown that for an isotropic material:

$$G = E/(2(1+v))$$

The shear modulus G can be easily calculated if the modulus of elasticity E and Poisson's ratio v are known.



(6-3)

### **Specifying Isotropic Material Entries**

Isotropic material models are designated with the MAT1 Bulk Data entry in the MSC Nastran Input File.

| Entry | Description   |
|-------|---|
| MATBV | Defines the material properties for linear isotropic materials. |

### **Patran Materials Application Input Data**

To define an isotropic material in Patran:

- 1. From the Materials Application form, set the Action>Object>Method combination to Create > Isotropic > Manual Input.
- 2. Click Input Properties...

Isotropic linear elastic material models require the following material data via the Input Options subform on the Materials Application form.

| Isotropic-Linear Elastic            | Description  |
|-------------------------------------|--|
| Elastic Modulus                     | Defines the elastic modulus. This property is generally required. May vary with temperature via a defined material field.  |
| Poisson's Ratio                     | Defines the Poisson's ratio. This property is generally required. May vary with temperature via a defined material field.  |
| Density                             | Defines the mass density. This property is optional.   |
| Coefficient of Thermal<br>Expansion | Defines the coefficient of thermal expansion. This property is optional. May vary with temperature via a defined material field.   |
| Reference Temperature               | Defines the stress free temperature. This property is optional. When defining temperature dependent properties, this is the reference temperature from which values will be extracted or interpolated. |

The material density, used to define the mass of the structure, and the damping value are used in dynamic loadings, while the expansion coefficient is used to identify the thermal strains.

# **Orthotropic Materials**

An orthotropic material has three mutually orthogonal planes of symmetry. With respect to a coordinate system parallel to these planes, the constitutive law for this material is given by the following more general form of Hooke's Law:



| ε <sub>11</sub> |   | $1/(E_1)$         | $-(v_{12})/(E_1)$ | $-(v_{13})/(E_1)$ | 0            | 0            | 0            | $\sigma_{11}$            |
|-----------------|---|-------------------|-------------------|-------------------|--------------|--------------|--------------|--------------------------|
| ε <sub>22</sub> |   | $(-v_{12})/(E_1)$ | $1/(E_2)$         | $(-v_{23})/(E_2)$ | 0            | 0            | 0            | $\sigma_{22}$            |
| ε <sub>33</sub> | = | $(-v_{13})/(E_1)$ | $(-v_{23})/(E_2)$ | $1/(E_3)$         | 0            | 0            | 0            | $\sigma_{33}$            |
| $\gamma_{12}$   |   | 0                 | 0                 | 0                 | $1/(G_{12})$ | 0            | 0            | $\boldsymbol{\tau}_{12}$ |
| $\gamma_{23}$   |   | 0                 | 0                 | 0                 | 0            | $1/(G_{23})$ | 0            | $\tau_{23}$              |
| $\gamma_{13}$   |   | 0                 | 0                 | 0                 | 0            | 0            | $1/(G_{13})$ | $\tau_{13}$              |

### **3-D Orthotropic**

Due to symmetry of the compliance matrix,  $\varepsilon_{11} v_{21} = \varepsilon_{22} v_{12}$ ,  $\varepsilon_{22} v_{32} = \varepsilon_{33} v_{23}$ , and  $\varepsilon_{33} v_{13} = \varepsilon_{11} v_{31}$ . Using these relations, a general orthotropic material has nine independent constants:

 $\boldsymbol{\varepsilon}_{11}\,,\,\boldsymbol{\varepsilon}_{22}\,,\,\boldsymbol{\varepsilon}_{33}\,,\,\boldsymbol{v}_{12}\,,\,\boldsymbol{v}_{23}\,,\,\boldsymbol{v}_{31}\,,\,\boldsymbol{G}_{12}\,,\,\boldsymbol{G}_{23}\,,\,\boldsymbol{G}_{31}$ 

These nine constants must be specified in constructing the material model.



## 2-D Orthotropic

Orthotropic material models can be used with 2-D elements, such as plane stress, plane strain, and axisymmetric elements. For example, the orthotropic stress-strain relationship for a plane stress element is:

$$C = \frac{1}{(1 - v_{12}v_{21})} \begin{bmatrix} E_1 & v_{21}E_1 & 0\\ v_{12}E_2 & E_2 & 0\\ 0 & 0 & (1 - v_{12}v_{21})G \end{bmatrix}$$
(6-4)

## **Specifying Orthotropic Material Entries**

2-D and 3-D othrotropic materials are characterized in MSC Nastran using the following bulk data entries.



| Entry        | Description  |
|--------------|--|
| MAT3         | Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry.  |
| MAT2<br>MAT8 | Defines the material property for an orthotropic material for solids and isoparametric shell elements.   |
| MATORT       | Specifies elastic orthotropic material properties for three-dimensional and plane<br>strain behavior for linear and nonlinear analyses in MSC Nastran Implicit<br>Nonlinear in a more general way than MAT2 or MAT8. |

## **Patran Materials Application Input Data**

To define an orthotropic material in Patran:

- 1. From the Materials Application form, set the Action>Object>Method combination to Create>2D or 3D Orthotropic>Manual Input.
- 2. Click Input Properties...

The required properties for orthotropic linear elastic material models vary based on dimension, element type, and thermal dependencies. 3-D orthotropic material models require the following material data (2-D requires a reduced set) via the Input Properties subform on the Materials Application form.

| Orthotropic-Linear Elastic                | Description   |
|---|---|
| Elastic Modulus 11/22/33                  | Defines the elastic moduli in the element's coordinate system. This is<br>required data. May vary with temperature via a defined material field.  |
| Poisson's Ratio 12/23/31                  | Defines the Poisson's ratios relative to the element's coordinate system.<br>This is required data. May vary with temperature via a defined<br>material field.  |
| Shear Modulus 12/23/31                    | Defines the shear moduli relative to the element's coordinate system. This is required data. May vary with temperature via a defined material field.  |
| Coefficient of Thermal Expansion 11/22/33 | Defines the coefficients of thermal expansion relative to the element's coordinate system. These properties are optional. May vary with temperature via a defined material field.                               |
| Reference Temperature                     | Defines the stress free temperature which is an optional property. When<br>defining temperature dependent properties, this is the reference<br>temperature from which values will be extracted or interpolated. |
| Density                                   | Defines the mass density which is an optional property.   |

# **Anisotropic Materials**

Anisotropic material exhibits different elastic properties in different directions. The significant directions of the material are labeled as preferred directions, and it is easiest to express the material behavior with respect to these directions.



The stress-strain relationship for an anisotropic linear elastic material can be expressed as

$$\sigma_{ij} = C_{ijkl} \varepsilon_{kl} \tag{6-5}$$

The values of  $C_{ijkl}$  (the stress-strain relation) and the preferred directions (if necessary) must be defined for an anisotropic material.

### **Specifying Anisotropic Material Entries**

Anisotropic materials are characterized in MSC Nastran using the following bulk data entries.

| Entry | Description  |
|-------|--|
| MAT2  | Defines the material properties for linear anisotropic materials for two-<br>dimensional elements. |

### **Patran Materials Application Input Data**

To define anisotropic material in Patran:

- 1. From the Materials Application form, set the Action > Object > Method combination to Create>2D or 3D Anisotropic>Manual Input.
- 2. Click Input Properties...

Anisotropic linear elastic material models require the following material data via the Input Properties subform on the Materials Application form.

| Anisotropic-Linear Elastic          | Description   |
|-------------------------------------|---|
| Stress-Strain Matrix, Cij           | Defines the upper right portion of the symmetric stress-strain matrix relative to the element's coordinate system.  |
| Coefficient of Thermal<br>Expansion | Defines the coefficients of thermal expansion relative to the element's coordinate system. They are optional properties.  |
| Reference Temperature               | Defines the stress free temperature which is an optional property. When<br>defining temperature dependent properties, this is the reference<br>temperature from which values will be extracted or interpolated. |
| Density                             | Defines the mass density which is an optional property.   |

# Viscoelastic

Viscoelasticity models rate effects in the deformation of materials. Their behavior becomes time dependent and typical phenomena associated with this behavior are relaxation and creep. Relaxation is the diminishing of stress at constant deformation level (figure 1). Creep is the progression of deformation at constant load level (figure 2)





Figure 2: Example of creep behavior

The data required to perform this analysis are the static material constants like Young's moduls and Poisson's ratio or the material constants used in the strain energy function of a hyperelastic material. In a viscoelastic analysis these material constants correspond to the instantaneous or short term behavior of the material. In addition viscoelastic material data are needed to describe the rate effects in the material.

Viscoelastic analysis can be performed in the time domain or in the frequency domain.

In time domain viscoelasticity we study the transient response due to arbitrary time varying loads to capture the rate effects in the deformation process.

In the time domain two models are available:

- The **hereditary integral model** for linear viscoelasticity and the **Simo model** for nonlinear viscoelasticity. The Simo model is an extension of the linear model to finite strain viscoelasticity and leads to similar hereditary integrals as the linear model.
- The Bergstrom-Boyce (B-B) model, which is a phenomenological highly nonlinear model.

In frequency domain viscoelastity we study the stationary dynamic response due to harmonic loads to capture the rate effects in the deformation process. The stationary response due to harmonic loads that are applied with a certain excitation frequency is a response with the same frequency. Frequency domain viscoelasticity is always based on a linear perturbation around a static equilibrium state, which can be the undeformed stressfree state or some other deformed equilibrium state and is therefore limited to small amplitude vibrations around the static equilibrium state. Frequency domain viscoelasticity leads to the concepts of storage modulus and loss modulus, which characterize the frequency dependent stiffness and damping properties of a material. Within the considered amplitude range, the size of the amplitude may still have a substantial effect on the harmonic response, which is known as the Payne effect or Fletcher-Gent effect. With the Payne effect, storage modulus and loss modulus of a material also become vibration amplitude dependent, turning the frequency



response problem into a nonlinear problem. Without the Payne effect the frequency response problem is always linear.

In the frequency domain the hereditary integral models for linear viscoelasticity and nonlinear viscoelasticity (Simo model) are available only. The Bergstrom-Boyce model is not available in the frequency domain.

Because of the presence of damping there is energy dissipation, which in general will be frequency dependent Viscoelasticity can be applied:

- To determine the current state of deformation based on the entire time history of loading.
- To include temperature dependencies.
- To use in conjunction of isotropic, orthotropic, and anisotropic material for small strain problems.
- To use in conjunction with rubber or foam material for large strain problems.

# **Hereditary Integral Model**

The stress-strain equations in viscoelasticity are not only dependent on the current stress and strain state (as represented in the Kelvin model), but also on the entire history of development of these states. This constitutive behavior is most readily expressed in terms of hereditary or Duhamel integrals. These integrals are formed by considering the stress or strain build-up at successive times. Two equivalent integral forms exist: the stress relaxation form and the creep function form. In SOL 400, the stress relaxation form is used.

## **Isotropic Viscoelastic Material**

For an isotropic viscoelastic material, SOL 400 assumes that the deviatoric and volumetric behavior are fully uncoupled and that the behavior can be described by a time dependent shear and bulk modules. The bulk moduli is generally assumed to be time independent; however, this is an unnecessary restriction of the general theory.

# Large Strain Viscoelasticity

For an elastomeric time independent material, the constitutive equation is expressed in terms of an energy function W. For a large strain viscoelastic material, Simo generalized the small strain viscoelasticity material behavior to a large strain viscoelastic material.

# **Thermal-Rheologically Simple Materials**

The relaxation behavior of most viscoelastic materials is temperature dependent. For many of these materials it is observed that when displaying the relaxation curves for different temperatures on a logarithmic time scale, the shape of the curves doesn't change, but they shift along the logarithmic time axis as shown for two normalized relaxation curves in figure Figure 6-3. Such material behavior is called Thermo-Rheologically simple (TRS) material behavior. In general it is expected that when the temperature is higher than the reference temperature, the relaxation is faster so the curve makes a left-shift and when the temperature is lower than the reference temperature, the relaxation is slower so the curve makes a right-shift

A "reduced" or "pseudo" time can be defined for the materials of this type and for a given temperature field. This new parameter is a function of both time and space variables. The viscoelastic law has the same form as



one at constant temperature in real time. If the shifted time is used, however, the transformed viscoelastic equilibrium and compatibility equations are not equivalent to the corresponding elastic equations.

In the case where the temperature varies with time, the extended constitutive law implies a nonlinear dependence of the instantaneous stress state at each material point of the body upon the entire local temperature history. In other words, the functionals are linear in the strains but nonlinear in the temperature.

The time scale of experimental data is extended for Thermo-Rheologically Simple materials. All characteristic functions of the material must obey the same property. The shift function is a basic property of the material and must be determined experimentally. As a consequence of the shifting of the mechanical properties data parallel to the time axis (see Figure 6-3), the values of the zero and infinite frequency complex moduli do not change due to shifting. Hence, elastic materials with temperature-dependent characteristics neither belong to nor are consistent with the above hypothesis for the class of Thermo-Rheologically Simple viscoelastic solids.



## **Narayanaswamy Model**

The annealing of flat glass requires that the residual stresses be of an acceptable magnitude, while the specification for optical glass components usually includes a homogenous refractive index. The design of heat treated processes (for example, annealing) can be accomplished using the Narayanaswamy model. This allows you to study the time dependence of physical properties (for example, volumes) of glass subjected to a change in temperature.

The glass transition is a region of temperature in which molecular rearrangements occur on a scale of minutes or hours, so that the properties of a liquid change at a rate that is easily observed. Below the glass transition temperature  $T_g$ , the material is extremely viscous and a solidus state exists. Above  $T_g$ , the equilibrium structure is arrived at easily and the material is in liquidus state. Hence, the glass transition is revealed by a change in the temperature dependence of some property of a liquid during cooling. If a mechanical stress is applied to a liquid in the transition region, a time-dependent change in dimensions results due to the phenomenon of visco-elasticity.

If a liquid in the transition region is subjected to a sudden change in temperature, a time-dependent change in volume occurs as shown in Figure 6-4. The latter process is called structural relaxation. Hence, structural relaxation governs the time-dependent response of a liquid to a change of temperature.





## **Specifying Viscoelastic Material Entries**

The viscoelastic MATVE and MATTVE material entries are provided for cases where dissipative losses caused by "viscous" (internal friction) effects in materials must be modeled. For time domain analysis, this option is used with an elastic model to define classical linear, small strain, viscoelastic behavior, or with hyperelastic or foam models to define finite linear, large deformation, viscoelastic behavior. As described in the previous section, viscoelastic relaxation data can be fit using the experimental data fitting capability available in Patran.

| Entry  | Description  |
|--------|--|
| MATVE  | Specifies isotropic viscoelastic material properties to be used for quasi-static or dynamic analysis in SOL 400.   |
| MATTVE | Specifies temperature-dependent visco-elastic material properties in terms of<br>Thermo-Rheologically Simple behavior to be used for quasi-static or transient<br>dynamic analysis in SOL 400. |

# **Elasto-plastic Behavior**

Most materials of engineering interest initially respond elastically. Elastic behavior means that the deformation is fully recoverable, so that, when the load is removed, the specimen returns to its original shape. If the load exceeds some limit (the "yield load"), the deformation is no longer fully recoverable. Some parts of the deformation will remain when the load is removed as, for example, when a paper clip is bent too much,



or when a billet of metal is rolled or forged in a manufacturing process. Plasticity theories model the material's mechanical response as it undergoes such nonrecoverable deformation in a ductile fashion. The theories have been developed most intensively for metals, but they are applied to soils, concrete, rock, and ice. These materials behave in very different ways (for example, even large values of pure hydrostatic pressure cause very little inelastic deformation in metals, but quite small hydrostatic pressure may cause a significant, non-recoverable volume change in a soil sample), but the fundamental concepts of plasticity theories are sufficiently general that models based on these concepts have been successfully developed for a wide range of materials. A number of these plasticity modes are available in the SOL 400 material library.

In nonlinear material behavior, the material parameters depend on the state of stress. Up to the proportional limit, i.e., the point at which linearity in material behavior ceases, the linear elastic formulation for the behavior can be used. Beyond that point, and especially after the onset of yield, nonlinear formulations are required. In general, two ingredients are required to ascertain material behavior:

- 1. an initial yield criterion to determine the state of stress at which yielding is considered to begin
- 2. mathematical rules to explain the post-yielding behavior

There are two major theories of plastic behavior that address these criterion differently. In the first, called *deformation theory*, the plastic strains are uniquely defined by the state of stress. The second theory, called flow or *incremental* theory, expresses the increments of plastic strain (irrecoverable strains) as functions of the current stress, the strain increments, and the stress increments. Incremental theory is more general and can be adapted in its particulars to fit a variety of material behaviors. The plasticity models in SOL 400 are "incremental" theories, in which the mechanical strain rate is decomposed into an elastic part and a plastic (inelastic) part through various assumed flow rules.

The incremental plasticity models are formulated in terms of:

- A *yield surface*, which generalizes the concept of "yield load" into a test function which can be used to determine if the material will respond purely elastically at a particular state of stress, temperature, etc.;
- A *flow* rule that defines the inelastic deformation that must occur if the material point is no longer responding purely elastically;
- and some evolution laws that define the *hardening* the way in which the yield and/or flow definitions change as inelastic deformation occurs.

The models also need an elasticity definition, to deal with the recoverable part of the strain models divide into those that are rate-dependent and those that are rate-independent.

SOL 400 includes the following models of inelastic behavior.

- Metal Plasticity (von Mises, Hill, or Barlat)
- Pressure-Dependent models models the behavior of granular (soil and rock) materials or polymers, in which the yield behavior depends on the equivalent pressure stress.
  - Linear Mohr-Coulomb
  - Parabolic Mohr-Coulomb


# **Elastoplastic Material Entries**

Each of the elastoplastic models described in this section can be selected with the MATEP bulk data entry.

| Entry  | Description  |
|--------|--|
| MATEP  | Specifies elasto-plastic material properties to be used for large deformation analysis.  |
| MATTEP | Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi-static, or transient dynamic analysis. |

#### **Patran Materials Application Input Data**

Table 6-4

To define an inelastic material in Patran:

- 1. From the Materials Application form, set the Action>Object>Method combination to Create > Isotropic-or-Orthotropic-or-Anisotropic > Manual Input.
- 2. Click Input Properties..., and select Elastoplastic from the Constitutive Model pull-down menu.

The required properties for describing elasticplastic behavior vary based on material type, dimension, type of nonlinear data input, hardening rule, yield criteria, strain rate method, and thermal dependencies.

Table 6-4 shows the various input options and criteria available to you for defining elastoplastic behavior.

|                             |   |   | ·····   |   |
|-----------------------------|---|---|---|---|
| Constitutive<br>Model       | Nonlinear Data<br>Input                     | Hardening Rule  | Yield Criteria  | Strain Rate<br>Method   |
| <ul> <li>Plastic</li> </ul> | <ul> <li>Stress/Strain<br/>Curve</li> </ul> | <ul><li>Isotropic</li><li>Kinematic</li><li>Combined</li></ul>                              | <ul> <li>von Mises</li> <li>Tresca</li> <li>Mohr-Coulomb</li> <li>Drucker-Prager</li> <li>Parabolic Mohr-Coulomb</li> </ul> | <ul> <li>Piecewise Linear</li> <li>Cowper-<br/>Symonds</li> </ul>               |
|                             | <ul> <li>Hardening Slope</li> </ul>         | <ul><li>Isotropic</li><li>Kinematic</li><li>Combined</li><li>Piecewise<br/>Linear</li></ul> | <ul> <li>von Mises</li> <li>Tresca</li> <li>Mohr-Coulomb</li> <li>Drucker-Prager</li> <li>None</li> </ul>                   | <ul> <li>None</li> <li>Piecewise Linear</li> <li>Cowper-<br/>Symonds</li> </ul> |

#### Elastoplastic Model Summary

#### Patran Nonlinear Data Input



The type of nonlinear data input you choose to use to define elastoplastic material behavior determines the input data required for the Input Properties subform on the Materials Application form.

Stress/Strain Curve – All stress-strain curves are input as piecewise linear. Patran transfers the stressstrain curve input on the material property field directly to the TABLES1 entry.

The number of linear segments used to define the stress-strain curve may be different from one material to another. The same strain breakpoints need not be used for all of the different material's stress-strain curves. It is recommended to define the stress-strain curves throughout the range of strains which the analysis is likely to predict. If the analysis predicts a plastic strain greater than the last point defined by you, SOL 400 continues the analysis after shifting the last strain breakpoint on that curve to match the predicted value, thereby changing (reducing) the work hardening slope for the last segment of the curve.

- Hardening Slope The hardening slope and the yield point are required with this Nonlinear Data Input option.
- Perfectly Plastic Perfect plasticity is described by simply specifying the yield point.

Tables 6-5 through 6-7 provide descriptions for the input data for each of the four types of nonlinear input.

| Property Name                           | Description   |
|---|---|
| Stress /Strain Curve or<br>Yield Stress | Defines the Cauchy stress vs. logarithmic strain (also called equivalent tensile stress versus total equivalent strain) by reference to a tabular field. The field is selected from the <b>Field Definition</b> list. The field is created using the Fields application. For <b>Perfectly Plastic</b> models, only a <b>Yield Stress</b> needs to be entered. |
|   | Can also be strain rate dependent if <b>Strain Rate Method</b> is <b>Piecewise</b><br><b>Linear</b> . Accepts field of yield <b>stress vs. strain</b> rate.   |
| Coefficient C                           | Visible if Strain Rate Method is Cowper-Symonds.  |
| Inverse Exponent P                      | Visible if Strain Rate Method is Cowper-Symonds.  |
| Alpha                                   | When set to Linear Mohr-Coulomb, defines the slope of the yield surface in square root J2 versus J1 space. This property is required.   |
| Beta                                    | When set to <b>Parabolic Mohr-Coulomb</b> , defines the beta parameter in the equation that defines the parabolic yield surface in square root J2 versus J1 space. This property is required.   |
| <b>Note:</b> Perfectly Plastic is       | identical to Stress/Strain except that no hardening rules apply.  |

Table 6-5 Isotropic - Stress/Strain Curve or Perfectly Plastic: All Yield Functions



| Property Name  | Description   |  |  |  |
|--|---|--|--|--|
| Stress vs. Strain or Tensile<br>Yield Stress   | Same as description for Isotropic Elastic-Plastic. If <b>Strain Rate Method</b> is <b>Piecewise Linear</b> , accepts field of yield stress vs. strain rate. |  |  |  |
|  | Or defines an isotropic yield stress. It is a required property when the <b>Plasticity Type</b> is <b>Perfectly Plastic</b> .                               |  |  |  |
| Stress 11/22/33 Yield<br>Ratios  | Defines the ratios of direct yield stresses to the isotropic yield stress in the element's coordinate system.   |  |  |  |
| Stress 12/23/31 Yield<br>Ratios  | Defines the ratios of shear yield stresses to the isotropic shear yield stress (yield divided by square root three) in the element's coordinate system.     |  |  |  |
| <b>Note:</b> Perfectly Plastic is identical to Elastic-Plastic except that no hardening rules apply. |   |  |  |  |

#### Table 6-6 Anisotropic/Orthotropic - Stress/Strain Curve or Perfectly Plastic: All Yield Functions

#### Hardening Slope - Nonlinear Data Input

 Table 6-7
 Isotropic/Anisotropic/Orthotropic - Hardening Slope

| Property Name           | Description  |
|-------------------------|--|
| Hardening Slope         | Slope of the stress-strain curve once yielding has started.  |
| Yield Point             | Defines the stress level at which plastic strain begins to develop.  |
| Internal Friction Angle | When yield function is set to Mohr-Coulomb or Drucker-Prager this gives the parameter describing the effect of hydrostatic pressure on the yield stress. |

# Strain Rate Dependent Yield

Strain rate effects cause the structural response of a body to change because they influence the material properties of the body. These material changes lead to an instantaneous change in the strength of the material. Strain rate effects become more pronounced for temperatures greater than half the melting temperature ( $T_m$ ), but are sometimes present even at room temperature.

Yield stress variation with strain rate is given using one of four options:

- 1. The breakpoints and slopes for a piecewise linear approximation to the yield stress strain rate curve are given. The strain rate breakpoints should be in ascending order, or
- 2. The Cowper and Symonds model is used. The yield behavior is assumed to be completely determined by one stress-strain curve and a scale factor depending on the strain rate.
- 3. The yield stress may be given as a function of the plastic strain, strain, and/or the temperature using the TABLD3 bulk data entry.
- 4. The Power Law, Rate Power Law, Johnson Cook model, or Kumar model.



Note: If multiple material models are used, they must all be expressed as piecewise linear or as Cowper and Symonds model.

#### **Perfectly Plastic**

A material is said to be "perfectly plastic" if, upon the stress state touching the yield surface, an infinitesimal increase in stress causes an arbitrarily large plastic strain. The uniaxial stress-strain diagram for an elastic-perfectly plastic material is shown in Figure 6-5. Some materials, such as mild steel, behave in a manner which is close to perfectly plastic.



Power Law Model

$$\sigma_y = A(\varepsilon_0 + \overline{\varepsilon^p})^m + B\overline{\varepsilon^p}^n$$

where  $\sigma_y$  is the yield stress,  $\varepsilon_0$  is the strain corresponding to initial yield stress,  $\overline{\varepsilon^p}$  is equivalent plastic  $\vdots$  strain,  $\overline{\varepsilon^p}$  is rate of equivalent plastic strain rate and *A*, *B*, *m*, and *n* are material parameters.

Rate Power Law Model

$$\sigma_y = A \overline{\varepsilon^p}^m \overline{\varepsilon^p}^n + B$$

where the parameters are same quantities as that of Power law.



Johnson-Cook Model

$$\sigma_{y} = (A + B\overline{\varepsilon^{p}}^{n}) \left( 1 + C ln \left( \frac{\overline{\varepsilon^{p}}}{\varepsilon_{0}} \right) \right) \left( 1 - \left( \frac{T - T_{room}}{T_{melt} - T_{room}} \right)^{m} \right)$$

where  $\sigma_y$  is the yield stress,  $\overline{\varepsilon}^p$  is the equivalent plastic strain,  $\overline{\varepsilon}^p$  is the current equivalent plastic strain rate,  $\varepsilon_0$  is strain rate of material characterization and A, B, C, m and n are material parameters. T,  $T_{room}$ ,  $T_{melt}$  are, respectively, the current, room, and melting temperatures of the material in absolute scale.

The following conditions should be noted for the Johnson-Cook model.

- T should be between  $T_{room}$  and  $T_{melt}$ . If  $T < T_{room}$ , T is set to  $T_{room}$ . If  $T > T_{melt}$ ,  $T = T_{melt} 0.01$ .
- $\dot{\epsilon}_{0>0}$  and  $\dot{\epsilon} \geq \dot{\epsilon}_{0}$ . If either condition is violated, the middle term in the above expression is set to 1.0.

Kumar Model

$$\sigma_{y} = B_{0} * sinh^{-1} \left[ \left( \frac{\varepsilon^{p}}{A} \right)^{1/n} e^{Q/(nRT)} \right]$$

where  $\sigma_y$  is the yield stress,  $\overline{\varepsilon^p}$  is equivalent plastic strain rate, Q is the activation energy, T is temperature, and A,  $B_0$ , and n are material parameters.

# Creep (MATVP, CREEP)

Creep is an important factor in elevated-temperature stress analysis. In SOL 400, creep is represented by a Maxwell model. Creep is a time-dependent, inelastic behavior, and can occur at any stress level (that is, either below or above the yield stress of a material). The creep behavior can be characterized as primary, secondary, and tertiary creep, as shown in (Figure 6-6). Engineering analysis is often limited to the primary and secondary creep regions. Tertiary creep in a uniaxial specimen is usually associated with geometric instabilities, such as necking. The major difference between the primary and secondary creep region. The creep strain rate is much larger in the primary creep region than it is in the secondary creep region. The creep strain rate is the slope of the creep strain-time curve. The creep strain rate is generally dependent on stress, temperature, and time.

The creep data can be specified in either an exponent form or in a piecewise linear curve.

$$\dot{\varepsilon}^c = \frac{d\bar{\varepsilon}^c}{dt} \tag{6-6}$$

Na Main Index



## **Viscoplasticity (Explicit Formulation)**

The creep (Maxwell) model can be modified to include a plastic element (as shown in Figure 6-7). This plastic element is inactive when the stress ( $\sigma$ ) is less than the yield stress ( $\sigma_v$ ) of the material. The modified model

is an elasto-viscoplasticity model and is capable of producing some observed effects of creep and plasticity. In addition, the viscoplastic model can be used to generate time-independent plasticity solutions when stationary conditions are reached. At the other extreme, the viscoplastic model can reproduce standard creep phenomena. The model allows the treatment of nonassociated flow rules and strain softening which present difficulties in conventional (tangent modulus) plasticity analyses.

It is recommended that you use the implicit formulation described in the following paragraphs to model general viscoplastic materials.





## **Creep (Implicit Formulation)**

This formulation, as opposed to that described in the previous section, is fully implicit. A fully implicit formulation is unconditionally stable for any choice of time step size; hence, allowing a larger time step than permissible using the explicit method. Additionally, this is more accurate than the explicit method. The disadvantage is that each increment may be more computationally expensive. There are two methods for defining the inelastic strain rate. The creep model definition option can be used to define a Maxwell creep model. The back stress must be specified through the field reserved for the yield stress in the MAT1 or other material definitions. There is no creep strain when the stress is less than the back stress. The equivalent creep strain increment is expressed as

$$\dot{\varepsilon}^{c} = A\overline{\sigma}^{m} \bullet (\dot{\varepsilon}^{c})^{n} \bullet T^{P} \bullet qt^{q-1}$$
(6-7)

and the inelastic deviatoric strain components are

$$\Delta \bar{\varepsilon}_{ij}^i = \frac{3}{2} \Delta \bar{\varepsilon}^i \frac{\sigma'_{ij}}{\bar{\sigma}}$$

where  $\sigma'_{ij}$  is the deviatoric stress at the end of the increment and  $\sigma_y$  is the back stress. *A* is a function of temperature, time, etc. Creep only occurs if  $\sigma$  sigma is greater  $\sigma_y$ .

One of three tangent matrices may be formed. The first uses an elastic tangent, which requires more iterations, but can be computationally efficient because re-assembly might not be required. The second uses an algorithmic tangent that provides the best behavior for small strain power law creep. The third uses a secant (approximate) tangent that gives the best behavior for general viscoplastic models.

As an example of the usage of MATVP for defining the creep behavior, see Creep of a Tube (Ch. 34) in the *Demonstration Problems Manual - Implicit Nonlinear*.



## **ANAND Solder Creep Model**

In the process of IC produce and package, solder is widely used to connect different chips or lines, its creep behavior has significantly influence of the IC performance and durability. ANAND model is widely accepted in IC industries to describe the creep behavior of solder materials. Enhanced capability of SOL 400 with ANAND creep model will benefit all current and potential Nastran users.

The Anand Solder model uses a single scalar internal variable representing deformation resistance (s), which denotes the averaged isotropic resistance to macroscopic plastic flow offered by the underlying isotropic strengthening mechanisms.

## **Specifying Creep Material Entries**

Each of the creep models described in this section can be selected with the MATVP bulk data entry. MATVP is the only form of creep data material input supported by SOL 400 when the advanced elements are used.

| Entry | Description  |
|-------|--|
| MATVP | Specifies viscoplastic or creep material properties to be used for quasi-static analysis in SOL 400. |

#### **Patran Materials Application Input Data**

To define creep behavior in Patran:

- 1. From the Materials Application form, set the Action>Object>Method combination to Create > Isotropic-or-Orthotropic-or-Anisotropic > Manual Input.
- 2. Click **Input Properties...**, and select **Creep** from the **Constitutive Model** pull-down menu and **MATVP** from the **Creep Data Input** pull-down menu.

Creep material models require the following MATVP material data via the **Input Properties** subform on the **Materials Application** form.

| Isotropic – Anisotropic –<br>Orthotropic | Description  |
|--|--|
| Coefficient                              | Specifies the coefficient, A.  |
| Exponent of Temperature                  | Defines temperature exponent.  |
| Temperature vs. Creep Strain             | References a material field of temperature vs. value. Overrides <b>Exponent of Temperature</b> if present. |
| Exponent of Stress                       | Defines stress exponent  |
| Creep Strain vs. Stress                  | References a material field of stress vs. value. Overrides <b>Exponent</b> of Stress if present.           |
| Exponent of Creep Strain                 | Defines creep strain exponent.   |
|  |  |

Figure 6-8



| Isotropic – Anisotropic –<br>Orthotropic | Description  |
|--|--|
| Strain Rate vs. Creep Strain             | References a material field of strain rate vs. value. Overrides <b>Exponent of Creep</b> Strain if present.  |
| Exponent of Time                         | Defines time exponent.   |
| Time vs. Creep Strain                    | References a material field of time vs. value. Overrides <b>Exponent</b> of Time if present.   |
| Back Stress                              | Defines the back stress for implicit creep   |
| ANAND                                    | The Anand solder material model which consists of a simple set of<br>constitutive equations for large, isotropic, viscoplastic problems.<br>(if this is used, next 9 fields for parameters of Material properties<br>must be filled with value.) |
| PREXF                                    | Pre-exponential factor. A, (s <sup>-1</sup> )  |
| ACTEN                                    | Activation energy. Q   |
| MULST                                    | Multiplier of stress.  |
| STNRT                                    | Strain rate sensitivity of stress. M   |
| SATCO                                    | Deformation resistance saturation coefficient. s,(MPa)   |
| STNSA                                    | Strain rate sensitivity of saturation. n   |
| HRCN                                     | Hardening constant. h0,(MPa)   |
| STNHR                                    | Stain rate sensitivity of hardening. A   |
| DEFRS                                    | Initial value of deformation resistance. s <sub>0</sub> , (MPa)  |

# Composite (PCOMP or PCOMPG)

Composite materials are composed of a mixture of two or more constituents, giving them mechanical and thermal properties which can be significantly better than those of homogeneous metals, polymers and ceramics.

Laminate composite materials are based on layering homogeneous materials using one of several methods. In order to define a laminate composite material, you must define the homogeneous materials that form the layers, the thickness of each layer, and the orientation angle of the layers relative to the standard coordinate axis being used for the model. The orientation is particularly important for orthotropic and anisotropic materials, whose properties vary in different directions. The material in each layer may be either linear or nonlinear. Tightly bonded layers (layered materials) are often stacked in the thickness direction of beam, plate, shell structures, or solids.





Figure 6-9 identifies the locations of integration points through the thickness of beam and shell elements with and without a composite formulation.

Note that when the COMPOSITE option is used, as shown in Figure 6-9, the layer points are positioned midway through each layer. When the COMPOSITE option is not used, the layer points are equidistantly spaced between the top and bottom surfaces. MSC Nastran Implicit Nonlinear performs a numerical integration through the thickness. If the COMPOSITE option is used, the trapezoidal method is employed; otherwise, Simpson's rule is used.



Figure 6-10 shows the location of integration points through the thickness of continuum elements. MSC Nastran Implicit Nonlinear forms the element stiffness matrix by performing numerical integration based on the standard isoparametric concept.



| * | * |
|---|---|
| * | * |
| * | * |
| * | * |

Figure 6-10 Integration Points through the Thickness of Continuum Elements

## **Specifying Composite Material Entries**

You specify the material properties and orientation for each of the layers. Additional stress and strain output is generated for each layer and between the layers.

The format of the **PCOMP** bulk data entry in the *QRG* is as follows:

| 1     | 2    | 3  | 4      | 5     | 6    | 7    | 8      | 9     | 10 |
|-------|------|----|--------|-------|------|------|--------|-------|----|
| PCOMP | PID  | ΖO | NSM    | SB    | FT   | TREF | GE     | LAM   |    |
|       | MID1 | Т1 | THETA1 | SOUT1 | MID2 | Т2   | THETA2 | SOUT2 |    |
|       | MID3 | Т3 | THETA3 | SOUT3 | etc. |      |        |       |    |

| Field  | Contents  |
|--------|---|
| PID    | Property identification number.   |
| Z 0    | Distance from the reference plane to the bottom surface.  |
| NSM    | Nonstructural mass per unit area.   |
| SB     | Allowable shear stress of the bonding material.   |
| FT     | Failure theory.   |
| TREF   | Reference temperature.  |
| LAM    | "Blank", "SYM", "MEM", "BEND" option.   |
| MIDi   | Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. |
| Ti     | Thicknesses of the various plies.   |
| THETAİ | 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 **PCOMPG** bulk data entry in the *QRG* is as follows:

#### **PCOMPG**

| 1      | 2       | 3    | 4   | 5      | 6     | 7    | 8  | 9   | 10 |
|--------|---------|------|-----|--------|-------|------|----|-----|----|
| PCOMPG | PID     | Z0   | NSM | SB     | FT    | TREF | GE | LAM |    |
|        | GPLYID1 | MID1 | Τ1  | THETA1 | SOUT1 |      |    |     |    |
|        | GPLYID2 | MID2 | Т2  | THETA2 | SOUT2 |      |    |     |    |

| Field | Contents   | Contents   |  |  |  |  |  |  |
|-------|--|--|--|--|--|--|--|--|
| PID   | Property identi  | Property identification number. (0 < Integer < 10000000)   |  |  |  |  |  |  |
| ZO    | Distance from telement thickness   | Distance from the reference plane to the bottom surface. (Real; Default = -0.5 times the element thickness.)   |  |  |  |  |  |  |
| NSM   | Nonstructural i  | mass per unit area. (Real)   |  |  |  |  |  |  |
| SB    | Allowable shear<br>Required if FT  | r stress of the bonding material (allowable interlaminar shear stress).<br>is also specified. (Real > 0.0)   |  |  |  |  |  |  |
| FT    | Failure theory. '<br>failure calculati   | Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed)   |  |  |  |  |  |  |
|       | "HILL" for the Hill theory.<br>"HOFF" for the Hoffman theory.<br>"TSAI" for the Tsai-Wu theory.<br>"STRN" for the Maximum Strain theory. |  |  |  |  |  |  |  |
| TREF  | Reference temp   | perature. (Real; Default = 0.0)  |  |  |  |  |  |  |
| GE    | Damping coeff  | icient. (Real; Default = 0.0)  |  |  |  |  |  |  |
| LAM   | Laminate Optio   | ons. (Character or blank, Default = blank).  |  |  |  |  |  |  |
|       | "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 <b>PSHELL</b> entry) are computed.   |  |  |  |  |  |  |
|       | "BEND"   | All plies must be specified, but only bending terms (MID2 on the derived <b>PSHELL</b> entry) are computed.  |  |  |  |  |  |  |
|       | "SMEAR"  | All plies must be specified, stacking sequence is ignored $MID1=MID2$ on the derived <b>PSHELL</b> entry and $MID3$ , $MID4$ , $TS/T$ , and $12I/T**3$ terms are set to zero). |  |  |  |  |  |  |
|       |  |  |  |  |  |  |  |  |



| Field   | Contents   |   |  |  |  |  |  |
|---------|--|---|--|--|--|--|--|
|         | "SMCORE"   | All plies must be specified, with the last ply specifying core properties and<br>the previous plies specifying face sheet properties. The stiffness matrix is<br>computed by placing half the face sheet thicknesses above the core and the<br>other half below with the result that the laminate is symmetric about the<br>mid-plane of the core. Stacking sequence is ignored in calculating the face<br>sheet stiffness. |  |  |  |  |  |
| GPLYIDi | User-defined G<br>unique with res<br>starting with th  | User-defined Global (External) Ply ID. 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. (Integer > 0)   |  |  |  |  |  |
| MIDi    | Material ID of<br>1 at the bottom<br>(Integer > 0 or   | the various plies. The plies are identified by serially numbering them from layer. The MIDs must refer to MAT1, MAT2, or MAT8 bulk data entries. blank, except MID1 must be specified.)   |  |  |  |  |  |
| Ti      | Thicknesses of   | the various plies. (Real or blank, except T1 must be specified.)  |  |  |  |  |  |
| THETAİ  | Orientation ang<br>element. (If the<br>side 1-2 of the<br>the bottom laye<br>element coordin | gle of the longitudinal direction of each ply with the material axis of the material angle on the element connection entry is 0.0, the material axis and element coincide.) The plies are to be numbered serially starting with 1 at tr. The bottom layer is defined as the surface with the largest -Z value in the nate system. (Real; Default = $0.0$ )  |  |  |  |  |  |
| SOUTi   | Stress or strain   | output request. (Character: "YES" or "NO"; Default = "NO")  |  |  |  |  |  |

Using the material properties for each of the lamina with conventional shell elements, MSC Nastran calculates the equivalent **PSHELL** and **MAT2** entries generated as shown in Figure 6-11. If nonlinear material behavior including progressive failure occurs, then the advanced elements should be activated using the **PSHLN1** bulk data entry.





Figure 6-11 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 Sb, and the failure theory method FT on the **PCOMP/PCOMPG** entries.

As shown in Figure 6-11, 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=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. The ID numbers of the MAT2s are important. The MID1 material has an ID in the range of 100000000 to 199999999. The MID2 material has an ID in the range of 200000000 to 299999999, etc. These ranges are used to inform MSC Nastran that the material is part of a composite analysis. If you are using the equivalent properties in a future analysis instead of using the PCOMP/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 **MAT2**s, 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 usage of **PCOMP**, see Laminated Strip under Three-point Bending (Ch. 7) in the *Demonstration Problems Manual - Implicit Nonlinear*.

# Cohesive Zone Modeling (MCOHE)

MSC Nastran has a library of interface elements, which can be used to simulate the onset and progress of delamination. These elements are defined with the CIFHEX, CIFPENT, CIFQDX, and CIFQUAD bulk data entries. The constitutive behavior of these elements is expressed in terms of tractions versus relative displacements between the top and bottom edge/surface of the elements (see Figure 6-12).



Figure 6-12 3-D Linear Interface Element

Considering a 3-D interface element, the relative displacement components are given by one normal and two shear components, expressed with respect to the local element system:

$$v_n = u_1^{top} - u_1^{bottom}$$

$$v_s = u_2^{top} - u_2^{bottom}$$

$$v_t = u_3^{top} - u_3^{bottom}$$
(6-8)

Based on the relative displacement components, the effective opening displacement is defined as:

$$v = \sqrt{v_n^2 + v_s^2 + v_t^2}$$
(6-9)

# **Progressive Composite Failure**

MSC Nastran provides two methods to model the failure of composite materials. The first is based upon Marc technology which is activated by the MATF bulk data entry and is discussed here. The second is provided by e-Xstreme and is activated using MATDIGI. Failure is indicated by the failure criteria described in the previous section. When failure occurs, the element stiffness is degraded. MSC Nastran offers two



different methods for the material degradation as described below. While MSC Nastran allows up to three failure criteria to be used for failure index calculation, only the first criterion is used for progressive failure. The material will not heal; the damaged elements keep the degraded properties after unloading.

#### Model 1 – Selective Gradual Degradation

This model uses a selective degradation of the moduli depending on failure mode. The moduli are decreased gradually when failure occurs. Within an increment, it attempts to keep the highest failure index less than or equal to one. Whenever a failure index F larger than one occurs, stiffness reduction factors  $r_i$  are calculated based upon the value of the failure indices. The incremental contribution to the total reduction factor is calculated as

$$\Delta r_i = -(1 - e^{1 - F}) \tag{6-10}$$

This is done differently for different failure criteria as described below. Six such reduction factors are stored and updated. They are then used for scaling the respective material modulus according to

$$E_{11}^{new} = r_1 E_{11}^{orig} \tag{6-11}$$

$$E_{22}^{new} = r_2 E_{22}^{orig} \tag{6-12}$$

$$E_{33}^{new} = r_3 E_{33}^{orig} \tag{6-13}$$

$$G_{12}^{new} = r_4 G_{12}^{orig} \tag{6-14}$$

$$G_{23}^{new} = r_5 G_{23}^{orig} \tag{6-15}$$

$$G_{31}^{new} = r_6 G_{31}^{orig} \tag{6-16}$$

The Poisson's ratios are scaled in the same way as the corresponding shear modulus.

For the maximum stress and maximum strain criteria the reduction factors are calculated separately from each separate failure index:  $r_1$  is calculated from the first failure index as given by equation 6-11 above,  $r_2$  is calculated from the second failure index from equation 6-12 etc. Thus, there is no coupling of the different failure modes for these criteria.

#### Model 2 - Selective Immediate Degradation

This model uses selective degradation just as Model 1, but the stiffness is abruptly decreased. As soon as failure is indicated, the stiffnesses are set to  $a_1$  – the residual stiffness factor. The same rules as in Model 1 for how the different factors are defined depending on the type of failure is applied here.

The different options are flagged through the MATF bulk data entry.



## **Micro-mechanics Material Models (MATDIGI)**

MSC Nastran SOL 400 integrates with e-Xstream Digimat Material Modeling System. This allows the user to give the material properties on a component basis in a composite material. This is activated through the **MATDIGI** bulk data entry.

#### Interface

Digimat-CAE/MSC Nastran SOL 400 contains the material library containing the Digimat capabilities and the required interfaces in order to be linked with the MSC Nastran SOL 400 Finite Element (FE) solver. Linking MSC Nastran SOL 400 libraries with Digimat-CAE/MSC Nastran SOL 400 gives the user access to all linear and nonlinear small-strain material models available in Digimat for FE small-strain analyses, just like any other MSC Nastran SOL 400 material models. It also enables to take into account fiber orientation computed by an injection molding code. The interface thus allows to model the impact of the injection process on the structural behavior of composite parts. Note that not all the available Digimat material models can be used with the Digimat-CAE/MSC Nastran SOL 400 interface.



116 MSC Nastran Getting Started Guide Progressive Composite Failure



# 7 Contact

- Introduction to Contact 118
- Contact Types 118
- Contact Definition Method 123
- Contact Bodies 124
- Defining Contact Interactions 131
- Contact in MSC Nastran 131
- Contact Settings for a SOL400 Analysis 132
- Contact Considerations 134
- Contact Parameters 135



# Introduction to Contact

The analysis of a finite element model (FEM) with contact bodies, interacting with each other is called contact analysis. It is the analysis of contact bodies (deformable or rigid) interacting with each other. Many engineering problems involve contact between two or more components. In these problems a force normal to the contacting surfaces acts on the two bodies when they touch each other. If there is friction between the surfaces, shear forces may be created that resist the tangential motion (sliding) of the bodies. The aim of contact simulations is to identify the areas on the surfaces that are in contact and to calculate the contact pressures generated.

Contact can be a useful tool when performing FEA simulations. It allows the interaction between multiple bodies without adding additional elements to the model. By adding the contact condition to a model correctly it is possible to create a model that more realistically represents reality. However, by adding contact to a model, it might cause the analysis not to complete because of the added computational complexity and the number of things the analyst has to take into account.

In a finite element analysis (FEA), contact conditions are a special class of discontinuous constraint, allowing forces to be transmitted from one part of the model to another. The constraint is discontinuous because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The analysis has to be able to detect when two surfaces are in contact and apply the contact constraints accordingly. Similarly, the analysis must be able to detect when two surfaces separate and remove the contact constraints.

By default, MSC Nastran does not assume contact exist between all bodies in a model, simply because of the computation cost involved in checking for contact between every element and every other element in a model. This can however be specified, at the expense of the run times, but another reason why this is not specified by default is because different contact conditions such as varying friction values might exist throughout the model and therefore the analyst needs a means to specify and control it.

Contact in a model is applied mathematically and unless otherwise specified, MSC Nastran defaults will be used that are suitable for general contact cases. Default parameters will in most cases be sufficient, however, it might cause problems under more complex or less common scenarios, or induce unnecessarily longer solving times if the analyst doesn't understand these parameters and set them incorrectly.

Contact can be deformable-deformable or rigid-deformable.

# **Contact Types**

Two types of contact interaction exist between two bodies namely *glue* and *touch* contact. Although there are variations in how these contact interactions are applied depending on the software used, these remain the two main contact interactions.

**Touching Contact:** In this case, both glue and touching contact can be separate



|                  | 0 0   |
|------------------|---|
| Full Contact     | <ul> <li>No separation, no sliding</li> </ul>                             |
|                  | <ul> <li>SOL 400 calls it Permanent Glue Contact</li> </ul>               |
| Sliding contact  | <ul> <li>No separation, sliding</li> </ul>                                |
|                  | <ul> <li>SOL 400 does not have it</li> </ul>                              |
| Breaking contact | <ul> <li>Full, but may separate if force/stress &gt; threshold</li> </ul> |
| Linear contact   | <ul> <li>Glued elastic stiffness</li> </ul>                               |

• Glued Contact: In this case, the bodies are glued together

The difference between glue and touching is glue contact does not allow sliding, but touching does.

# **Touching Contact**

In touching contact bodies can separate

- Touch contact is a valuable tool to use in FEA models although it comes with the cost of a much longer analysis time and it does require additional considerations to ensure that the model solves successfully.
- With this method of contact, the parts in the model can move as it is allowed to by the constraints on the body, however with touch contact active between two bodies the additional condition is that the one body is not allowed to penetrate the other. With the movement of the body allowed it is therefore important to consider the position of one body relative to another.

## **Glued Contact**

Glue contact is an ideal option to use in a model with multiple parts in a structure. It allows two bodies to be fixed at the contact surface without having the elements of both bodies be the same size and attempting to have the nodes on the two surfaces match.

It can be applied as a linear contact which means that two element faces remain in contact no matter what happens with the structure around it. As long as the contact condition is detected between two bodies, this will not cause for a longer analysis time and it should not cause further struggle with the model.

Glue contact can also be applied with certain conditions. For instance, it can be specified that if the stress on the contact surface exceeds a specified stress, the contact should be released and for the remainder of the analysis be treated as a broken contact.

MSC Nastran internally generates MPC equations to represent the glued contact. MPCs are generated by MSC Nastran internally to represent the glued contact. The MPCs generated to model glued contact can be written during the analysis in a punch (\*.pch) file. You can also request for the MPC equations be output to a punch file so that the you can visualize what is glued and what is not by plotting MPCs.

These MPCs can then be directly imported into the Patran database to provide a visual reference to the glued contact. The MPC punch file is created with the following Case Control Command:



NLOPRM MPCPCH = BEGN



#### **Permanent Glue Contact**

This is a special case of contact, where the initial configuration is used to determine the contact constraint, and these contact constraint should not change throughout the analysis. It is designed to help users quickly assemble components with dissimilar meshes. Nodes or segments which are not initially in contact do not come into contact, and in fact may penetrate the model. The constraint is a glue type, meaning there will be no relative normal or tangential displacement. The bodies will never separate.

Permanent glued is designed to help users:

- To quickly assemble components with dissimilar meshes and connect dissimilar meshes
- Help in simple assembly modeling when no other contact occurs.

It is applicable to SOL 101, 103, 105, 107, 108, 109, 110, 111, 112, SOL 200, and SOL 400. Permanent glued contact is activated if the BCTABLE or BCTABL1 that is referenced in the first Loadcase (SOL 100\*) or in the first Step (SOL 200) has a value of IGLUE greater than zero for all contact body pairs.

If you require conventional (general) contact for the complete simulation, but permanent glued contact is invoked, enter bulk data BCPARA, 0, NLGLUE, 1 to deactivate the permanent glue in a subsequent step. Because glued contact is very useful in assembly modeling problems encountered in engineering practice, several special cases are considered as well.

Some characteristics of the permanent glue contact are as follows:

- It should not be used in models that experience large rotations.
- Bodies should be in contact initially since contact detection is performed only in the beginning of the analysis
- A linear solution. Permanent contact constraint MPC equations are used. No nonlinear increments or iterations involved

#### **Step Glue Contact**

Step glued contact is available only for SOL 400. Step Glued contact is activated using a value a negative value of IGLUE for each contact pair. It is similar to Glued Contact, there are two conditions.



The contact status is checked at the beginning of the step, and those nodes or segments that are in contact will remain in glued contact for the entire step. The constraints will change due to large rotations.

Furthermore, if a large tensile force or stress developed over the interface in the current loadcase, no separation would occur for these regions which are initially in contact. Performing an unglue of breaking glue would also not be enforced during the step for these regions. This may be successfully used to model the union of dissimilar meshes, where at a later time one wanted to separate the bodies (e.g., opening of a door).

• When using step glue conventional contact occurs for the nodes/segments of the body which are not in contact at the beginning of the step. That means when they come into contact, they will glue, but they may separate within the same step

#### **General Glue Contact**

Two bodies may come into contact and separate at any point in the simulation, but when in contact, there is no relative sliding. The fundamental constraint is no relative normal displacement or tangential displacement when bodies are in contact. General contact is available only in SOL 101, SOL 400, SOL 600, and SOL 700.

One can consider this equivalent to two surfaces that have infinite friction. The word *glued* only refers to the constraint on the tangential behavior. Bodied that are in glued contact may lose contact if the separation (force or stress based) separation criteria is exceeded, due to unglue or breaking glue.

Some characteristics of the general glue contact are as follows:

- Simulates a glued joint
- Initially, the bodies do not have to be in contact. They can come in contact during the analysis and become glued
- After being glued together, the bodies can separate again or stay glued based on user specified criteria uch as breaking glued contact
- Just like touching contact, the general glued contact utilizes the nonlinear solver which is an incremental and iterative process
- Because contact status can change during the analysis in general glued contact, it is an incremental and iterative process.
- Separation force is infinite FNTOL = 1E20

#### **Breaking Glue Contact**

In engineering problems that involve delamination, it is often useful to indicate that two surfaces are glued together, but may separate if a certain stress level is reached. The simplest is based upon a normal stress (preferred) or a force condition. For problems like tape peeling, it is useful to include both the normal and shear stress condition. This can be invoked by JGLUE on BCTABLE or BCONPRG:

- 0: glued contact nodes will stay in contact. Default.
- 1: to invoke the standard separation behavior
- 2: breaking glued with a breaking criterion



BKGL, keyword for breaking glued:

- BGST, maximum tangential stress (default=0.0)
- BGSN, maximum normal stress (default=0.0)
- BGM, the first exponent associated with tangential stress (default=2.0)
- BGN, the exponent associated with normal stress (default=2.0)

#### **Summary of Glue Contact**

In summary, the glue condition between bodies can be defined via BCTABLE or BCONPRG through the IGLUE keyword as follows:

| IGLUE Keyword | Description   |
|---------------|---|
| 0             | No gluing   |
| 1             | Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.   |
| 2             | Activates a special glue option to insure that there is no relative tangential and<br>normal displacement when a node comes into contact. An existing initial gap or<br>overlap between the node and the contacted body will not be removed, as the<br>node will not be projected onto the contacted body. To maintain an initial gap,<br>ERROR should be set to a value slightly larger than the physical gap. |
| 3             | Ensures full moment carrying glue when shells contact. The node will be projected onto the contacted body.  |
| 4             | Insures full moment carrying glue when shells contact. The node will not be<br>projected onto the contact body and an existing initial gap or overlap between the<br>node and the contacted body will not be removed, as the node will not be<br>projected onto the contacted body.   |

In SOLs 101 and 400, if contact is initially not true set NLGLUE on BCPARA to 1. For SOL 400 with a mixture of glued and non-glued bodies, BCPARA, 0, NLGLUE, 1 must also be used

## **Cohesive Contact**

This is a special case of glued contact. In the modern industries, the product structure like airplane, automobile, and so on, is becoming more and more complicated with many parts. The assembly process may be done through a variety of processes such as rivets, bolts, spot weld, seam welds, or adhesives. In the numerical simulations of the assemblies of many parts into a single structure, it is often too costly to model each one of these discrete connectors, and the glued contact capability provides an effective and efficient way to simplify and reduce the computation costs.



While this method is easy to use, however, it often results in too stiff of a structure because effectively the connection is rigid. To alleviate this, a flexible glued contact capability is available. With cohesive glued

contact, it is necessary to model the detailed connectors but provides the stiffness of the connectors in the glued contact.

There are many cases in which parts are connected using either discrete entities, such as rivets or bolts, or a spot or seam weld. It is often too expensive to model these discrete entities so the glued approach is often used to model these connections. While easy to use, this method is effectively a rigid connection and results in behavior that is too stiff.

To compensate for this undesirable behavior, instead of entering a non-default constant penalty factor for the segment-to-segment contact algorithm, you can use cohesive contact. It allows you to apply a separate finite stiffness in the normal and tangential directions.



The advantages of cohesive contact are:

- Provides a user controllable soft contact mechanism
- Reduces computational costs and improves accuracy of assembly analyses
- Simplifies analysis of bonded Joints: Replace detailed model with simplified model
- Implemented for small sliding segment to segment contact
- User specifies either stiffness or contact stress vs. relative displacement on contact interaction menu.

# **Contact Definition Method**

Before discussing contact in MSC Nastran it should be noted that there are two possible methods of specifying contact in MSC Nastran. The two contact methods are:

- Contact table method (BCTABLE): With this method of specifying contact, it is difficult to define the contact interactions between many bodies in the contact table without making any mistakes. Hence, a new method (contact pairs method) was introduced
- Contact pairs method: In this method, the contact pairs can be created directly and instead of specifying contact parameters for each pair, contact interaction can be defined and simply referenced for each pair. This makes it easier to define the contact condition for a large amount of contact bodies.



The following figure shows the process of defining contact with both the contact table method as well as the contact pairs method.



Figure 7-1 Contact Nastran

Both methods of applying contact will have the same outcome but with the contact table it is easier to define contact for a small number of contact bodies, whereas the contact pairs method will be easier to define contact for a large number of contact bodies.

Both methods have their own merits. Any one of the two methods can be used for any model although it is recommended to use the contact table for a model with a small number of contact bodies and contact pairs for a model with a large number of contact bodies. Contact can be deformable-deformable or rigid-deformable.

## **Contact Bodies**

These types of contact can be applied between various types of bodies in various combinations. Contact bodies can be created from:



- Deformable body: It is defined by element IDs or element properties
- Rigid body: It is defined by geometry (curves and surfaces) or 4-node patches



## Deformable Contact Bodies (3D/2D/1D)

A deformable contact body is a normal FEM body that can deform and experience stresses while in and due to contact. For any deformable contact body with a non-square shape or a shape that is not exactly captures by the elements, analytical edge discontinuities can be activated when creating the contact body. Analytical edge discontinuities is a function that Nastran uses to determine the possible true curvatures of the edge that could not be captured accurately by the elements and uses this shape when determining contact.

An example of this is two circular objects in contact. Since the circular objects will consist of a number of straight lines or faces it is difficult to have an accurate distributions of the forces acting between the two bodies. The contact might be applied only to some of the faces or have inconsistencies in the contact forces. Although this might affect the results somewhat, this will also make it difficult for Nastran to correctly detect the contact or fail to detect the contact at all. With analytical edge discontinuities Nastran uses the change in angle between the two element faces to construct a closer to circular contact surface.





## **Creating a Deformable Body**

Here, we discuss about creating a deformable body with BCBODY1. It is used in contact pair Flowchart. With BCBODY1, the contact body contain the following:

- BCBODY dimension (2D for plane analysis or 3D)
- ID of BSURF or BCPROP for list of element or properties
- Reference of a BCBDPRP entry that contain some contact body physical datas
  - Friction coefficient (Friction will be discussed later)
  - Contact body Thermal datas for thermal chained or coupled analysis (not discussed in this document)
  - Specific info if analytical contact is used to increase accuracy of the geometrical boundary of the contact body

Same card to define Rigid Body.





#### **Defining Body Contact via BSRRF**

#### BSURF

**Contact Body or Surface** 

Defines a contact body or surface by Element IDs.

#### Format:

| 1     | 2     | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|-------|-------|-------|-------|-------|-------|-------|-------|-------|----|
| BSURF | ID    | ELID1 | ELID2 | ELID3 | ELID4 | ELID5 | ELID6 | ELID7 |    |
|       | ELID8 | ELID9 | etc.  | 1     |       |       |       |       |    |

#### Alternate Format:

| BSURF | ID    | ELID1 | THRU  | ELID2 | BY   | INC |  |  |
|-------|-------|-------|-------|-------|------|-----|--|--|
|       | ELID3 | THRU  | ELID4 | BY    | INC2 |     |  |  |

Defining Body Contact via BCPROP



#### Property-based MSC Nastran entry:

- BCPROP: Defines a contact body to Element Property (Referenced by BSID in BCBODY)
- More easy than BSURF

| 1      | 2   | 3   | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|------|-----|-----|-----|-----|-----|----|
| BCPROP | ID  | IP1 | IP2  | IP3 | IP4 | IP5 | IP6 | IP7 |    |
|        | IP8 | IP9 | etc. |     |     |     |     |     |    |

#### Example

| BCPROP | 1     | 101     | 20     | 301   |               |   |  |
|--------|-------|---------|--------|-------|---------------|---|--|
|        | \$ DI | EFORM B | ODY CO | NTACT | LBC SET: CLIP |   |  |
|        | BCBO  | DDY 1   | 3      | D     | DEFORM 2      | 0 |  |
|        | BCPI  | ROP 2   | 1      | 2     |               |   |  |

# **Rigid Contact Bodies (3D/2D)**

A rigid contact body is a useful method in which the body is assumed to remain rigid during the simulation which means no calculations are performed for the inside of the body. This means you can have a body in the analysis which affects the rest of the structure without adding significant time to the analysis.

- Rigid bodies can be modeled with geometry or finite elements
  - Curves for 2D contact (NURBS2D): allows for 3 DOF: UX, UY, and ROTZ
  - Surfaces for 3D contact (NURBS): allows for 6 DOF: UX, UY, UZ, ROTX, ROTY, and ROTZ



- 2D elements (PATCH3D), 3/4-node patches
- It is recommended to use Bezier or NURBs



- Continuity of the normal vector along the surface
- A mathematical description
- Robustness of the Contact Algorithm
- It can be velocity/position/load driven
- It can change shape (growth factor)
- It can have initial velocity to save CPU at start of analysis and touch deformable bodies

The rigid contact body is applied to add the required stiffness to the model without adding additional solving time. Since a rigid body is simply geometry that is specified as a body, there are no nodes on the body to either constrain or apply a load to that body.

For rigid contact bodies the approach velocity function is available to assist the initial contact to be detected in the model. With this a direction is specified for which Nastran will move the rigid body in this direction until contact between rigid and deformable bodies are detected before a load is applied, thereby ensuring that the contact condition is met at the start of the analysis.

When creating a rigid contact body, it is important to check the inward normals of the contact body. This will show an arrow that points in the direction of the solid part of the rigid body and it is therefore very important that the arrow points in the correct direction.

Creating a Deformable Body





# **Control of Rigid Body**

Control of rigid body with BCBODY or BCRIGID. Rigid bodies can be stationary (default) or moved in space. There are three different methods of controlling rigid body movements. The input is done via CONTROL entry:

- 0: velocity controlled
- 1: position controlled
- Positive Integer (grid point location): load controlled



- APPROV control the initial body approach (increment 0)
- GROW control the growing changing of a rigid body (shrink or expand)



# Contact in MSC Nastran

| Linear: SOL101                     | <ul> <li>3D analysis only</li> <li>Small deflection theory</li> <li>Sliding neglected</li> <li>All aspects of the simulation are 'linear' with the exception structural contact</li> <li>Glued</li> </ul>   |
|------------------------------------|---|
| SOL101/103/105/108/109/11<br>1/112 | <ul> <li>Permanent Glued Contact is a special case of glued contact</li> <li>Primarily used to join 2 dissimilar meshes</li> <li>Contact must initially be true (bodies should be in contact initially)</li> <li>When edges or grids are to be glued, gluing can also be done for the rotational DOFs (Moment Carrying Glue)</li> <li>Permanent contact constraint MPC equations are used. No nonlinear increments or iterations involved.</li> </ul> |
| Nonlinear: SOL 400                 | <ul> <li>Touch/glue</li> <li>Large displacement: allows sliding between element edges/faces</li> <li>Large deformation</li> <li>2D (axi, plane stress/strain), 3D analysis</li> </ul>   |

In MSC Nastran, there are:

- Two algorithms of contact are available
  - Node-to-segment
  - Segment-to-segment
- Two friction models are supported
  - Bilinear coulomb friction
  - Bilinear shear friction
- Two type of contact behavior
  - Touching (including auto contact)

# **Defining Contact Interactions**

When defining the contact interaction between two contact bodies, whether with a contact table or contact pairs, some contact properties can be selected to help with the contact detection.



- Primary and secondary option flag: It is a tool that specifies where the contact is applied to. In this option select the A=1 for all 3D contact bodies, which will specify that contact should occur on the outside surface of the contact body, whereas for 1D and 2D contact bodies some other options are available. These include options such as which side of a shell element should be in contact or whether the edges of a 1D element should be included.
- Coordinate modification: It is a useful tool that will allow MSC Nastran to move nodes (within a tolerance distance) to avoid some contact problems that might occur. Two options are available here with the combination of both options available as the third.

In the case of stress free initial contact, nodes are moved in the beginning of the analysis to ensure that the analysis will start with the nodes in contact. For this option, the nodes will be moved if either the nodes are already penetrating the contact bodies or a small gap is exist between the two bodies. The nodes will then be moved (within the tolerance distance) until the nodes are aligned.

# Contact Settings for a SOL400 Analysis

When creating a SOL400 analysis, under the solver control settings or in the BCPARA Nastran entry, the types of contact detection specified with MSC Nastran are:

- Node-to-segment contact
- Segment-to-segment

## Node-to-segment (NTS) Contact

Node-to-segment (N2S or NODESURF) contact is the default contact detection type specified with MSC Nastran. It uses the paradigm of a secondary node interacting with a primary patch—that is, it works on a primary-secondary basis with one body being defined as the primary and the other as the secondary.

- This is a mature capability, but has some limitations
- Continuous displacement fields, but discontinuous stress (pressure) fields (limited using analytical contact body definition)
- Available for standard linear contact
- The rule for the contact is that the nodes of the secondary body cannot penetrate the element faces of the primary body.

#### **Limitations with Node-to-Segment Contact**

A common problem is that for a large difference in element size an incorrect primary-secondary configuration is selected which will mean that nodes from the primary will penetrate the secondary body and cause incorrect contact results since the contact rule has not been broken. So, important to always choose the body with the smaller element size to be the secondary body. In cases where this might still be a problem, there is the option to select double sided contact. This means that both bodies will be checked for contact to avoid this problem. This will the result in the contact condition will be checked twice per calculated increment increasing the analysis time significantly.



# Segment-to-Segment (STS) contact

The segment-to-segment (S2S or SEGTOSEG) contact detection method results in easier contact detection. The primary-secondary concept does not exist with this method and both bodies are checked for contact. This method adds additional points to the element face. It also checks the contact between these point and thus allows for a larger difference in element size on the contact surface.

- It is the new method of contact and is also the preferred method.
- It uses contact segments defined on both sides of the contact interaction
- The primary-secondary notion disappears
- Better precision in the contact displacement and stress fields
- The preferred method for future evolution
- Not available with all functionality (e.g. enhanced linear contact with LINCNT)

For models such as these the large displacements option (LGDISP Nastran entry) should be used along with the contact condition. The large-sliding option can be used for any model with large or small-sliding while the contact main input

| Contact bodies | BCBODY1            |
|----------------|--------------------|
|                | BSURF              |
|                | BCPROP             |
|                | BCHANGE (optional) |
| Parameters     | BCPARA,            |
|                | BCOMPRG            |
|                | BCOMPRP            |
|                | BCBDPRP            |
| Interaction    | BCTABLE1           |
|                | BCONECT            |

With segment-to-segment contact comes two options:

- Small-sliding segment-to-segment: For models where the contact bodies do not move relative to each other, such as the bending of a plate around a cylinder the small-sliding option is sufficient since the same elements will be in contact during the analysis. Small-sliding option will only be sufficient for a model with small-sliding contact.
- Large-sliding segment-to-segment contact: For a model with large relative movement between elements in contact, such as in roller bending, the elements in contact have to be redefined for every increment and therefore the large-sliding option has to be used.



# **Contact Considerations**

When running an analysis that contains contact, whether it is glue or touch contact, there are certain considerations to be made for the model itself in order to ensure that the contact condition is applied correctly.

- Always start by using a small version of the model in order to test whether the model is set up correctly. This applies to models with contact especially since touch contact will significantly increase the solving time, which will result in an unnecessary loss of time if an analysis fails or produces incorrect results due to an error with the contact condition.
- Although it is theoretically possible to have elements on the contact surface that vary greatly in size, this is not always the case. Some errors have been found in detecting the contact correctly when the size of the element faces on the contact surface varied too greatly.

This is because you are trying to have contact between one surface with element on the other surface. For example: fifty contact points with another surface with only five. Although you can find the contact, it is possible to miss a few of elements when checking the contact. The different sizes of elements can be further evaluated to find the required element size for the contact surface. However a good starting point is to have a maximum of five elements on one surface in contact with one element on the other surface.

• A possible way to decrease the size of the elements on the contact surface without increasing the total amount of elements significantly is to separate the section in contact from the rest of the body and only increase the amount of elements in that section and apply self-contact to that body. This will allow the element size to gradually reduce from the one body to the next across the contact surface.

| Possible Conta | Resolving the Contact problems |  |  |  |  |  |  |  |
|----------------|--------------------------------|--|--|--|--|--|--|--|
|                |                                |  |  |  |  |  |  |  |
|                |                                |  |  |  |  |  |  |  |
|                |                                |  |  |  |  |  |  |  |
|                |                                |  |  |  |  |  |  |  |
|                |                                |  |  |  |  |  |  |  |
|                |                                |  |  |  |  |  |  |  |

• When working with load controller rigid contact bodies, especially with geometries such as a wheel, it is important to remember that the body will be treated as rigid and therefore the contact surface will not increase due to deformation. In such cases the deformable contact body might require even smaller element faces on the contact surface to ensure that the contact condition is detected correctly and a penetration condition for the rigid body into the deformable body is avoided.




Another case is when applying a linear motion to a rigid body for a model where the rigid body is moved over a deformable body and a load being applied at each step after the body is moved. In order to select an appropriate distance to move in each step (by choosing the step size) it is important to select a distance small enough that the rigid body does not move into the deformable body due to the deformable body.



Although having smaller time steps will increase the number of steps to solve for and therefore increase the total analysis time, having too large steps will either cause the analysis to fail or have an even longer analysis time in order to solve the penetration condition. It is therefore important to choose a number of steps for the analysis by considering the distance that the rigid body will move in each step as well as the size of the elements on the contact face.

Applying contact to a model correctly can therefore allow you to decrease the analysis time by reducing the total amount of elements. Alternatively it is possible to create a more realistic model by including a realistic interaction between bodies in your model.

## **Contact Parameters**

Contact parameters defines options for detecting and handling contact



- Global contact parameters: BCPARA
- Contact Pairs Geometrical properties: BCONPRG
- Contact Pairs Physical properties: BCONPRP
- Contact Body parameters: BCBDPRP



# 8 Constraints

- Introduction 138
- Constraints 138
- Enforced Motion Constraints (SPCD and SPCR) 139
- Applying Constraints 139
- Multipoint Constraints (MPC) 140
- Static Loads 141
- CWELD/CFAST/CSEAM Element Enhancements 142



## Introduction

This chapter describes the procedures for applying loads and constraints which can be used in SOL 400 models. Each type of load or constraint is described on how it is used in finite element modeling and in the MSC Nastran input. Further information on how the various types of loads and constraints are used can be found in standard finite element text books, and further information on the MSC Nastran input format can be found in the *MSC Nastran Quick Reference Guide*.

### Constraints

In static analysis, the rigid body modes must be restrained in order to eliminate singularity of the stiffness matrix. The required constraints may be supplied with single point constraints, multipoint constraints, or free body supports. If free body supports are used, the rigid body characteristics will be calculated and checked for the sufficiency of the supports.

Boundary conditions are imposed in the form of constraints on selected degrees of freedom on the model. 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.

Other than single-point constraints, MSC Nastran provides a method for creating linear constraint relationships between several degrees of freedom known as multiple-point constraints or MPCs.

### Single-Point Constraints (SPC and SPC1)

A <u>Single-Point Constraint (SPC)</u> is a constraint that is applied to a single degree of freedom, which can either be 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:

- To constrain a structure.
- 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).
- To remove degrees of freedom that are very weak and coupled to the structure. For example, this condition can arise in rotation about an axis 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).

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 8-1, that permit unrestrained motion at point G in any direction perpendicular to the axis of the rods.





## Enforced Motion Constraints (SPCD and SPCR)

An enforced motion constraint is used to apply a prescribed motion at a grid point, which may be either a component of motion at a grid point or the displacement of a scalar point. SPCD and SPCR define an enforced displacement value for static analysis and an enforced displacement, velocity or acceleration in dynamic analysis. SPCD provides the final total displacement, velocity, or acceleration at the end of the current loadcase, and SPCR determines the incremental displacement, velocity, or acceleration during the current loadcase, relative to the value of the previous loadcase. For details on SPCD and SPCR refer to *MSC Nastran Nonlinear (SOL400) User's guide.* 

## **Applying Constraints**

Once you have designed a model, constraints are added that forces selected portions of your model to remain fixed or to move by a specified value. These constraints can be either:

- Single Point Constraints
- Multipoint Constraints

## **Single Point Constraint**

A Single Point Constraint (SPC) is constraint on a single degree of freedom. It assigns a zero or nonzero value to a single degree of freedom. It can be expressed as:

$$\delta_i = u$$

(8-1)



where u is the value of the prescribed displacement on the degree of freedom  $\delta_i$ . The case of u = 0 is the most common case, and is often used as a boundary condition, to "fix" or "ground" the movement of a point in a certain direction.

Since the value of  $\delta_i$  is known, one could, in principle, eliminate the specified degree of freedom from the

other degrees of freedom to be solved for as unknowns. This would reduce the size of the system of equations to be solved, but on the other hand it would take time to perform the elimination, and this approach adds complexity to the code.

SOL 400 uses a different technique. A number which is large compared to the stiffness coefficients (say, for discussion,  $10^{20}$ ) is added to the diagonal term  $K_{11}$  of the equation for the degree of freedom to be

constrained. Also, if the degree of freedom is to be constrained to a nonzero value u, then  $\mathbf{u} \times 10^{20}$  is added to the right hand side of the modified equation. This modified equation is now:

$$K_{i1}\delta_1 + \dots + (K_{ii})10^{20}\delta_i + \dots + K_{in}\delta_n = F_i + (u \times 10^{20})$$
(8-2)

Assuming all  $K_{ij}$  to be small with respect to  $10^{20}$ , the solution of the system of equations is obtained with negligible error.

The modified system of equations remain well conditioned. The value used by SOL 400 for the large number is  $10^{10}$  times the largest stiffness coefficient found on the diagonal of the stiffness matrix.

## Multipoint Constraints (MPC)

The multipoint constraint, or MPC entry, provides the capability to model rigid bodies and to represent other relationships which can be treated as rigid constraints. The MPC entry provides considerable generality but lacks user convenience. Specifically, the user must supply all the coefficients in the equations of constraint defined through the MPC entry.

To enhance user convenience, nine rigid body elements are available in MSC Nastran. See Table 8-1. These elements require the specification of the degrees of freedom that are involved in the equations of constraint. All the coefficients in these equations of constraint are calculated in MSC Nastran.

| Name   | Description  | Rotation | DOF              |
|--------|--|----------|------------------|
| RROD   | A open-ended rod which is rigid in extension                       | Y        | m = 1            |
| RBAR   | Rigid bar with six degrees of freedom at each end.                 | Y        | $1 \le m \le 6$  |
| RJOINT | Rigid joint with six degrees of freedom at each end.               | Y        | $1 \le m \le 6$  |
| RTRPLT | Rigid triangular plate with six degrees of freedom at each vertex. | Y        | $1 \le m \le 12$ |

Table 8-1Rigid Element and MPC Entries



Table 8-1 Rigid Element and MPC Entries (continued) Finite m = DependentName Description Rotation DOF Y RBE2 A rigid body connected to an arbitrary number of grid  $m \geq 1$ 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. Y RBE1 A rigid body connected to an arbitrary number of grid  $m \ge 1$ points. The independent and dependent degrees of freedom can be arbitrarily selected by the user. Y RBE3 Defines a constraint relation in which the motion at a  $1 \le m \le 6$ 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. Ν RSPLINE Defines a constraint relation whose coefficients are derived  $m \ge 1$ 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. Ν RSSCON Define a multipoint constraint relation which models a  $m \ge 5$ clamped connection between shell and solids. MPC Rigid constraint that involves user-selected degrees of Ν m = 1freedom at both grid points and at scalar points. The coefficients in the equation of constraint are computed and input by the user.

### Static Loads

A load is applied to the model only if it is specifically called out in the case control section. If 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.

For more information related on different types of loads refer to MSC Nastran Nonlinear (SOL400) User's guide.

### **Using Patran to Apply Loads and Boundary Conditions**

The following table outlines the SOL 400 applied conditions that are supported and can be written into the MSC Nastran SOL 400 input file.



### **Object Type**

- Displacement/Velocty/Acceleration
- Nodal
- Element Uniform
- Element Variable
- Force Nodal
- Pressure Element Uniform
- Element Variable
- Temperature Nodal
- Element Uniform
- Element Variable
- Inertial Load Element Uniform
- Initial Displacement Nodal
- Initial Velocity Nodal
- Distributed Load Element Uniform
- Element Variable
- CID Distributed Load Element Uniform
- Element Variable
- Total Load Element Uniform
- Contact Element Uniform
- Crack (VCCT) Nodal
- Initial Plastic Strain Element Uniform
- Initial Stress Element Uniform
- Initial Temperature Nodal

The Loads and Boundary Conditions application controls which loads and boundaries and contact information will be created in the MSC Nastran input file.

For more information related on using Patran to apply loads and boundary conditions of loads refer to *MSC Nastran Nonlinear (SOL400) User's guide.* 

### **CWELD/CFAST/CSEAM Element Enhancements**

The CWELD /CFAST elements have been changed so that there is now a consistent formulation between linear and SOL 400 nonlinear analysis. Additionally, the CSEAM element is now supported in SOL 400 and has a consistent formulation between linear and SOL 400 nonlinear analysis.



### **Benefits**

- The CWELD/CFAST elements provide the same element output formats in both linear and SOL 400 nonlinear solutions.
- In the CFAST/CSEAM/CWELD analysis, the auxiliary points generated are in the solution set.
- The CWELD/CFAST algorithm has been improved to find the Best Possible Projection with zero projection tolerance improvements.
- The improved CWELD with options PARTPAT and ELPAT, and CFAST elements do not move GA and GB if both are supplied by the user, thus maintaining user mesh integrity.
- The 3x3 mesh limitation has been removed for the CWELD with options PARTPAT and ELPAT and the CFAST elements.
- There are no required changes in the user element input description of the CFAST/CSEAM/CWELD elements.
- The CFAST/CWELD/CBUSH provides nonlinear force output for SOL 400 ANALYSIS=NLTRAN.
- MPC force output is available for the connector element constraints.
- Besides global search algorithm control there is now local connector element connectivity control via the new CONCTL bulk data entry.
- A brief summary of connector projection results is output in the F06 file for each connector type.
- A new SWLDPRM, CSVOUT, UNITNUM entry will produce a comma separated file useful for reports.
- The CSEAM and CWELD (not by default) can now contribute mass to the structure.

### **Description of Features**

Details of the improved CWELD/CFAST algorithm are described below.

### **Formulation Changes**

In the new consistent formulation for the CWELD/CFAST elements for linear analysis, RBE3 elements are written internally, and the auxiliary points are in the solution set and both are identifiable by the SWLDPRM, PRTSW entry.

- The auxiliary grids generated start with GRID ID 101000001. There are always four auxiliary grids for patch A and four auxiliary grids for patch B.
- The RBE3 elements generated start with 100001002. An RBE3 is generated for each auxiliary point for each patch A and B tying each patch grid to that auxiliary point. There is a RBE3 generated for GA tying GA to its patch auxiliary points and a RBE3 generated for GB tying GB to its patch auxiliary points.
- Both linear and nonlinear output is consistent.

The new consistent CWELD/CFAST is selected by default. The old CWELD/CFAST can be actuated by using the PARAM, OLDWELD, YES entry.

It has been determined in testing that the above formulation changes produce little or no change in solution results when comparing the old CWELD/CFAST against the new CWELD/CFAST results.



## **Enhanced Search Algorithm**

For the new connector logic, the search algorithm has been enhanced based on user inputs in an attempt to achieve the best possible connections. The new search tolerance starts with a zero projection tolerance. This may result in changes from the previous connector results using the old CWELD/CFAST elements.

The list below gives a brief summary of the highlights of the improved CWELD/CFAST algorithm.

- 1. For the CFAST and the CWELD with options PARTPAT and ELPAT, grids GA and GB internally keep the user-specified IDs and the user-specified locations. This change was primarily introduced because many users complained that the location of GA and GB represented their modeling procedures and desired mesh locations.
- 2. For the CWELD with ELEMID/GRIDID option, grids GA and GB internally keep the user-specified IDs and the user-specified locations, but in the case when GA and GB are associated with shell patches, a duplicate internal grid is generated to avoid singularity of the generated RBE3.

CWELD, 5646, 22, , ELEMID, 3276, 3115 , 2191, 1941 CTRIA3, 2191, 8, 3272, 3276, 3271

Grid 3276 as input from standard mesh modeling procedures will automatically be placed in the independent degree of freedom set, or may have been placed by the analyst in a SPC or MPC set at generation time. In either case, the CWELD algorithm must create an internal constraint on this grid using a RBE3 element. This causes a set conflict which is avoided by generating an internal grid.

3. For two stacked connectors having a common patch with a common grid, the program checks duplicated GA/GB and only a single RBE3 is generated for the common patch.

CWELD, 11, 100, 9001, PARTPAT, 3001, 3002 CWELD, 12, 100, 9002, PARTPAT, 3002, 3003

4. If the user specifies both grids GA and GB, for the CFAST and the CWELD with options PARTPAT and ELPAT, the SWLDPRM, GSMOVE entry is nonfunctional.

In the CFAST and the CWELD with options PARTPAT and ELPAT, if the user specifies both GA and GB they will not be moved. This may cause the CFAST/CWELD search algorithm to fail for some welds that had passed under the old CWELD/CFAST search algorithm. If this occurs, the user can do one of four things:

- a. Determine a better location for GA and GB of the failing welds so that they may project.
- b. Remove GA and GB from the CWELD/CFAST entry and replace with a GS entry allowing the CWELD/CFAST algorithm to move and project and generate internal GA and GB locations.
- c. Use SWLDPRM, MOVGAB, 1 to generate internal GA and GB grids at the corrected locations for all CWELD/CFAST. The locations of the original user specified GA and GB are unchanged.
- d. Use the new CONCTL Bulk Data entry with SWLDPRM, MOVGAB, 1" to allow local control of specific welds to correct the locations of grids GA and GB.

CONCTL, 83, , MOVGAB, 1

Where: SET3, 83, ELEM, 1345, 2678



- 5. The maximum tolerance for SWLDPRM, PROJTOL has been relaxed.
  - a. Regardless of the value of SWLDPRM PROJTOL, the algorithm starts by assuming a zero projection tolerance for the projections of GA/GB for the CWELD option PARTPAT or the CFAST option PROP and for GAHi/GBHi for the CWELD options PARTPAT and ELPAT and any CFAST option.
  - b. The tolerance is increased by 0.02 until a projection is found or the PROJTOL value is reached.
  - c. This can be turned off while computing the auxiliary grid projection onto EIDA/EIDB by setting PROJTOL= value where 0.0. value . 1.0. In this case, the projection calculation starts at tolerance = PROJTOL|. For the rest of the projection search, the algorithm reverts back to (a) and (b) above.
- 6. A brief Connector Summary of projection results is always output in the f06 file for each connector type: FST-ELEM, WLD-PARTPAT, WLD-ELEMID, etc.
- 7. For linear connectors, MPCFORCE output is available. In nonlinear SOL 400, the RBE3 elements generated become Lagrange elements if the default RIGID=LAGRANGE is used and are no longer in the MPC set; hence, there will be NO MPCFORCE output for RIGID=LAGRANGE.

CWELD will not contribute to MASS by default even if its associated MATi entry has a nonzero density. To react to a nonzero density SWLDPARM,WMASS,1 is required. If mass is computed, the PARAM,COUPMASS effects the mass calculation.

- 8. In the improved CFAST and CWELD, GA and GB are not moved and internal coincident grids are not generated at a new location; thus, two additional restrictions are required.
  - a. There can be no user-supplied constraints on GA and GB. A fatal message will be issued if there are any.
  - b. The CWELD length must be >  $10^{-6}$ . The point to patch option defined by ELEMID or GRIDID will, however, still create a new GS internally to obtain a minimum required length; i.e., LDMIN . length/D . LDMAX. For the point-to-patch connection, GS is used as GB. The algorithm will use the new GS as GB but keep the user-supplied GS unchanged. Since the point-to-patch is often used to tack two shell corners, the default LDMIN may cause the connector to be unstable. To avoid this, it is recommended that the user set LDMIN=1.E-6 on the appropriate PWELD entry.
- 9. CFAST and the CWELD with options PARTPAT and ELPAT with the improved formulation has removed the restriction that a connector patch cannot span more than three elements. It will now span over a patch of as many elements that the value of diameter D of the patch encloses and for which projections can be found.

The following Figure, all element grids contained in the green circle region of say patch A will be used in a RBE3 connection in addition to the RBE3 connections generated for the four auxiliary points. The example for this figure can be found at /tpl/connectr14/cei\_103.dat.

The CWELD ELEMID option still only connects by design two elements. The diameter is only used to compute the beam stiffness.

The green circle passes through the four auxiliary points of the patch (the nine digit grid IDs.). The user-specified diameter D on the PWELD and PFAST entries determine the locations of the four auxiliary points. The green circle diameter is approximately 1.253D. The element grids shown outside the green circle belong only to the respective auxiliary points.



For higher-order shell elements CQUAD8 or CTRIA6 with no missing midside nodes, the RBE3 relationships use only the midside nodes. If one or more midside node is missing, then the corner nodes are used.

The diameter D on the PFAST/PWELD entry is used to determine the projection location of these auxiliary points as well as the stiffness properties of the patch to patch connection.

A single RBE3 then connects the four auxiliary points and the shell grids within the green circle to the connector grid GA=4000065.

The SWLDPRM, PRTSW entry will list the additional grids located within the green circle under FMESH SHELL A or B GRIDS where FMESH is the f06 file listing title of the additional shell grids connected in the RBE3 relationships for Finer MESH.

Table 2-1 shows the grids associated with auxiliary grid 101000023 of Figure 2-1 for its RBE3 generation. The WTi's are weight factors based on patch shape functions. Grids G1, G2, G3 are selected for RBE3 EID 100001026 because they are the shell grids of the triangular element that contains the projected auxiliary point.

| RBE3 | EID       | GAH3      | REFC | Weight | Ci  | Gi         |
|------|-----------|-----------|------|--------|-----|------------|
|      | 100001026 | 101000023 | 123  | WT1    | 123 | G1=4007883 |
|      |           |           |      | WT2    | 123 | G2=4007884 |
|      |           |           |      | WT3    | 123 | G3=4007902 |

Table 2-2 shows the grids associated with grid GA=4000065 of Figure 2-1 through RBE3=100001022. G5 through Gn are the grids contained within the green circle. Grids G2=4007884, 4007869, 4007815, and 4007830 are NOT included in any of the G5-Gn entries because they are included in their associated auxiliary point RBE3 elements 100001026 because they are the shell grids of the triangular element that contains the projected auxiliary point.

| RBE3 | EID       | GRID_A  | REFC   | Weight | Ci  | Gi             |
|------|-----------|---------|--------|--------|-----|----------------|
|      | 100001022 | 4000065 | 123456 | WT1    | 123 | GAH1=101000021 |
|      |           |         |        | WT2    | 123 | GAH2=101000022 |
|      |           |         |        | WT3    | 123 | GAH3=101000023 |
|      |           |         |        | WT4    | 132 | GAH4=101000024 |
|      |           |         |        | 1.0    | 123 | G5= 4007885    |
|      |           |         |        |        |     |                |
|      |           |         |        | 1.0    | 123 | Gn=last        |



10. The CWELD/CFAST/CBUSH has the additional enhancement that for nonlinear transient SOL 400 with ANALYSIS=NLTRAN they will request Element FORCE output for the CWELD/CFAST/CBUSH elements. FORCE=ALL will request force output for all CWELD/CFAST/CBUSH elements. Element FORCE output in SOL 400 nonlinear transient analysis is unique to CWELD/CFAST/CBUSH elements. Other elements will not list force output in SOL 400 nonlinear transient.

If in a SOL 400 nonlinear transient, you have FORCE=ALL and carry this over to say an ANALYSIS=NLSTAT, you will get Element FORCE of ALL elements capable of force output, such as CWELD, CFAST, CBEAM, CQUAD4, etc.

11. For user convenience, an additional SWLDPRM command useful for reports creates a comma separated file of the SWLDPRM, PRTSW, using the command SWLDPRM, CSVOUT, UNITNUM where UNITNUM is assigned via the:

```
ASSIGN USERFILE=myfile.csv,UNIT= UNITNUM, FORM= FORMATTED, DELETE, STATUS=NEW.
```

### **Additional Information**

- 1. In SOL 400, for ANALYSIS=NLSTAT or ANALYSIS=NLTRAN, the generated RBE3 constraints become Lagrange elements and may undergo large rotation. For ANALYSIS=NLTRAN with initial conditions (IC=n) in case control that cause large initial stresses in the structure at time t=0, the case control entry RIGID needs to have the value RIGID=LINEAR to insure convergence.
- 2. For user desiring to postprocess the CFAST/CSEAM/CWELD connectors with their own methods, the following is useful:
  - a. The GEOM2 table contains, after module MODGM2, a record ELCORR that correlates the CFAST/CSEAM/CWELD and its associated RBE3 elements. Also, this module will, for linear analysis and for nonlinear SOL 400 analysis run with RIGID=LINEAR, place the internal generated RBE3 into the GEOM4 table.
  - b. In SOL 400 with RIGID=LAGRANGE (Default), internally generated RBE3 elements go into the GEOM2 (as do all other user specified rigid elements) not the GEOM4 table.
  - c. The CWELD/CFAST/CBUSH force output for ANALYSIS=NLTRAN in SOL 400 is OP2 file output on OEFNLXX data block. If SCR=POST is run, then this force data is also written to the data base file OEFNL3 and op2 file OEFNL.op2 is also written.
- 3. The DISPLACEMENT (CONNECTOR=) Case Control Command works in the same fashion for both the old connector formulation and the new connector formulation.

For the CFAST with option ELEM and the CWELD with option ELPAT, the shell elements connected on each patch must have same property identification number of PSHELL entry.

If parameter OSWPPT is used to specify the offset for internally generated grid IDs, its value should be greater than the maximum identification number of GRID entries to avoid conflict IDs.



### **Connector Stiffness**

Connector contribution to a structural model's overall stiffness is sensitive to the models mesh size and the orientation of the connector relative to the mesh. Thus, the discretization process itself may cause, for example, a model using a fine mesh to be stiffer in torsion than a corresponding model using a coarse mesh. For production models that correlate well with test, refining the mesh may cause an inherent overall loss of stiffness due to mesh refinement and hence loss of correlation.

To allow you some control over stiffness, improved connectors (CWELD with ELPAT or PARTPAT or CFAST) are provided with two options to provide additional connector stiffness. The two options may be used individually or in combination.





D<sub>ratio</sub> = DRATIO \* D<sub>connector</sub> (Circle not shown)

Aconnector = m \* D<sup>2</sup>connector /4; Aconnector is used by PWELD only for "beam" properties of Patch to Patch connection

A<sub>box</sub> = L<sup>2</sup>; L= $\sqrt{\pi}$  D<sub>ratio</sub> / 2; D<sub>patch</sub> =  $\sqrt{2}$  L =  $\sqrt{\pi/2}$  D<sub>ratio</sub>  $\approx$  1.253 D<sub>ratio</sub>

A disadvantage of this method is that as DRATIO is increased using the global command SWLDPRM, DRATIO, value. So, some connector elements may begin to fail because they may no longer be able to find a patch projection. To overcome this, the SWLDPRM, NREDIA, can be increased to a value as high as 8 to allow failing welds to halve their patch diameters up to eight times.

If the SWLDPRM, NREDIA is not an approach the user wishes to pursue, then for these failing elements, the bulk data entry CONCTL, SETID, , DRATIO, value can be used to define a set for failing connectors and set a value of DRATIO for these connectors that allows them to find a projection.

The second stiffening algorithm attempts, based on the diameter of the connector, to determine a measure of the mesh discretization. This feature is activated by SWLDPRM, SKIN, 1 or CONCTL, SETID, , SKIN, 1. The default is a 0 which implies no stiffening. There is an associated stiffening factor SWLDPRM, SCLSKIN with value = 0.10 as default.

Depending on the complexity of the model and the overall mesh size and the number of connectors within the model and the diameter of the connectors relative to the mesh, the default value tends to stiffen a structural model from about 0.4% to about 4%. A value of SCLSKIN=10.0 stiffens coarser mesh models by about 10% to 11% and finer mesh models by about 2% to 6%.

The contribution of the stiffening algorithm to the overall stiffness of the FEM model eventually reaches a limit. For example, a very large value SCLSKIN=100 increases the stiffness of the models overall by only about 0.1% to 2% over the stiffness obtained for SCLSKIN=10.

For a correlated structural model evaluated at a specific mesh size, with an aim to refine the mesh for some portion of this model containing connectors, while leaving other portions containing connectors with an unmodified mesh, it is recommended that you enter the SKIN, 1 and SCLSKIN, real value on the CONCTL bulk data entry referring to the connectors within the area of the refined mesh. Different refined mesh areas within the structural model can have different values of SCLSKIN associated to the specific connectors in each refined region.



For postprocessing of the SKIN option, for the affected shell elements, an updated EPT table is available after module MODGM2. It contains the PSKNSHL record that correlates the property data of the shells involved and a list of shell elements for each patch modified.

### **Detailed Projection Algorithm for Best Possible Projection**

- 1. The Enhanced algorithm applies to the following projection calculations:
  - a. Find projections of GA/GB for the PARTPAT format of CWELD and PROP format of CFAST.
  - b. For ELPAT format of the CWELD and the ELEM format of the CFAST, the user has specified the specific shell elements and therefore no element search for GA/GB projections is required. Though no search is required, GA/GB, however, must project onto the user specified EIDA/EIDB.
  - c. Find projections of the auxiliary grids GAHi/GBHi for the PARTPAT and ELPAT options on the CWELD and any CFAST option respectively.
- 2. The projection algorithm searches for possible projections from shell elements with shell grids that are closest to GS. The closest grid may connect to several shell elements; hence, more than one shell element may get a projection from GS for curved patches. The shell elements with projections from GS are collected and the selection is based on the SHIDA/SHIDB pair with the smallest angle between their normal vectors.
  - a. The old CWELD/CFAST algorithm used the first shell elements found to get a projection and used SWLDPRM, GMCHK, 1 and 2 to provide some control. These two options have no effect on the new formulation.
  - b. Backward connections sometimes occur if the patch is near the boundary of a structure, and there is a vertical flange associated with the patch elements. In this case, SWLDPRM, GMCHK, 3 may be used to prevent backward projection.
- 3. The minimum angle selected above must be SWLDPRM GSPROJ if GSPROJ. 0.0
- 4. If the user has not specified both GA and GB and the algorithm cannot find a GSPROJ satisfied projection, then for SWLDPRM GSMOVE entry, the point GS will be moved in an attempt to satisfy the projection requirement.
- 5. Reminder: If user has specified both GA and GB and CFAST and the CWELD with options PARTPAT and ELPAT are used, then GSMOVE will be ignored and the connector will fail to connect if the user has taken the default SWLDPRM, NREDIA, 0 for NREDIA. Failed connectors issue USER FATAL MESSAGE 7635.
- 6. If the GSMOVE specification limit is reached for the CFAST or the CWELD with options PARTPAT and ELPAT and SWLDPRM NREDIA ? 0, then the diameter of the connector will be reduced by half to compute new locations of auxiliary grids. If necessary this is repeated until the NREDIA specified value is reached.
  - a. When the NREDIA ? 0 is initiated, the GS at its current location is used for GSMOVE . 0.
  - b. When the NREDIA ? 0 is initiated, the GS at its original location is used for the new option  $_{\rm GSMOVE}~<~0.$



#### CHAPTER 8 151 Constraints



# Boundary Conditions

9

| Introduction    | 153            |       |     |
|-----------------|----------------|-------|-----|
| Zero and Enfor  | ced Displacem  | ents  | 153 |
| Fixed Direction | Grid Point For | ces   | 153 |
| p-Element Loa   | ds and Constra | aints | 154 |
| Thermal Loads   | (TEMP and TE   | MPD)  | 155 |
| Inertial and Dy | namic Loads    | 156   |     |



## Introduction

SOL 400 supports the following loads and boundary conditions:

- Constrained nodal displacements (zero displacements at specified degrees of freedom). Enforced nodal displacements (nonzero displacements at specified degrees of freedom in the nodal coordinate system). Both constrained and enforced displacements can be specified as relative or absolute.
- Forces applied to nodes in any coordinate system.
- Follower force distributed loads.
- Temperature applied to nodes. Temperature can be applied as a load in a structural analysis. The reference temperature is user definable.
- Inertial body forces, acceleration and velocity can be applied in the global coordinate system.

## Zero and Enforced Displacements

### **Enforced Motion Constraints (SPCD and SPCR)**

SPCD and SPCR define an enforced displacement value for static analysis and an enforced displacement, velocity or acceleration in dynamic analysis. SPCD provides the final total displacement, velocity, or acceleration at the end of the current loadcase, and SPCR determines the incremental displacement, velocity, or acceleration during the current loadcase, relative to the value of the previous loadcase.

When a GRID has deformation due to an applied load or motion applied in the previous STEP and the user wishes to prescribe displacements in the current STEP relative to this unknown magnitude, then the SPCR capability of applying relative motion will provide an efficient procedure.

The primary applications for enforced motion constraints are as follows:

- 1. To apply a motion accurately on a structure;
- 2. To apply an incremental motion on a structure;

It should be noted that SPCD/SPCR are treated as loads. SPC and SPC1 are requested by the SPC Case Control command, while SPCD and SPCR are requested by the LOAD or DLOAD Case Control commands. A degree of freedom referenced by SPCD must be also on an SPC or SPC1 entry. A degree of freedom referenced by SPCR must be also on a SPC1, but cannot be on an SPC.

SPCR: It defines an enforced relative displacement value for a load step in SOL 400 and SOL 600.

SPCD: It defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

## **Fixed Direction Grid Point Forces**

You can categorize a particular type of load as either a point (concentrated) load or surface/volumetric (distributed) load, depending on application conditions. The spatial distribution of the load can be uniform



or nonuniform. Special loading types also exist in various analyses. For example, centrifugal loading exists in stress analysis, and convection and radiation exist in heat transfer analysis. You can add point loads directly to the nodal force vector, but equivalent nodal forces first must be calculated by MSC Nastran from distributed loads and then added to the nodal force vector.

For detailed information on Grid Point Forces and Grid Point Follower Forces refer to *MSC Nastran Nonlinear (SOL400) User's guide.* 

## p-Element Loads and Constraints

### **Point Loads**

The FORCE entry is used to define a static load applied to a geometric grid point in terms of components defined by a local coordinate system. The orientation of the load components depends on the type of local coordinate system used to define the load. The directions of the load components are the same as those indicated on Figure 2-2 of Grid Point and Coordinate System Definition, 32 for displacement components. The FORCE1 entry is used if the direction is determined by a vector connecting two grid points, and a FORCE2 entry is used if the direction is specified by the cross product of two such vectors. The MOMENT, MOMENT1 and MOMENT2 entries are used in a similar fashion to define the application of a concentrated moment at a geometric grid point. The SLOAD entry is used to define a load at a scalar point. In this case, only the magnitude is specified since only one component of motion exists at a scalar point.

### Line and Pressure Loads in p-Element Analysis

The GMLOAD entry is used to specify line and pressure loads as follows:

- Applying directly to the finite elements entities, FEEDGEs and FEFACEs.
- Applying to the geometry entities GMCURV and GMSURF.

### **Body Loads**

Body loads consists of thermal loads and gravity loads.

For thermal loading, the TEMP Bulk Data entry is used for a temperature distribution that is trilinear over the element, whereas the TEMPF entry is used over a set of elements. The initial or reference temperature of a body must be supplied using these entries. Gravity loads are defined by the GRAV entry.

### **Boundary Conditions**

The SPC and SPCD entries are used for the point constraints that are allowed only on corner GRID points. The GMSPC entry is used to define zero constraints for FEEDGEs, FEFACEs, GMCURVs, and GMSURFs. For nonzero constraints, the GMBC entry is used. In general, for multiple input data for FEEDGE, FEFACE, GMCURV, and GMSURF entities, the hierarchy set to resolve these conflicts is as follows:

- GRIDs
- FEEDGEs
- GMCURVs
- FEFACEs



GMSURFs

## 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 (TEMP and TEMPD bulk data entries in the *QRG* and interpolated to grid points within elements.

Alternatively, temperature data can be specified on an element-by-element basis as shown in Table 9-1.

| Elements                          | Temperature Data   | Bulk Data Entry |
|-----------------------------------|--|-----------------|
| CROD, CONROD,<br>CTUBE            | Average temperature at ends A and B.   | TEMPRB          |
| CBAR, CBEAM, CBEND                | Average temperature and cross-sectional temperature gradients at ends A and B. | TEMPRB          |
| CBEAM3                            | Temperature field and gradients along the beam                                 | TEMPB3          |
| CQUAD4, CTRIA3,<br>CQUAD8, CTRIA6 | Average temperature and gradient in the thickness direction.                   | TEMPP1          |

#### Table 9-1 Bulk Data Entries Used for Temperature Definition on Elements

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.



## Inertial and Dynamic Loads

### **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. Gravity also includes the effects of nonstructural mass and lumped mass defined through the CONM1 and CONM2 entries. 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.

### **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.

You must select one of two methods for the internal calculation of the loading vector.

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 9-2.

|         | No Offset               | Offset   |
|---------|-------------------------|----------|
| Lumped  | Method=1 or<br>Method=2 | Method=1 |
| Coupled | Method=2                | Neither  |

| Table 9-2 | Restrictions when using the RFORCE Entry |
|-----------|--|
|-----------|--|

In addition, for problems with elements that have edge grid points (CQUAD8, CTRIA6, CTRIAX6, CHEXA, CPENTA, and CTETRA), correct centrifugal loads are produced only if the parameter PARAM,COUPMASS,x (where x is greater than 1), is included in the input file and Method 2 is used.

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 nonzero 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.

### 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. The ACCEL and ACCEL1 bulk data entries in the *QRG* are used in the same way as other load entries (such as GRAV, FORCE, and MOMENT, etc.) through the MSC Nastran case control commands.

### **Initial Stress and Initial Plastic Strain Mapping from Previous Results**

MSC Nastran allows you to enter a set of initial stresses through the ISTRESS bulk data entry that simulates the stress state in the structure at the beginning of an analysis. A typical example is prestress in a tensioned fabric roof. The set of initial stresses must be self-equilibrating and should not exceed the yield stress of the material.

MSC Nastran allows you to define the equivalent plastic strain using the IPSTRAIN bulk data entry throughout the model. This is useful in metal forming analysis in which the previous amount of equivalent plastic strain is often required. This history dependent variable represents the amount of plastic deformation that the model was subjected to, and is used in the work (strain) hardening model. This is only used to determine the value of the strain hardening once plasticity occurs.

Preprocessors such as Patran and SimXpert are very useful in mapping stress states between analyses.



# 10 Iteration Control in Nonlinear Analysis

- Introduction 159
- Nonlinear Characteristics and General Recommendations 160
- Starting the Analysis 161
- Load Incrementation and Iteration 166
- Load Increment Size 167
- Convergence Controls 169
- Solution Parameters 170
- Defining Subcases in Patran 171



## Introduction

MSC Nastran SOL 400 provides a comprehensive FE solution for multi-physics problems such as structure analysis, thermal analysis, as well coupled analysis. Even though it is originally targeted to deal with various nonlinearities as geometry nonlinearity, material nonlinearity, as well boundary nonlinearity (contact), it is also embedded with the powerful capabilities of MSC Nastran on linear analyses.

The FEM is a powerful tool for analyzing complex problems in structural and continuum mechanics. The analysis of a structure using the FEM has four basic steps:

- 1. Modeling, in which the structure is subdivided into an assemblage of discrete volumes called finite elements, and properties are assigned to each element.
- 2. Evaluation of element characteristics, such as stiffness and mass matrices, followed by assembling the element characteristic matrices to obtain the assembled or "global" matrices characteristic of the entire structure. A similar process is followed to obtain the total loads, in vector form, applied to the structure.
- 3. Solution of the system equations for displacements, natural frequencies and mode shapes, or buckling load factors.
- 4. Calculating other quantities of interest, such as strains, stresses and strain energy. MSC Nastran SOL 400 provides a comprehensive FE solution for multi-physics problems such as structure analysis, thermal analysis, as well coupled analysis. Even though it is originally targeted to deal with various nonlinearities as geometry nonlinearity, material nonlinearity, as well boundary nonlinearity (contact), it is also embedded with the powerful capabilities of MSC Nastran on linear analyses. The FEM is a powerful tool for analyzing complex problems in structural and continuum mechanics. The analysis of a structure using the FEM has four basic steps:

SOL 400 uses the finite element displacement method, in which a system of equations is solved to obtain the displacements at all node points of the structure. Comprehensive presentations of the FEM together with numerous applications are available in textbooks and the research literature. Solving the equations of finite element analysis involves the manipulation of large matrices of numbers, which is best done using computers. While modern computers are extremely fast and have vast amounts of memory, the "bottom line" in finite element analysis is that it is very easy to discretize a structure to the point that it is considered to be a "large" problem.

Typically, the geometry of the problem will dictate how fine of a mesh is required to get an acceptable solution. While modern graphical user interfaces (GUI's) such as Patran can provide some guidance and tools for evaluating the mesh density, in the end it is up to the user to ensure that the problem is adequately discretized. The size of FEA problems is typically measured by the number of degrees of freedom (or degrees of freedom) in the finite element mesh.

For nonlinear problems, the size is even more important because the methods used to solve these nonlinear equations are usually iterative in nature, meaning that the system of equations must be solved many, many times to follow the behavior of the structure as it changes. The changes characterized may be the shape (called large deformation), or the status of the material (metal materials typically yield when a certain stress level is exceeded), or loading changes (the load orientation may follow the deformation of the structure, or different parts of the structure may come into contact changing the load path). The purpose of this chapter is to



describe the numerical methods and procedures required to solve the linear and nonlinear equations used to perform finite element simulations.

We begin this chapter with some general observations and recommendations on how to approach the solution to nonlinear FEA problems, and then move through the details of how to get an accurate, efficient solution using the algorithms and methods available in MSC Nastran SOL 400. Along with the efficiency, we discuss the concept of "robustness", which means choosing alogorithms that will tolerate changes in the problem (sometimes abrupt, such as contact characterizing impact) with out causing the solution algorithms to fail. A robust numerical method automates changes in the solution parameters, such as load increment size, as required by the problem yet still provides a reasonably accurate and efficient solution.

The characteristics of the solution methods and strategies in this chapter include: solution iteration methods, transient analysis effects such as time step selection, and time integration method, arc-length methods for post-buckling, and convergence measurements and controls. This chapter may be read as a tutorial, but is probably best used as a reference when considering specific problems encountered while trying to solve a nonlinear finite element problem.

# Nonlinear Characteristics and General Recommendations

Modeling for nonlinear analysis is not exempted from the guidelines for good modeling practice pertaining to linear analysis, which are summarized as follows:

- The analyst should have some insight into the behavior of the structure to be modeled; otherwise, a simple model should be the starting point.
- Substructuring should be considered for the modularity of the model and/or synergism between projects and agencies involved. The structure represented by a substructure is always linear.
- The size of the model should be determined based on the purpose of the analysis, the trade-off between accuracy and efficiency, and the scheduled deadline.
- Prior contemplation of the geometric modeling will increase efficiency in the long run. Factors to be considered include selection of coordinate systems, symmetric considerations for simplification, and systematic numbering of nodal points and elements for easy classification of locality.
- Discretization should be based on the anticipated stress gradient, i.e., a finer mesh in the area of stress concentrations.
- Element types and the mesh size should be judiciously chosen. For example, avoid highly distorted and/or stretched elements (with high aspect ratio).
- The model should be verified prior to the analysis by some visual means, such as plots and graphic displays.

Nonlinear analysis requires deeper insight into structural behavior. First of all, the type of nonlinearities involved must be determined. If there is a change in constraints due to contact during loading, the problem may be classified as a boundary nonlinear problem and would require contact body creation and contact modeling. The material nonlinearity is characterized by material properties. However, the material nonlinear effects may or may not be significant depending on the magnitude and duration of the loading, and occasionally on environmental conditions. The anticipated stress level would be a key to this issue. The geometric nonlinearity is characterized by large rotations which usually cause large displacements. Intuitively,



geometric nonlinear effects should be significant if the deformed shape of the structure appears distinctive from the original geometry without amplifying the displacements. There is no distinct limit for large displacements because geometric nonlinear effects are related to the dimensions of the structure and the boundary conditions. Strain is nondimensional, and a strain greater than 2% indicates a geometrically nonlinear problem.

## Starting the Analysis

Nonlinear analysis is intrinsically a multi-increment load process where the applied loads and/or displacements are solved for, not in a single load increment but in a number of load increments. The multiple step procedure is necessary for the FE code to update changing conditions in the model during the analysis. This situation is routinely encountered in nonlinear analysis because the material properties and/or boundary conditions can change during the analysis (e.g., with the onset of plasticity (material nonlinearity)), or with the occurrence of contact (boundary conditions nonlinearity). Below are the steps in a general linear and a nonlinear analysis. The presence of an extra loop of iterations (Newton-Raphson iterations) is the unique feature of a nonlinear solution procedure.

### A. Steps in Linear Analysis:

- 1. Set up the model (done by user, <u>before</u> the model is submitted).
  - Mesh the part
  - Apply Material Properties
  - Apply Boundary Conditions
  - Submit Job
- 2. Job Solution (done by FE Code).
  - Assembly of stiffness matrix
  - Solution of stiffness matrix.
  - · Compute displacements, strains, stresses (and other results)
- 3. View Results.

Figure 10-1 shows the basic steps that MSC Nastran follows when solving a linear statics analysis.







You are guaranteed a solution if the boundary conditions and material properties are set up correctly. The stiffness matrix is assembled and solved only once in the entire analysis.

### **B. Steps in Nonlinear Analysis:**

- 1. Set up the model (done by you, before model set-up).
  - Mesh the part.
  - Apply Material Properties.
  - Apply Boundary Conditions.
  - Submit Job.
- 2. Job Solution (done by FE Code):



Newton-Raphson Iteration scheme begins: Apply a portion of the total load to start: (1% in this case):

- Assembly of stiffness Matrix.
- Solution of the Stiffness Matrix.
- Check for convergence (this is an important step which is only seen in nonlinear analysis). If converged, the solution/structure is in equilibrium. Go to step 3. If not converged, update information and reassemble, resolve stiffness matrix. Keep iterating till convergence is achieved.
- 3. (After convergence) Get displacements, strains, stresses.
- 4. Apply the next increment of load and go to Step 2. Keep doing this until all the load is applied.
- 5. View Results.

### Load Increments and Iterations

### **Load Increments**

In the loading history, the total change of loading applied during a step can be subdivided into smaller parts to allow the solution to converge. These subdivisions within a subcase are termed load increments. The recommended procedure for SOL 400 nonlinear analyses is to use the adaptive load incrementation process which is the default method used when the job is set up in Patran. In this case, the number of load increments is specified in Patran on the Load Increments subform and are defined in the bulk data file by the NINC field on the NLSTEP entry. Selecting fixed incrementation and specifying the number of increments divides the total load change applied during the subcase into NINC equal parts for FIXED load incrementation, but only provides the initial load increment size in the case of adaptive load incrementation. The automated adaptive load incrementation method is strongly recommended over the manual method because it gives the algorithm the freedom to decrease the load when loading changes occur, such as when bodies come into contact or sudden discontinuities in the loading history occur. These types of changes often cause convergence problems when fixed stepping algorithms are used. See the NLSTEP bulk data entry for more details.

### Iterations

In the incremental solution process, the unbalanced forces that occur during a load increment are reintroduced internally into the solution until the solution has converged. The process of redistributing the unbalanced force within a load increment is known as an iteration. The iteration is the lowest level of the solution process. Iterations continue within a load increment until the solution converges or any of the specified convergence parameters are exceeded. A complete description of the numerical procedure used to solve the nonlinear problem is given in this chapter.





Figure 10-2 MSC Nastran Advanced Nonlinear Flow Diagram

The important point to note is that the total load is applied gradually in steps (or increments) and for each load step, the solution is arrived at after one or more iterations. If the behavior of the model is generally linear, few iterations are required to solve that load step. If the model behavior is complex/nonlinear, many iterations might be required. Each iteration involves an assembly and solution of the stiffness matrix. Hence, nonlinear problems inherently take longer than linear models (of the same size) to solve. At the end of each iteration, a check is made to see if the solution has converged. If the convergence check fails, the iteration is repeated with the new information. This process repeats until convergence is achieved. Following that, the next increment of load is applied. The load increments are applied until the full load of the model is solved.



To achieve accurate results, three key points have to be paid attention to:

- Iteration method;
- Load increment control; and
- Convergence criterion.

The three points will be discussed in later sections.

### What Triggers a Nonlinear Analysis in SOL 400

As for MSC Nastran, the model is treated as a nonlinear analysis, if it consists of:

- Large displacement by LGDISP parameter (for geometric nonlinearity);
- Large strain by LRGSTRN (for geometric nonlinearity);
- any contact (boundary nonlinearity);
- any active nonlinear material input option (material nonlinearity)
- linear perturbation (individual or combined nonlinearities)
- or any combination thereof.

The model may contain superelements, but only the residual structure (superelement 0) may consist of nonlinear elements, mixed with any type of linear elements. As aforementioned, other potentially nonlinear elements in the residual structure become actively nonlinear only if a LGDISP or LRGSTRN is used and/or if they use the nonlinear material data specified on any one of the entries: MATEP, MATVE, MATVP, MATF, MATS1, MATS3, MATS8, MATSORT, MATG, MATHE, MATHP, MATSMA, and MCOHE, and/or perturbation analyses.

### **General Recommendations for Nonlinear Analysis**

With these points in mind, additional recommendations are imperative for nonlinear analysis:

- Identify the type of nonlinearity and localize the nonlinear region for computational efficiency. If unsure, perform a linear analysis to help understand the problem.
- Segregate the linear region by using superelements and/or linear elements if possible. Notice that the potentially nonlinear elements can be used as linear elements.
- The nonlinear region usually requires a finer mesh. Use a finer mesh if severe element distortions or stress concentrations are anticipated.
- The step/subcase structure should be utilized properly to divide the load or time history for conveniences in data recovery, and database storage control, not to mention changing constraints and loading paths.
- The load or time for the subcase with NLSTEP should then be further divided into increments, for the purpose of convergence control. Automatic adaptive load incrementation, such as is available via NLSTEP, is the recommended method.
- Many options are available in solution methods to be specified on the NLSTEP data entries. The defaults should typically be used on all options to gain experience before experimenting with other options.



- Normal rotation for the "drilling degree of freedom" of shell elements restrained by the default value of 100 on the K6ROT parameter when the geometric nonlinearity is involved. In rare cases it may be necessary to adjust this value. This can help with convergence, but may also affect the results.
- Understand the basic theory of plasticity, creep, or rubber elasticity before using these capabilities.
- The time step size for a transient response analysis should be carefully considered based on the highest natural frequency of interest because it has significant effects on the efficiency as well as the accuracy of the solution. The automated procedures used by NLSTEP is adequate for this purpose.

## Load Incrementation and Iteration

### **Nonlinear Solution Procedure**

Based on the extensive numerical experiments, an attempt was made to establish a robust, general strategy suitable for most problems without requiring insight or experience. Variations in combining theories, algorithms, criteria, and parameter values with numerous test problems resulted in a succinct implementation.

The major feature of the nonlinear analysis is the requirement for the incremental and iterative processes to obtain a solution. The main issue is how to choose the most efficient method from the options available for the incremental and iterative processes in the solution of nonlinear equilibrium equations. The increment size for loads or time steps has the most significant effect on the efficiency and the accuracy of the computation, particularly in the path-dependent problems. The incremental and iterative processes are complementary to each other because the larger the increment size the more iterations the solution requires. While an excessively small increment reduces the computing efficiency without any significant improvement in accuracy, a large increment may deteriorate the efficiency as well as the accuracy; it may even cause divergence.

It is impossible to optimize the incremental step size in the absence of prior knowledge of the structural response. The NLSTEP adaptive load incrementation procedure should be exercised to determine the increment size based on the severity of the nonlinearity. Needless to say, no incremental load steps are required when the response is linear. In principle, the size of the load increment (or time increment for creep analysis) should be so chosen to yield a uniform rate of change in strains or stresses for the material nonlinear problems and a uniform rate of change in displacements for geometric nonlinear problems. Some adaptive method controls are available via NLSTEP, such as an automatic time step adjustment criteria and bisection/cut-back of loads upon divergence. The default method is to use the initial load increment value specified on the NLSTEP entry, then increase or decrease the size if the number of iterations required to converge varies significantly from the NDES value. If substantially more than NDES are required the time step size is decreased. Conversely, if substantially more iterations than NDES are required the time step size will be decreased. The magnitude of increase/decrease is controlled by the SFACT value.

The increment size can be varied from step to step by specifying different NLSTEP. It is recommended to define separate NLSTEP for every step even if the same values are specified, so that changes can be accommodated in the step level as needed.



## **Adaptive Solution Strategies**

Nonlinear finite element computations comprise material processes, element force computations, and various global solution strategies. The computational procedure involves incremental and iterative processes ranging from local subincrements to global solution processes. Performance of the finite element program can be scrutinized from three different perspectives: computational efficiency, solution accuracy and effectiveness. All of these attributes of the nonlinear program can be improved by adaptive algorithms.

There is a broad range of processes for which adaptive algorithms may be adopted in the computational procedure of nonlinear finite element analysis. The size of the load or time increment has the most profound effect on the efficiency as well as accuracy. However, it is difficult to determine optimal load or time increment size. The adaptive algorithm alleviates this difficulty. The most CPU consuming processes in nonlinear analysis are the stiffness matrix update operation and element force calculation. From the efficiency point of view, the number of stiffness matrix updates and the number of iterations should be minimized, which may be conflicting requirements.

Adaptive features implemented in MSC Nastran nonlinear capabilities are:

- Newton's iteration for static and implicit dynamic analysis
  - Adaptive stiffness matrix update strategies
  - Selective BFGS updates
  - Selective line search processes
  - Adaptive bisection and recovery of load increment
- Arc-length methods for static post-buckling and snap-through problems
  - Crisfield's arc-length method
  - Riks and modified Riks methods
  - Adaptive arc-length adjustment
  - Selective BFGS Updates
  - Adaptive switching algorithm for limiting cases
  - Adaptive correction in case of path reversal
- Direct time integration for transient response analysis
  - Quadratically accurate dynamic operators
  - Automatic time step adjustment

## Load Increment Size

Selecting a proper load step (time step) increment is an important aspect of a nonlinear solution scheme. Large steps often lead to many recycles per increment and, if the step is too large, it can lead to inaccuracies or even nonconvergence. On the other hand, using too small steps is inefficient.

The NLSTEP bulk data entry provides a unified load stepping scheme that replaces existing options entries such as NLPARM, TSTEPNL, NLPCI, and NLSTEP. This option can be used for statics and dynamic analyses, to select fixed or adaptive time stepping control, and to define the convergence criteria, as well to



make other options for mechanical, thermal and coupled analysis. MSC has made every attempt to make MSC Nastran SOL 400 as robust, efficient, and user friendly as possible; especially when used with either of the MSC graphical user interfaces (GUI's) Patran or SimXpert. For many problems the defaults entered are appropriate to minimize the job setup and obtain accurate results.

The NLSTEP option has a keyword CTRLDEF that automatically sets up the entries for the time stepping adjustment and convergence tolerance based upon how nonlinear you believe the problem is. This makes it possible to use smart default based on users' judgment of the nonlinearity of the model to be analyzed.

Under the keyword CTRLDEF, these three options (QLINEAR, MILDLY, and SEVERELY) adjust the parameter to provide you the desired results. As the names imply, CTRLDEF should be set to QLINEAR for linear solutions, MILDLY for mildly nonlinear, and SEVERELY for severely nonlinear behavior.

### **Fixed Load Incrementation**

When a fixed load stepping scheme is used, it is important to select an appropriate load step size that captures the loading history and allows for convergence within a reasonable number of recycles. For complex load histories, it is often necessary to break up the analysis into separate load cases with different step sizes. For fixed stepping, there is an option to have the load step automatically bisect/cut back in case of failure to obtain convergence. When an increment diverges, the intermediate deformations after each recycle can show large fluctuations and the final cause of program exit can be any of the following: maximum number of recycles reached, elements going inside out or, in a contact analysis, nodes sliding off a rigid contact body (see Chapter 6: Setting Up, Monitoring, and Debugging the Analysis for more on user fatal messages and their causes). These deformations are normally not visible as post results. If the cutback feature is activated and one of these failures occurs, the state of the analysis at the end of the previous increment is restored from a copy kept in memory or disk, and the increment is subdivided into a number of sub-increments. The step size is halved until convergence is obtained or the user-specified number of cutbacks has been performed. Once a subincrement is converged, the analysis continues to complete the rest of the original increment. No results are written to the results file during sub-incrementation. When the original increment is finished, the calculation continues to the next increment with the original increment count maintained. These issues are avoided by using the NLSTEP increment option. The adaptive load incrementation procedure of NLSTEP, using the MSC Nastran defaults, is the recommended method for load incrementation.

### Adaptive Load Incrementation in SOL 400 (NLSTEP)

An NLSTEP entry can be selected to execute a nonlinear static or nonlinear transient analysis in SOL 400. An NLSTEP entry is used in lieu of an NLPARM or TSTEPNL used in earlier solution sequences (SOL 106 or SOL 129) or both. If a NLSTEP is present anywhere in a STEP, then any NLPARM or TSTEPNL entries in the STEP is ignored. When used for coupled analysis, the NLSTEP entry must be above the first SUBSTEP command. A single NLSTEP entry is used for all SUBSTEPs of the STEP.

### **NLSTEP Bulk Data Entry**

The following section gives a description of the most commonly used parameters of NLSTEP. A complete description of all parameters can be found in the *QRG* under the NLSTEP entry.



For details, refer to chapter 4 of Sol 400 UG (refer to the exact section)

### **Convergence Controls**

Three methods are available for determining if convergence is obtained on any given iteration: residual force, displacement, and strain energy. You can select one of these three criteria for convergence or you may specify a combination of residual and displacement. The AND combination signals that both residual and displacement must be met, while the OR combination specifies that either one can satisfy convergence criteria. If you are using residual there may be cases in which the force residuals are null in which case is it necessary to switch over to displacement. An Autoswitching option (on by default) allows for this switching.

The default measure for convergence in SOL 400 is residual which is based on the magnitude of the maximum residual load compared to the maximum reaction force. This method is appropriate since the residuals measure the out-of-equilibrium force, which should be minimized. This technique is also appropriate for Newton methods, where zero-load iterations reduce the residual load. The method has the additional benefit that convergence can be satisfied without iteration. You have complete control over how convergence is defined through the Iterations Parameters form in Patran or through the options on the NLSTEP entry.

The basic procedures are outlined below.

- 1. Residual checking: Residual checking has one drawback. In some special problems, such as free thermal expansion, there are no reaction forces. If the value of the residuals and reactions is less than 1.e-6, this test is ignored. If the AUTOSW flag on the NLSTRAT entry is ON, the program automatically uses displacement checking in this cases.
- 2. Displacement checking: With this method, convergence is satisfied if the maximum displacement of the last iteration is small compared to the actual displacement change of the increment. If the value of the incremental and iterative displacement is less than 1.e-8, this test is ignored. A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution.
- 3. Strain energy checking: This is similar to displacement testing where a comparison is made between the strain energy of the latest iteration and the strain energy of the increment. With this method, the entire model is checked. A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution. The advantage of this method is that it evaluates the global accuracy as opposed to the local accuracy associated with a single node.
- 4. Residual or displacement checking: This procedure does convergence checking on both residuals (Residual 1) and displacements (Procedure 2). Convergence is obtained if one converges.
- 5. Residual and displacement checking: This procedure does a convergence check on both residuals and displacements (Procedure 4). Convergence is achieved if both criteria converge simultaneously.

Different problems require different schemes to detect the convergence efficiently and accurately. To do this, the combinations of residual checking and displacement checking are also available (as mentioned in the last two steps.



### **Solution Parameters**

Solution parameters control a range of functions in the SOL 400 analysis. Functions such as selecting the solver type, establishing a restart, specifying domain decomposition are all part of the solution parameters.

### **Defining Solution Parameters in Patran**

To set solution parameters:

- 1. Click on Analysis Application to open the Analysis Application form.
- 2. On the Analysis Application form, click Solution Type... and select Implicit Nonlinear. Click Solution Parameters.

| Solution Parameters                                       | . • X   |
|---|---|
| Solve   | ers / Options   |
| Contac  | ct Parameters   |
| Direc   | ct Text Input   |
| Restar  | rt Parameters   |
| Advano  | ed Job Control  |
| Domain  | Decomposition   |
| Assumed Strain  | Constant Dilatation<br>Reduced Integration<br>Shell Shear Correction<br>SoL 400 Run |
| Default Initial Temperature:<br>Default Load Temperature: |   |
| Results   | Output Format   |
| ок  | Defaults Cancel   |

| Solver Options            | Specifies the solver to be used in numerically inverting the system of linear equilibrium equations.  |
|---------------------------|---|
| Contact Parameters        | Defines options for detecting and handling contact.   |
| Direct Text Input         | This subform is used to directly enter entries in the File Management,<br>Executive Control, Case Control, and bulk data sections of the MSC<br>Nastran input file. |
| <b>Restart Parameters</b> | Includes a <b>Restart</b> option in the MSC Nastran input file.   |
| Advanced Job<br>Control   | Sets alternate versions of the solver and alternate formats for the results file.   |
| Domain<br>Decomposition   | Designates that domain decomposition be done manually, semi-automatically, or automatically.  |


#### lect Group(s)/SET LOADSTEP NAME: Default SOLUTION SEQUENCE: 400 fault\_group Form Type: Advanced 🔻 Select Result Type Displacements Option Sorting Element Stresses Constraint Forces Mutit-Point Constraint Forces Element Forces Applied Loads Nonlinear Applied Loads Element Strain Energies By Node/Element Format: Rectangular · Tensor Von Mises 💌 Element Points Bilinear 🔻 Plane & Curv Composite Plate Opt Ply Stresses \* DISPLACEMENT(SORT1,REAL)=AI FEM Suppress Print for Result Type SPCFORCES(SORT1.REAL)=AILFEM Intermediate Output Option No\* Delete SUBTITLE Default This load case is the default load case that always ap OK Defaults Cancel

## **Requesting Output For a Step**

- Creating a step includes requesting output.
- You must suppress your output as much as possible. (As nonlinear jobs can quickly fill up your disk).
  - Usually you will output results at every so many increments to reduce the size of the output file.
  - Results are written in file job\_id.op2/master/dball, which must be read back for postprocessing, including visualizing results

## **Defining Subcases in Patran**

To define a subcase:

- 1. Click on the Analysis Application button to bring up the Analysis Application form.
- 2. From the Analysis Application form click Subcases.

## **Define History by Selecting Load Steps**

- Creating a step includes selecting load cases.
- Create the load steps to define the load history. The load step selection order specifies the order in which loads, and boundary conditions are applied.
- Make sure that you are specifying total load, and not the incremental load. Put all loads in the load case that belong to the step, even those you already used in the previous general step.
- Only those steps that are selected will be included in the analysis.



| Static S                     | Solution Parameters     |  |
|------------------------------|-------------------------|--|
| Linearity:                   | NonLinear 🔻             |  |
|                              | Solvers / Options       |  |
| Nonlinear Geometric Effects: |                         |  |
| Large Dis                    | placement/Large Strains |  |
| Follower Loads:              |                         |  |
| Loads Fo                     | llow Deformations 🔹     |  |
|                              |                         |  |
| Load Increment Params        |                         |  |
| Iteration Parameters         |                         |  |
| Brake Squeal Parameters      |                         |  |
| OF                           | C Defaults Cancel       |  |

| Subcase Name                                      | Specifies a name for a new subcase.   |
|---|---|
| Available Load Cases                              | Selects one or more available load cases to be applied to the new subcase.  |
| Subcase Options                                   |   |
| <ul> <li>Subcase</li> <li>Parameters</li> </ul>   | Controls load increment and iteration parameters for the subcase.<br>Also defines the nonlinear effects for the subcase.  |
| <ul> <li>Output Requests</li> </ul>               | Defines the nodal and element results quantities and also determines<br>the frequency of results reporting.   |
| Direct Text Input                                 | This subform is used to directly enter entries in the File Management,<br>Executive Control, Case Control, and Bulk Data Sections of the MSC<br>Nastran input file. |
| <ul> <li>Select</li> <li>Superelements</li> </ul> | Defines which superelements should be included in the subcase.  |
| <ul> <li>Select Explicit<br/>MPCs</li> </ul>      | Selects explicit MPCs to be included in the subcase.  |

#### **Subcase Parameters**

The subcase parameters represent the settings in MSC Nastran Case Control and Bulk Data Section that take effect within a subcase and do not affect the analysis in other subcases. Subcase parameters are dependent on the type of analysis being performed. The set of subcase parameters applicable for each analysis type are described in the following sections. For more information, see Chapter 3: Solution Methods and Strategies in Nonlinear Analysis in the *MSC Nastran Implicit Nonlinear User's Guide*.



## **Static Subcase Parameters**

For static nonlinear analysis, the subcase parameters control the iteration process and the load incrementation.

| Entry  | Description  |  |
|--------|--|--|
| NLSTEP | Defines parameters for automatic load/time stepping used in SOL 400.   |  |
| NLPARM | Nonlinear Static Analysis Parameter Selection.   |  |
| NLPCI  | Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis. |  |
|        | <b>Note:</b> The arc length method cannot be used with contact.  |  |

#### **Defining Static Subcase Parameters in Patran**

- 1. Click Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases, and choose Static from the Analysis Type pull-down menu.
- 3. Click Subcase Parameters

| Linearity:  | NonLinear  |
|-------------|--|
|             | Solvers / Options  |
| Noninear (  | Seometric Effects:   |
| Nonlinear G | eometric Effects:  |
| Large Disp  | lacement/Large Strains   |
| FollowerLo  | ade.   |
| Landa Fall  | ous.   |
|             |  |
| Use Con     | tact Table   |
| Use Con     | tact Table   |
| Use Con     | tact Table Load Increment Params Eeration Parameters   |
| Use Con     | tact Table Load Increment Params Renation Parameters Contact Table   |
| Use Con     | tect Table Load Increment Paramo Renation Paramoters Context Table ActiveCreactive Denterts                  |
| Use Con     | text Table Load Increment Parama Revalue Parama Context Table ActiveCreacitye Demonts Brake Squed Parameters |

| Linearity   | Prescribes the nonlinear effects for the subcase.                                     |
|---|---|
| Nonlinear Solution<br>Parameters                    |   |
| <ul> <li>Nonlinear<br/>Geometric Effects</li> </ul> | Defines the type of geometric or material nonlinearity to be included in the subcase. |
| <ul> <li>Follower Forces</li> </ul>                 | Specifies whether forces will follow displacements.                                   |
|   |   |



| Load Increment Params | Defines whether the load increments will be fixed or adapted in each iteration, and the method by which adaptive load increments will be determined. |
|-----------------------|--|
| Iteration Parameters  | Sets forth the iterative procedures that are employed to solve the equilibrium problem at each load increment.                                       |
| Contact Table         | Activates, deactivates, and controls the behavior of contact bodies in the analysis.   |

## **Specifying Creep Subcase Parameters**

The creep analysis option is activated in MSC Nastran through the NLSTEP bulk data entry. The creep time period and control tolerance information are input through the as usual on the NLSTEP form. This option can be used repeatedly to define a new creep time period and new tolerances. These tolerances are defined in the section on Iteration Parameters. Alternatively, a fixed time step can also be specified. In this case, no additional tolerances are checked for controlling the time step.

| Entry   | Description   |
|---------|---|
| NLSTEP  | Nonlinear static analysis parameter selection for doing creep analysis.   |
| MATVPMA | Defines creep characteristics based on experimental data or known empirical creep law.<br>This material definition should be used with advanced elements.     |
| CREEP   | Defines creep characteristics based on experimental data or known empirical creep law.<br>This material definition should be used with conventional elements. |

#### **Defining Creep Subcase Parameters in Patran**

- 1. Click the Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases... and choose Creep from the Analysis Type pull-down menu.

| Creep Solution Parameters                             | Description   |  |
|---|---|--|
| <ul> <li>Procedure</li> </ul>                         | Selects implicit or explicit creep method.  |  |
| <ul> <li>Nonlinear Geometric<br/>Effects</li> </ul>   | Defines the type of geometric or material nonlinearity to be included in the subcase. |  |
| <ul> <li>Follower Forces</li> </ul>                   | Specifies whether forces will follow displacements.                                   |  |
| Increment Type  | Defines a fixed or adaptive increment method.   |  |
| <ul> <li>Adaptive Increment<br/>Parameters</li> </ul> | For adaptive methods, sets boundaries for incrementation.                             |  |



| Iteration Parameters | Sets forth the iterative procedures that are employed to solve the equilibrium problem at each load increment. |
|----------------------|--|
| Contact Table        | Activates, deactivates, and controls the behavior of contact bodies in the analysis.                           |

## **Normal Modes Subcase Parameters**

For normal modes nonlinear analysis, the subcase parameters control the eigenvalue extraction techniques and the range of frequencies to be targeted for extraction.

| Entry | Description  |
|-------|--|
| EIGRL | Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method (recommended). |
| EIGR  | Defines data needed to perform real eigenvalue analysis.   |

#### **Defining Normal Modes Subcase Parameters in Patran**

- 1. Click the Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases... and choose Normal Modes from the Analysis Type pull-down menu.
- 3. Click Subcase Parameters...

| Normal Modes Solution Parameters                             | Normal Modes Solution Parameters                          |
|--|---|
| Extraction Method: Lanczos 🔻                                 | Extraction Method: Inverse Power                          |
| Lanczos Parameters:<br>Lower Frequency (Hz):                 | Inverse Power Sweep Parameters:     Lower Frequency (Hz): |
| Upper Frequency (Hz):  | Upper Frequency (Hz):                                     |
| Estimated # of Roots: 100                                    | Estimated # of Roots: 100                                 |
| Desired # of Roots: 10                                       | Desired # of Roots: 10                                    |
| Diagnostic Output Level: 🗸 0                                 | Desired # of Positive Roots 1                             |
| 1 2  | Desired # of Negative Roo 1                               |
| Results Normalization 3<br>Normalization Method: Mass V Mass | Results Normalization<br>Normalization Method: Mass       |
| Normalization Point = Maximum Point                          | Normalization Point = Vialing                             |
| Normalization Component                                      | Normalization Component                                   |
| OK Defaults Cancel   | OK Defaults Cancel  |

Defines the method to use to extract the real eigenvalues.



Extraction Method

Number of Modes
 Indicates an estimate of the number of eigenvalues to be located.



| <ul> <li>Lowest/Highest Frequency</li> </ul> | Defines the lower and upper limits to the range of frequencies to be examined.   |  |
|--|--|--|
| Sequence Checking                            | Requests that Sturm sequence checking be performed on the extracted eigenvalues. |  |

## **Specifying Transient Dynamic Subcase Parameters**

For transient dynamic nonlinear analysis the subcase parameters control the iteration process and the load incrementation.

| Entry   | Description   |
|---------|---|
| NLSTEP  | Parameters for automatic load/time stepping for both static and transient nonlinear analysis (recommended). |
| TSTEPNL | Traditional way to specify nonlinear dynamic analysis parameter selection, has been replaced by NLSTEP      |

#### **Defining Transient Dynamic Subcase Parameters in Patran**

- 1. Click the Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases... and choose Transient Dynamic from the Analysis Type pulldown menu.
- 3. Click Subcase Parameters...

| Transient Dynamic Solution Param  | Transient Dynamic Solution Param   |
|---|--|
| Linearity: NonLinear V  | Linearity: Linear •  |
| Time Integration Method: Direct   | Time Integration Method: V Direct  |
| Nonlinear Geometric Effects:  | Nonlinear Geometric Effects:   |
| ✓       None (Small Displacements and Strains)         Large Displacement/Large Strains         Loads Follow Deformations | None (Small Displacements and Strains)         Follower Loads:           Loads Follow Deformations |
| Load Increment Params   | Load Increment Params  |
| Iteration Parameters  | Iteration Parameters   |
| Brake Squeal Parameters   | Brake Squeal Parameters  |
| OK Defaults Cancel  | OK Defaults Cancel   |

Linearity

Prescribes the nonlinear effects for the subcase.

Nonlinear Solution Parameters



| <ul> <li>Nonlinear Geometric<br/>Effects</li> </ul> | Defines the type of geometric or material nonlinearity to be included in the subcase.   |
|---|---|
| <ul> <li>Follower Forces</li> </ul>                 | Specifies whether forces will follow displacements.   |
| Load Increment Params                               | Defines whether the load increments will be fixed or adapted in<br>each iteration and the method by which adaptive load<br>increments will be determined. |
| Iteration Parameters                                | Sets forth the iterative procedures that are employed to solve the equilibrium problem at each load increment.  |
| Contact Table                                       | Activates, deactivates, and controls the behavior of contact bodies in the analysis.  |

#### **Specifying Body Approach Subcase Parameters**

For body approach analysis the subcase parameters control the iteration process and the load incrementation.

| Entry  | Description                           |
|--------|---------------------------------------|
| BCMOVE | Specifies movement of rigid surfaces. |

#### **Defining Body Approach Subcase Parameters in Patran**

- 1. Click the Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases... and choose Body Approach from the Analysis Type pull-down menu.
- 3. Click Subcase Parameters...

| 1.0       |                          |
|-----------|--------------------------|
| Synchron  | zed                      |
|           |                          |
| Use Contr | ict Table                |
|           | Load Increment Params    |
|           | Iteration Parameters     |
|           | Contact Table            |
|           | Active/Deactive Elements |
|           |                          |

**Body Approach Solution Parameters** 

• Total Time Places a time step option in the Load Step.



| <ul> <li>Synchronized</li> </ul> | If ON, specifies that when the first rigid body comes into contact, the rest stop moving. |
|----------------------------------|---|
| Contact Table                    | Activates, deactivates, and controls the behavior of contact bodies in the analysis.      |

## Specifying Complex Eigenvalue Subcase Parameters in Patran

For transient dynamic nonlinear analysis the subcase parameters control the iteration process and the load incrementation.

#### **Defining Complex Eigenvalue Subcase Parameters in Patran**

- 1. Click the Analysis Application button to bring up Analysis Application form. Click on Solution Type and check to see that Implicit Nonlinear is the selected Solution Type, then click OK.
- 2. On the Analysis form select Subcases... and choose Complex Eigenvalue from the Analysis Type pull-down menu.
- 3. Click Subcase Parameters...



| Formulation         | Specifies whether a direct or modal superposition solution will be performed. |
|---------------------|---|
| Solution Parameters |   |
| Complex Eigenvalue  | Opens Complex Eigenvalue Extraction input parameters form.                    |



|                               | Eigenvalue Extraction                                       |
|-------------------------------|---|
|                               | COMPLEX EIGENVALUE EXTRACTION                               |
|                               | Extraction Method: Complex Lanczos                          |
|                               | Upper Hessenberg  |
|                               | Alpha of Point A =  |
|                               | Omega of Point A = 0.0                                      |
|                               | Alpha of Point B = 10.0                                     |
|                               | Omega of Point B = 10.0                                     |
|                               | Width of Region = 1.0                                       |
|                               | Estimated Number of Roots =                                 |
|                               | 50  |
|                               | Number of Desired Roots =                                   |
|                               |   |
|                               | Results Normalization                                       |
|                               | Normalization Method: Point                                 |
|                               |   |
|                               | Normalization Point =                                       |
|                               |   |
|                               |   |
|                               | Convergence Oriteria -                                      |
|                               |   |
|                               | OK  |
|                               |   |
| Initial Condition Load Factor | Multiplier applied to loads and constraints used as initial |
|                               | conditions of complex eigenvalue analysis.                  |
| Enable Rotor Dynamics         | Toggle this ON to enable Specify Spinning Properties        |
| 5                             | form which allows user to input rotordynamics               |
|                               | ionin which allows user to input fotordynamics              |
|                               | properties.   |
| Contact Table                 | Toggle Use Contact Table ON to use of contact table         |
|                               | with on them, contract nation in the englyric Ones togolad  |
|                               | rather than contact pairs in the analysis. Once toggied     |
|                               | ON, the Contact Table button brings up contact table        |
|                               | input form.   |
| Produ Saucal                  | Allows user to input nonemators used for here 11            |
| Dreak Squeal                  | Allows user to input parameters used for break squeal       |
|                               | analysis.   |

## **Specifying Frequency Response Subcase Parameters**

When setting up a frequency response analysis in Patran it is necessary to use the same menus as if you were setting up a SOL 108 or 111 analysis, and then manually edit the executive input command lines to specify SOL 400 as the solution type.

## **Thermal Solution Subcases**

Three solution types are supported in the Patran MSC Nastran thermal interface:

- Steady state.
- Transient.



• Structural-thermal analysis.

These analysis types employ nonlinear solution algorithms so that nonlinear material properties or boundary conditions can be included in the model.



# **11** Trouble Shooting

Overview 182 **Review Fatal Error Message** 182 Review the .sts File 183 Review the .f06 File 184 SOL 400 Analysis Messages 188 Reviewing Convergence 190 **Review Plot Results from Converged Increments** Reviewing the MSC Analysis Manager 193 Tips for Starting with Nonlinear Analysis 194 Review Fatal Error Message 182 

192



#### **Overview**

Sometimes you may receive non-convergence error messages when a job does not converge. In such a scenario, review the following diagnostics tools to help you:

- The fatal error message
- Plot results from converged increments
- The .sts file
- The .f06 file
- The .log file

## **Review Fatal Error Message**

Even with a fatal error message, there are still insights to be gained from the completed increments.

#### Nastran 2013



For details, go the end of the . £06 file and search up for %. The nonlinear iteration printout will provide helpful diagnostics on your job

| <sup>0</sup> Fraction of               |            |        |       |        |            |           |          | SUBCASE       | 1         | STEP 1   |       |     |     |
|--|------------|--------|-------|--------|------------|-----------|----------|---------------|-----------|----------|-------|-----|-----|
| 0 NON-                                 | LINEA      | RI     | TER   | AT     | ION M      | ODULE     | OUTP     | UT            |           |          |       |     |     |
| load step                              |            |        |       |        |            |           |          |               |           |          |       |     |     |
| STIFFNESS UPDATE TIME 0.               | 01 SECONDS | 3      |       |        |            |           | SUBCA    | SE 1          | ST        | EP       | 1     |     |     |
| ITERATION TIME 0.                      | 06 SECONDS | 5      |       |        |            |           |          | May di        | ien       |          |       |     |     |
| Load Erro                              | or Facto   | r      |       |        |            |           |          | Max u         | isp.      |          |       |     |     |
| LOAD NO ERROR FACT                     | TORS       | CONV   | ITR   | MAT N  | IO. AVG    | TOTL      |          | DISP -        |           | LINE S   | NO.   | TOT | TOT |
| STEP INC ITR DISP LOAD                 | WORK       | RATE   | DIV   | DIV B  | IS R_FORC  | E WORK    | AVG      | MAX           | AT GRID C | FACT N   | O QNV | KUD | ITR |
| \$1.00000E-01 1 1 1.00E+00 6.62E-02    | 1.00E+00   | 1.000  | 0     | 1 0    | 3.66E-01   | 3.753E-01 | 3.98E-03 | -3.745E-02    | 399 3     | 1.00 0   | 0     | 0   | 1   |
| SISIEM INFORMATION MESSAGE 8137        | NL3CON)    |        |       |        |            |           |          |               |           |          |       |     |     |
| SEPARATION condition has been de       | ected. Add | itiona | l ite | eratio | ns will be | performed |          |               |           |          |       |     |     |
| \$1.00000E-01 1 2 3.85E-01 6.42E-03    | 6.46E-01   | 0.097  | 0     | 1 0    | 2.31E-02   | 2.555E-01 | 2.93E-03 | -2.704E-02    | 395 3     | 1.00 0   | 0     | 1   | 2   |
| *** SYSTEM INFORMATION MESSAGE 8137    | NL3CON)    |        |       |        |            |           |          | 221.002300500 |           | 1000     |       |     |     |
| SEPARATION condition has been de       | ected. Add | itiona | 1 114 | eratio | ns will be | performed | 1.1      |               |           |          |       |     |     |
| \$1.00000E-01 1 3 4.36E-02 5.37E-04    | 5.32E-02   | 0.096  | 0     | 1 0    | 6.11E-04   | 2.501E-01 | 2.80E-03 | -2.603E-02    | 400 3     | 1.00 0   | 0     | 2   | 3   |
| \$1.00000E-01 1 4 8.27E-02 1.58E-02    | 3.76E-02   | 0.936  | 0     | 1 0    | 6.20E-03   | 2.668E-01 | 2.96E-03 | -2.830E-02    | 395 3     | 1.00 0   | 0     | 3   | 4   |
| *** SYSTEM INFORMATION MESSAGE 8137    | NL3CON)    |        |       |        |            |           | 12020200 |               |           | 0.10.010 |       |     |     |
| SEPARATION condition has been de       | ected. Add | itiona | 1 110 | ratio  | ns will be | performed |          |               |           |          |       |     |     |
| \$1.00000E-01 1 5 5.97E-05 1.18E-05    | 1.30E-03   | 0.597  | 0     | 1 0    | 2.88E-06   | 2.666E-01 | 2.96E-03 | -2.830E-02    | 400 3     | 1.00 0   | 0     | 4   | 5   |
| *** SYSTEM INFORMATION MESSAGE 8137    | NL3CON)    |        |       |        |            |           |          |               |           |          |       |     |     |
| SEPIRATION condition has been de       | ected, Add | itiona | 1 110 | ratio  | ns will be | performed | 28       |               |           |          |       |     |     |
| \$1.00000E-01 1 6 9.64E-03 1.44E-04    | 3.94E-03   | 0.010  | 0     | 1 0    | 1.355-04   | 2.684E-01 | 2.985-03 | -2.857E-02    | 395 3     | 1.00 0   | 0     | 5   | 6   |
| \$1.0000 E-01 1 7 3.88E-03 7.02E-05    | 9.462-04   | 0.637  | 0     | 1 0    | 6.57E-05   | 2.688E-01 | 2.98E-03 | -2.868E-02    | 395 3     | 1.00 0   | 0     | 6   | 7   |
|  |            |        |       | -      |            |           |          |               |           |          |       |     |     |
| time increment has been                | changed to | 1      | .2000 | 00E-01 |            |           |          |               |           |          |       |     |     |
| Contact previous time increment        |            | 1      | .0000 | 00E-01 |            |           |          |               |           |          |       |     |     |
| Senarations total period to be cove    | red        | 1      | .0000 | 00E+00 |            |           |          |               |           |          |       |     |     |
| total period covered as                | far        | 1      | .0000 | 0E-01  |            |           |          |               |           |          |       |     |     |
| due to number of requile               |            | -      |       |        |            |           |          |               |           |          |       |     |     |
| due to number or response              |            |        |       |        |            |           |          |               |           |          |       |     |     |
| \$2,20000E-01 2 1 1,00E+00 1,70E-01    | 1.00E+00   | 1.000  | 0     | 1 0    | 1.835-01   | 1.039E+00 | 5.02E-03 | -5.102E-02    | 395 3     | 1.00 0   | 0     | 7   | 8   |
| *** SYSTEM INFORMATION MESSAGE 8137    | (NL3CON)   |        | -     | -      |            |           |          | CTARE OF      | 520 0     |          |       |     | -   |
| SEPARATION condition has been det      | ected. add | itiona | 1 174 | TATIC  | na will be | nerformed |          |               |           |          |       |     |     |
| \$2,20000F=01 2 2 3,80F=01 4,50F=03    | 1.455-01   | 0.026  | 0     | 1 (    | 9.085-03   | 9.5048-01 | 4.505-03 | -4.4888-02    | 395 3     | 1.00 0   | 0     | 8   | 9   |
| \$2 20000E-01 2 3 2 51E-02 4 02E-05    | 0 505-03   | 0.026  | 0     | 1 1    | 8 095-05   | 0 4445-01 | 4 448-03 | -4 4495-02    | 395 3     | 1 00 0   | 0     | 0   | 10  |
| ************************************** | 2.026-03   | 0.020  | ~     |        | 0.092-03   | *****E=01 | 1.142-03 | -111102-02    | 330 3     | 1.00 0   |       | -   | **  |



The % symbol does not mean percent. It is a convenient way for you to search for the iteration printout. A few key things to look for are highlighted in the figure.

## Review the .sts File

SOL 400 produces a status file jobname.sts hat can be queried periodically to see how the analysis is progressing and if the job is completed. It is very useful for assessing if the analysis has completed successfully, and, if not, how much of it was completed before the analysis was terminated. This is extremely useful because the most common problem with obtaining nonlinear solutions is avoiding, detecting, and managing convergence problems.

The easiest way to have a real-time monitor of your SOL 400 job is to open the .sts file with a text editor that automatically updates when the file is changed.

The information in this file is especially important when manual or automatic time stepping procedures are being used to step through an analysis procedure. One line is written after each successful increment. A typical STS file is shown as follows. Its content is well self-explained by the file itself. The file below may be slightly different than the final version.

| informatic<br>version: M<br>date:  | on summar<br>ISC Nastr<br>Jul 15,  | y of 1<br>an 201<br>2014;   | job:<br>L4.0.0<br>Day T   | ./nlcO:<br>, Built<br>ime: 10  | LOa<br>t on Jul<br>5:11:26   | 10, 201  | 4  |  |   |   |  |   |  |   |
|--|--|---|---|--|--|--|--|--|---|---|--|---|--|---|
| aate:<br>subcase<br>/step #<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1<br>1 | Jul 15,<br>inc<br>#<br>0<br>1<br>2<br>3<br>4<br>5<br>6<br>7<br>8<br>9<br>10<br>vith exit<br>total w<br>total c | cycl<br>#<br> of<br>0<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2<br>2 | sepa<br>#<br>the ii<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | rme: 10<br>cut<br>mc <br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | cyc1<br>#<br>0<br>2<br>4<br>6<br>8<br>10<br>10<br>12<br>14<br>16<br>18<br>20<br>2.00<br>1.12 | split<br>#<br>of the<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | separ<br>#<br>analysi:<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | cut<br>#<br>5<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | rmesh<br>#<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0<br>0 | time step<br>of<br>  the inc<br>0.000E+01<br>1.0000E-01<br>1.0000E-01<br>1.0000E-01<br>1.0000E-01<br>1.0000E-01<br>1.0000E-01<br>1.0000E-01 | total time<br>of<br>the job<br>0.000E+01<br>2.0000E-01<br>3.0000E-01<br>5.0000E-01<br>7.0000E-01<br>8.0000E-01<br>1.0000E-01<br>1.0000E+00 | wall time<br>1.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00<br>2.00 | cpu time<br>0.68<br>0.70<br>0.75<br>0.79<br>0.83<br>0.87<br>0.91<br>0.95<br>1.00<br>1.04<br>1.08 | max resp. type<br>0.0000E+00 disp<br>-1.5000E-02 disp<br>-3.0000E-02 disp<br>-4.5000E-02 disp<br>-7.5000E-02 disp<br>-7.5000E-02 disp<br>-1.0500E-01 disp<br>-1.3500E-01 disp<br>-1.5000E-01 disp |
|  | exit DE<br>= 0 jo<br>= 1 jo  | b tern<br>b tern<br>b tern  | inate<br>ninate   | s norma<br>s abnor   | ally<br>rmally (   | check Fa   | tal Erron  | r Mess   | age in  | F06)  |  |   |  |   |

The first column shows the procedural step, while the second column shows the increment within the step. Not every increment size is equal, as can be seen in the TIME STEP OF INC column. The third column (cycl #) indicates the number of attempts made during this increment.

The status file is updated in real time so a running job can be monitored. Review the .sts file. Check for the following:

- Did the job finish normally?
- How far did the job get to?
- What is the maximum displacement?
  - Is the job running away?
  - Does the maximum displacement make sense?I



- Did it have a lot of cutbacks?
  - Are there many cutbacks (bi-sections)?
  - Are the increments are too large?
- Did it have a lot of separations
  - Are there many separations (contact analysis)?
  - Investigate the contact parameters.
- What is the total number of cycles?
  - What is the total number of cycles (iterations?
  - Is this equal to the number of matrix assembly and decompositions?
  - Can I reduce this to improve run time?



Also review the exit definition. It will inform you if the job terminated normally or abnormally. You can then check the .f06 file for more information.

Patran provides a convenient way to access the status file using the Monitor option of the Analysis form.

#### Review the .f06 File

The keyword search option in the . £06 file makes it easy to find errors in the SOL 400 run by searching the file for fatal message to immediately find the fatal error that terminated the job).

When reviewing the . f06 file, do the following:

- Review NLSTEP parameters
- Nonlinear iteration diagnostics
- Request for more diagnostics
- Review the intermediate results



## **Review NLSTEP parameters**

Check what NLSTEP parameters were used in a run. The following example shows a part of the .f06 file. It gives an idea of what was used in a run.

| LOC | OP CONTROLS FOR : SUBCASE 1,                       | STEP      | 1,   |
|-----|--|-----------|------|
| SOI | LUTION CONTROL PARAMETERS FROM : NLSTEP            |           | ID : |
|     | Total Time of Loading Case (TOTTIME)               | 1.00E+00  |      |
|     | Maximum Number of Iterations (MAXITER)             | 10        |      |
|     | Minimum Number of Iterations (MINITER)             | 1         |      |
|     | Maximum Number of Bisection (MAXBIS)               | 10        |      |
|     | Creep Option (CREEP)                               | 0         |      |
|     | Initial Time Step Fraction (DTINITF)               | 1.00E-01  |      |
|     | Minimum Time Step Fraction (DTMINF)                | 1.00E-05  |      |
|     | Maximum Time Step Fraction (DTMAXF)                | 5.00E-01  |      |
|     | Desired Number of Iterations (NDESIR)              | 4         |      |
|     | Factor for Time Step Increase (SFACT)              | 1.20E+00  |      |
|     | Output Flag (INTOUT)                               | 0         |      |
|     | Maximum Number of Increments (NSMAX)               | 99999     |      |
| )   | Option of Artificial Damping (IDAMP)               | 2 005 04  |      |
|     | Damping Ratio (DAMP)                               | 2.002-04  |      |
|     | Identification of TABSCIL (CRITID)                 | 0         |      |
|     | Option of Hear Criteria (LIMITAR)                  | 2         |      |
|     | Smalleer Dario (DEMAIL)                            | 1 005-01  |      |
|     | Largest Patio (PBIG)                               | 1.00E+01  |      |
|     | Time Step Skip Factor (ADJUST)                     | 1.002.002 |      |
|     | Number of Steps for Dominant Resp. (MSTEP)         | 10        |      |
|     | Bounds of Time Stepping (RB)                       | 6.00E-01  |      |
|     | Tolerance on Displacement (UTOL)                   | 1.00E+00  |      |
|     | Convergence Criteria (CONV)                        | P V       |      |
|     | - Displacement (EPSU)                              | -1.00E-02 |      |
|     | Tolerance - Residual Force (EPSP)<br>- Work (EPSW) | 1.00E-02  | 1    |
|     | Option of Rotations and Moments (MRCONV) .         | 3         |      |
|     | Matrix Update Option (KMETHOD)                     | PFNT      |      |
|     | Matrix Update Increment (KSTEP)                    | 1         |      |
|     | Maximum Quasi-Newton Vectors (MAXQN)               | 0         |      |
|     | Maximum Line Searches (MAXLS)                      | 0         |      |
|     | Line Search Tolerance (LSTOL)                      | 5.00E-01  |      |
|     | Error Tolerance in VE (ESTDESS)                    | 2 00F-01  |      |

In the .f06 file, there is another shorter summary of NLSTEP parameters right above this printout. That one is a subset of this one and is not as useful.

## **Review Nonlinear Iteration Diagnostics**

To know how is the job progressing, search for the % character. The nonlinear iteration printout provides helpful diagnostics on your job.



| <sup>o</sup> Fraction | of               |           |             |         |      |        |        |       |            |          | SUBCASE    | 1       | STI          | EP 1  |     |     |     |
|-----------------------|------------------|-----------|-------------|---------|------|--------|--------|-------|------------|----------|------------|---------|--------------|-------|-----|-----|-----|
| load stor             |                  | NON-      | LINEA       | RI      | TE   | RAT    | ION    | M     | DDULE      | OUTP     | UT         |         |              |       |     |     |     |
| Ivau step             |                  |           |             |         |      |        |        |       |            |          |            |         |              |       |     |     |     |
| STITENL               | 55 OPDAIL TIME   | 0         | . OI SECOND | 5       |      |        |        |       |            | SUBUR    | 1 36       | 10000   | DIFF         |       | -   |     |     |
| /                     | ITERATION TIME   | and Fre   | .06 SECOND  | s       |      |        |        |       |            |          | Max di     | SD.     |              |       |     |     |     |
| 1                     |                  | oad En    | or Facto    | N.      |      |        |        |       |            |          | _          |         |              | _     |     |     |     |
| LOAD                  | NO               | ERROR FAC | TORS        | CONV    | ITR  | MAT N  | 0.     | AVG   | TOTL       |          | DISP -     |         | LI           | NE_S  | NO. | TOT | TOT |
| STEP                  | INC ITR DISP     | LOAD      | WORK        | RATE    | DIV  | DIV 8  | IS R   | FORC  | E WORK     | AVG      | MAX        | AT GRID | CEA          | CT NO | QNT | KUD | ITP |
| \$1.00000E-01         | 1 1 1.00E+00     | 6.62E-02  | 1.00E+00    | 1.000   | 0    | 1 0    | 3.66   | 5E-01 | 3.753E-01  | 3.98E-03 | -3.745E-02 | 399     | 3 1.         | 0 00  | 0   | 0   | 1   |
| SISIER .              | INFORMATION MESS | AGE 8137  | NL3CON)     |         |      |        |        |       |            |          |            |         |              |       |     |     |     |
| SEPARAT               | ION condition ha | been de   | ected. Ad   | ditions | l it | eratio | ns wil | 11 be | performed. |          |            |         |              |       |     |     |     |
| \$1.00000E-01         | 1 2 3.85E-01     | 6.42E-03  | 6.46E-01    | 0.097   | 0    | 1 0    | 2.31   | LE-02 | 2.555E-01  | 2.93E-03 | -2.704E-02 | 395     | 3 1./        | 0 00  | 0   | 1   | 2   |
| *** SYSTEM            | INFORMATION MESS | AGE 8137  | NL3CON)     |         |      |        |        |       |            |          |            |         |              |       |     |     |     |
| SEPARAT               | ION condition ha | been de   | ected. Ad   | ditions | 1 10 | eratio | ns wil | 11 be | performed. |          |            |         |              |       |     |     |     |
| \$1.00000E-01         | 1 3 4.36E-02     | 5.37E-04  | 5.32E-02    | 0.096   | 0    | 1 0    | 6.11   | LE-04 | 2.501E-01  | 2.80E-03 | -2.603E-02 | 400     | 3 1./        | 0 00  | 0   | 2   | 3   |
| \$1.00000E-01         | 1 4 8.27E-02     | 1.58E-02  | 3.76E-02    | 0.936   | 0    | 1 0    | 6.20   | DE-03 | 2.668E-01  | 2.96E-03 | -2.830E-02 | 395     | 3 1.         | 0 00  | 0   | 3   | 4   |
| *** SYSTEM            | INFORMATION MESS | GE 8137   | NL3CON)     |         |      |        |        |       |            |          |            |         | - <b>-</b> - |       |     |     |     |
| SEPARAT               | ION condition a  | been de   | ected. Ad   | ditions | 1 11 | eratio | ns wil | 11 be | performed. | S        |            |         |              |       |     |     |     |
| \$1.00000E-01         | 1 5 5.9/2-05     | 1.18E-05  | 1.30E-03    | 0.597   | 0    | 1 0    | 2.88   | 8E-06 | 2.666E-01  | 2.96E-03 | -2.830E-02 | 400     | 3 1.         | 0 00  | 0   | 4   | 5   |
| *** SYSTEM            | INFORMATION MESS | GE 8137   | NL3CON)     |         |      |        |        |       |            |          |            |         |              |       |     |     |     |
| SEPARAT               | ION condition ha | been de   | ected. Ad   | ditions | 1 10 | eratio | ns wil | ll be | performed. | 3        |            |         |              |       |     |     |     |
| \$1.00000E-01         | 1 6 9.64E-03     | 1.448-04  | 3.94E-03    | 0.010   | 0    | 1 0    | 1.35   | 5E-04 | 2.684E-01  | 2.98E-03 | -2.857E-02 | 395     | 3 1./        | 0 00  | 0   | 5   | 6   |
| \$1.00000E-01         | 1 7 3.88E-03     | 7.02E-05  | 9.46E-04    | 0.637   | 0    | 1 0    | 6.51   | 7E-05 | 2.688E-01  | 2.98E-03 | -2.868E-02 | 395     | 3 1.         | 0 00  | 0   | 6   | 7   |
| 1                     |                  | -         |             |         |      |        |        |       |            |          |            |         | /            |       |     |     |     |
| Contract              | time increment   | has been  | changed t   | 0 1     | .200 | 00E-01 |        |       |            |          |            |         | 24           |       |     |     |     |
| Contact               | previous time    | incremen  | 5           | 1       | .000 | 00E-01 |        |       |            |          |            |         |              |       |     |     |     |
| Separatio             | Stotal period    | to be cov | ered        |         | .000 | 00E+00 |        |       |            |          |            |         |              |       |     |     |     |
| e oparado.            | total period     | covered s | o far       |         | .000 | 00E-01 |        |       |            |          |            |         |              |       |     |     |     |
|                       | due to number    | of recycl | es          |         |      |        |        |       |            |          |            |         |              |       |     |     |     |
| \$2.20000E-01         | 2 1 1.00E+00     | 1.70E-01  | 1.00E+00    | 1.000   | 0    | 1 0    | 1.83   | 8E-01 | 1.039E+00  | 5.02E-03 | -5.102E-02 | 395     | 3 1.         | 0 00  | 0   | 7   | 8   |
| *** SYSTEM            | INFORMATION MESS | AGE 8137  | (NL3CON)    |         |      | -      |        |       |            |          |            |         | -            |       |     |     |     |
| SEPARAT               | TON condition ha | a heen de | tected, ad  | ditions | 1 10 | eratio |        | 11 he | nerformed  |          |            |         |              |       |     |     |     |
| \$2.20000E-01         | 2 2 3.805-01     | 4.50E-03  | 1.458-01    | 0.026   | 0    | 1 0    | 9.05   | E-03  | 9.504E-01  | 4.505-03 | -4.4888-02 | 395     | 3 1.0        | 0 00  | 0   |     | 9   |
| \$2 20000E-01         | 2 3 2 518-02     | 4 035-05  | 9 595-03    | 0.026   | 0    | 1 0    | 8.00   | E-05  | 9 4448-01  | 4 448-03 | -4 4495-02 | 395     | 3 1          | 0 00  | 6   |     | 10  |
| *************         |                  |           | 2.022-03    | 0.020   | ~    |        | 0.05   |       | 2          |          |            | 323     |              |       | -   | -   | -0  |

Figure 11-1 Sample .f06 file

| LOAD<br>STEP  | NO.<br>INC IT | R DISP   | ERROR FACTORS<br>LOAD | WORK  | CONV<br>RATE | ITR<br>DIV | MAT | NO.<br>BIS | AVG<br>R_FORC | TOTL<br>WORK | AVG      | DISP -<br>MAX | AT | GRID  | c | LINE_FACT | SNO | NO.<br>QNV | TOT | TOT<br>ITR |
|---------------|---------------|----------|-----------------------|-------|--------------|------------|-----|------------|---------------|--------------|----------|---------------|----|-------|---|-----------|-----|------------|-----|------------|
| \$3.64000E-01 | 3 2           | 3.27E-01 | 9.26E-04 5.0          | 4E-02 | 0.013        | 0          | 1   | 0          | 3.38E-03      | 2.274E+00    | 5.46E-03 | -5.464E-02    |    | 397 3 | 3 | 1.00      | 0   | 0          | 11  | 12         |

- LOAD STEP: Step number minus 1 plus fraction of step, i.e. 0.08 = 8% of first step
- NO. INC: Increment number
- ITR: Iteration number within the load increment
- Error Factors: This must be smaller than the tolerances EPSU, EPSP and EPSW before the solution will move ahead.
  - DISP: Displacement errors
  - LOAD: load errors
  - WORK: work erroes
- CONV RATE: Should be between 0 and 1, bigger than 1 means, the solution will never converge
- ITR DIV: Divergence counter, MAXDIV triggers the divergence process
- MAT DIV: Divergence counter for element and material routines
- NO. BIS: Number of bisections
- AVG R\_FORCE: Average residual force (forces and moments). Should be small
- TOTL WORK: Approximate total work
- DISP AVG MAX AT GRID C: Average and maximum displacements at grid in direction c
- LINE S: FACT NO: Line search factor a and number of line searches (not used for PFNT)
- NO. QNV: Number of Quasi Newton Vectors (not used for PFNT)
- TOT KUD/ITR: Total number of stiffness updates / iterations



## **Request for More Diagnostics**

To look for more clues on why a job did not converge, you can request for more diagnostics using the NLOPRM case control command.

| NLOPRM  | Nonlinear Regular and Debug Output Control Parameters -<br>SOL 400   |
|---|--|
| Controls MSC N<br>constraints of Mi   | astran nonlinear solution output, debug printout, debug POST and punch-out of contact PC and MPCY Bulk Data entries. |
| Format:   |  |
| NLOPRM = [OUT   | CTRL = {STD,SOLUTION,INTERM}]  |
| $\begin{bmatrix} NLDBG = \begin{cases} 2\\ 2 \end{bmatrix}$ $\begin{bmatrix} DBGPOST = \end{cases}$ | NONE<br>NIBASIC NRDBG, ADVDBG, NJDBAS<br>NJDMED<br>SNONE<br>LTIME<br>LSTEP<br>LSUBC<br>ALL                           |
| DELIMIT=  | ]]   |
| GRIDINF=  | No KGRID }   |
| Example(s):   |  |
| NLOPRM<br>NLOPRM  | OUTCTRL=STD, SOLUTION DBGPOST=LTIME<br>OUTCTRL=(SOLUTION, INTERM), MPCPCH=(OTIME, STEP)                              |

#### **Basic Nonlinear Diagnostics**

Advanced nonlinear results can be obtained with:

nloprm nldbg=NLBASIC

| Basic Nonlinea<br>Diagnostics  | r     |          |                       |                      |                               |                            |             |    |
|--|-------|----------|-----------------------|----------------------|-------------------------------|----------------------------|-------------|----|
| \$3.64000E-01 3 1 1.00E+00 7.34E-02 100E+00 1.000<br>1 MSC NASTRAN JOB CREATED ON 06-AUG-14 AT 08:55:31<br>DEFAULT SCI | 0 1 0 | 1.03E-01 | 2.338E+00<br>FEBRUARY | 5.70E-03<br>26, 2015 | -5.799E-02<br>MSC Nastran 11/ | 395 3 1.00 0<br>27/13 PAGE | 0 10<br>109 | 11 |
| 0  |       |          |                       |                      | SUBCASE 1                     | STEP 1                     |             |    |
| 0 NON-LIXEAR ITERA   | TION  | MODUL    | E SOL                 | UTION                | DATA                          |                            |             |    |
| LOOP CONTROLS :<br>SUBCASE 1 STEP 1<br>NAXBIS  |       |          |                       |                      |                               |                            |             |    |



#### **Advanced Nonlinear Diagnostics**

Advanced nonlinear results can be obtained with:

nloprm nldbg=ADVDBG

This is useful for tracking down where the worst residuals are located.

| Advanced Nonlinear<br>Diagnostics   |  |
|---|--|
| maximum residual force at node 297 du<br>maximum reaction force at node 481 du<br>residual convergence ratio 2.214E-04  | egree of freedom 2 is equal to 1.107E-01<br>egree of freedom 1 is equal to 5.000E+02                         |
| maximum residual moment at node 297 d<br>maximum reaction moment at node 482 d<br>residual convergence ratio 1.918E-02  | egree of freedom 4 is equal to 3.025E-03<br>egree of freedom 5 is equal to 1.577E-01                         |
| maximum displacement change at node<br>maximum displacement increment at node<br>displacement convergence ratio 9.647E-03   | 370 degree of freedom 3 is equal to 2.399E-05<br>315 degree of freedom 1 is equal to 2.486E-03               |
| maximum rotation change at node<br>maximum rotation increment at node<br>rotation convergence ratio 2.582E-02   | 442 degree of freedom 6 is equal to 1.588E-04<br>444 degree of freedom 6 is equal to 6.148E-03               |
| strain energy change at this iteration is<br>strain energy change at this increment is<br>relative energy error is<br>\$1.00000E+00 10 3 9.65E-03 5(221E-04).55E-04 0.188 0 | 7.22710E-04<br>4.72430E+00<br>1.52977E-04<br>1 0 2.93E-04 2.385E+01 8.27E-03 -7.709E-02 397 3 1.00 0 0 35 36 |

#### **Review the Intermediate Results**

Intermediate results (= results before end of the job) can be obtained with:

NLOPRM OUTCTRL=SOLUTION, INTERM



For each converged increment an OP2 results file is written at the end of the increment. Allow to check results before the end of the job. If final OP2 is not written, intermediate results can be checked to help debuging.

## SOL 400 Analysis Messages

MSC Nastran generates a substantial amount of information concerning the problem being executed. The .f04 file provides information on the sequence of modules being executed and the time required by each of the modules; the .log file contains system messages.

MSC Nastran may terminate as a result of errors detected by the operating system or by the program. If the DIAG 44 is set, MSC Nastran will produce a dump of several key internal tables when most of these errors occur. Before the dump occurs, there may be a fatal message written to the .f06 file. The general format of this message is



\*\*\*SYSTEM FATAL ERROR 4276, subroutine-name ERROR CODE n

Some messages, like FATAL ERRORS or other text issued whenever an interrupt occurs that MSC Nastran is unable to satisfactorily process. The specific reasons for the interrupt are usually printed in the.f06 and/or.log file.

For example, in SOL 400, nonlinear analysis may be terminated by divergence. When it occurs, FATAL ERROR printout shows the following before the end of .f06 statement:

\* \* \* END OF JOB \* \* \*.

The following message can be found in the printout immediately after the diverged iteration information (starting with % in .f06).

```
NON-LINEAR INTERATION MODULE OUTPUT
*** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.
```

\*\*\* SOLUTION DIVERGES FOR SUBCASE m STEP n.

MSC Nastran has only two exit number, i.e.,

- 0: normal exit
- -1: fatal error

MSC Nastran provides many user fatal error messages (UFM). A normal/successful run exit message is as follows:

For a job that terminates abnormally you are referred to the FATAL ERROR MESSAGE in the jobname.f06 file.

One way to debug convergence issues is to use the NLOPRM bulk data entry, such as NLDBG = ADVDBG, DBGPOST options and so on, to load increment diagnostics in the .f06 file, which may help debug the model. Each iteration of each load increment generates a report in the .f06 file giving the convergence and stiffness update information.

If the job terminates before 100% of the loading is applied this convergence information can be used to identify possible reasons. The default numerical method used by SOL 400 is the full Newton method with the load increment size adjusted according to how many iterations are required to achieve convergence. The incremental load size will be adjusted up if convergence is achieved easily, and will be reduced if more than the target number of iterations is required to achieve convergence.

Convergence is achieved when the error factors on the selected criteria are below the required values (called the convergence criteria). In this convergence and stiffness update information, the following values are given:

- LOAD STEP: percentage of total load reached at this incremental load value
- NO. INC: increment number of this increment relative to all increments in this step
- ITR: number of iterations required to get convergence in this load increment

Under ----- ERROR FACTORS ---- Convergence / Error Factors that measure convergence

- DISP: displacement vector residual
- LOAD: load vector residual



- WORK: work value residual
- CONV RATE: convergence rate
- ITR DIV: number of iterations that diverged in this increment
- MAT DIV: number of iterations that diverged due to material issues in this increment
- NO. BIS: number of bisections/cutbacks in this load increment
- AVG R\_FORCE: average residual force
- TOTL WORK: integration of the forces over the displacements (e.g. total work) over the model

Under - - - - DISP - - - - Displacement Summary

- AVG: average displacement value that occurred in this load increment
- MAX: maximum displacement value that occurred in this load increment
- AT GRID: tells the grid/node number at which the maximum displacement occurred
- C: tells the degree of freedom (component) of grid/node at which the maximum displacement occurred

Under - - - - LINE\_S - - - - Line Search Parameters

- **FACT:** Line search factor (does not apply with full Newton method)
- NO: Number of line searches (does not apply with full Newton method)
- NO. QNV: Number of quasi-Newton vectors
- TOT KUD: Total number of stiffness updates in this analysis/subcase
- TOT ITR: Total number of iterations performed in this analysis/subcase

#### **Reviewing Convergence**

From a user's perspective, when performing a nonlinear simulation, the most difficult thing is to resolve convergence problems encountered in the analysis.

In MSC Nastran, convergence may refer to:

- Convergence in the iterative solver
- Convergence in eigenvalue extraction
- Convergence in aero-elasticity flutter calculation
- Convergence in optimization
- Convergence in equilibrium.

In this section, we will only be focusing on convergence equilibrium of the structural system. Figure 11-2 gives a quick summpar of steps you should undertake.

For structural analysis, a lack of convergence implies that the numerical solution has not reached equilibrium to the desired level of accuracy. Even if convergence is achieved, it is strongly recommended to determine how many iterations were required and to consider if this was a reasonable number or an excessive number.

Recall that the number of iterations has a significant influence on the computational costs, so not only is there the requirement that the solution converges, but that it also converges efficiently.



## Determining if Your Simulation has failed to Converge

If a SOL 400 fails to converge, the exit number = 1. Also the output provides a message:

- User-specified Upper Limit on Number of Iterations
- Minimum Time Step Size Exceeded/Time-Step Size Too Small
- Allowable Number of Bisections Exceeded
- Excessive Pivot Ratio





Check *Resolving Convergence Problems* Section. Look at residuals, contact changes, etc. to see what is changing from one iteration to the next. Consider linearizing materials, gluing all contacts, and/or adding extra constraints to help indentify unconstrained rigid body modes. If nothing else works, try running a modal solution to check for unconstrained rigid body modes and make sure constraints are behaving as expected. A last resort is to delete parts of the model and completely restrain the "cut" surface to isolate those parts of the model that work.

\* Getting even one converged increment is significant because it means you have adequately (although not necessarily correctly) input, loaded and constrained your model. The converged solution provides a lot of information on potential problems that might prevent the model from running to completion: contact penetrations, excessive strains or deformations, freefloating components (sometimes numerical damping will allow convergence of a component that is not properly constrained via boundary or contact conditions)

#### Figure 11-2 Flowchart to Debugging a SOL 400 Run

## **Review Plot Results from Converged Increments**

Insights can be gained from the partially completed run to find out why the job failed to converge.

- When a job fails to converge, SOL 400 will save results up to the last converged load increment
- Review displacements, stresses, load vs. deflection, etc...



Be careful with displacement scale factor



## Reviewing the MSC Analysis Manager

The **Analysis Manager** provides interfaces within Patran to submit, monitor and manage analysis jobs on local and remote networked systems. It can also operate in a stand-alone mode directly with MSC Nastran. The **Analysis Manager** automates the process of running analysis software even on remote and dissimilar platforms.

| Msc.Analysis Manager - [Analysis]   | Mgr1]   |  |          |          |
|---|---|--|----------|----------|
| Cueue Eat view Tools window   | neip  |  |          | _0_X     |
| default   | Job Control Performan   | ce Job Files   |          |          |
| Subnit     Grido Control     Grido Contro     Grido Control     Grido Control     Grido Control | Job Number:<br>Job Name:<br>Ovener:<br>Submit Host:<br>Running Host:<br>AM Host:<br>Job Status:<br>Elected Targe (r | 7<br>s4_job1<br>caserio<br>ACASERIO<br>ACASERIO<br>marc2001<br>Running | ×        | <u>*</u> |
| Administration  |   |  |          | _        |
|   |   | Abort  |          | <u> </u> |
| X (12> Filling data from F<br>∠ (13> Filling data from F<br>(14> Filling data from F<br>(15> Filling data from F<br>(15> Filling data from F<br>(16> Wait! Submitting jo<br>(17> Successful!  | ResourcesView<br>eneralView<br>liscView<br>RestartView<br>ob <s4_jobl> S</s4_jobl>                                  | 1><br>ubaitted Job #7  | Assigned |          |
| Uutput  |   |  |          |          |
| For Help, press F1  |   |  |          |          |



Files are automatically copied to where they are needed, the analysis is performed and pertinent information is relayed back to you. When the analysis is complete files are returned/deleted. Time consuming system tasks are reduced so that more time is available for productive engineering.

The Analysis Manager can also be used to monitor job progress (even non-jobs)

- Shows all job and host information
- Abort function can be used to stop running jobs

## Tips for Starting with Nonlinear Analysis

Start your model simple and gradually add complexity. It is better to start the analysis with a small load that will obtain convergence than start with a large load that requires the program to subdivide the increment. When starting with a new model, set up your model such that you get some initial results or a run-failure within a few minutes. This means that you may need to start with a coarse mesh.

Once you know your model runs to completion, you can add refinement and/or complexity. As a general comment, a 1000-2000 node job would fail within few minutes if there are set-up errors. This is what we want: if the job fails, it should fail fast. These initial few runs serve the purpose of testing the set-up parameters to make sure that they work right for this model. You can expect to make a few/several runs to determine that the parameters are correct for that class of problems.

Once these parameters are known, you can apply them to other models in that class of problems. Once the job runs to completion, you can add complexity/refinement. Now the job will take longer, but we are confident that it will run to completion.

Displacement control: In general, problems with applied displacements are numerically more *stable* than problems with applied forces. For example, if a cantilever beam loaded at the end with a force, formation of a plastic hinge can make the model go non-positive definite. With an applied displacement, this scenario is less likely.

#### **Output Messages**

#### **Additional Output**

When convergence is a problem it is necessary to obtain as much information as possible on the numerical process. The NLOPRM case control command is used to obtain additional result information.

In particular:

NLOPRM OUTCTRL=STD, INTERIM, NLDBG=NLBASIC, NRDBG, DBGPOST=LTIME

will provide the additional information to evaluate the performance. If contact bodies are present, it may be preferable to use:

```
NLOPRM OUTCTRL=STD, INTERIM, NLDBG=NLBASIC, NRDBG, DVDBG, N3DMED DBGPOST=LTIME
```



#### **Standard User Fatal Messages**

If you run the analysis and it does not run at all or ends before completing, you will get an error message in the jobname.f06 or jobname.log file that will give you an indication of what the problem is. Do a text search on the word *fatal* in the jobname.f06 file. The first thing to check is to that you were able to get a license to run the job. Licensing problems are a common reasons for a run to fail. If you are sure you have a license and submit the job correctly, you should get a jobname.sts file that will end with an exit number preceded by a description of why the run stopped.

Common exit numbers are:

| Exit 0  | - | success. The job ran to completion and did everything you asked.  |
|---|---|---|
| Exit 1  | - | syntax error in the input file. You should check the input syntax of<br>the line the error message points to, but it is likely that the actual<br>error was in the input block prior to where the message points. |
| Exit 1 with user fatal<br>EXCESSIVE PIVOT<br>RATIO in .f06                    | - | typically means no convergence due to rigid body motions or a<br>numerically ill-conditioned system. See recommendations for<br>equilibrium below.  |
| Exit 1 with other user fatal  | - | means the analysis ran into convergence problems part way through<br>and did not complete. Interpret the error message, look for model<br>problems  |
| For SOL 400 User<br>Fatal Messages (UFMs)<br>and their<br>Interpretation see: |   | http://simcompanion.mscsoftware.com/infocenter/   |

If you receive a Nastran input from another source, it is strongly recommended that you read it into either Patran or SimXpert and display the model to check for completeness. Any.sts file with nonzero total time lines means there are converged increments. Display the converged increments to see what is going on.

#### **User Fatal Message Scenarios**

- No increments/iterations successfully completed but solution has started:
  - \*\*\* USER FATAL MESSAGE 4296 (EQD4D)

ILLEGAL GEOMETRY FOR QUAD4 ELEMENT WITH ID = 97

If a UFM message similar to the above comes as soon as the job is submitted; i.e., at the first assembly of the first iteration, it indicates a meshing problem. Re-check mesh and re-mesh.

• If an increment or iteration has been successfully completed and one gets the message:

\*\*\* USER FATAL MESSAGE 4296 (EQD4D)

ILLEGAL GEOMETRY FOR QUAD4 ELEMENT WITH ID = 97



This exit message or a similar one may indicate excessive element deformation during a particular load increment in a particular iteration. The way to get around this error is to reduce step size. However, SOL 400 does that automatically and if the problem still persists, it gives this message. If the program is unable to recover after several cutbacks, it implies that the last converged solution is not really a good solution or the deformation is excessive for this mesh. The solution may be to either use Restart or rerun-the complete analysis with a tighter tolerance.

It may be necessary to change the original mesh to anticipate where the deformation will occur. Also, it should be recognized that lower-order triangular or tetrahedral elements (with PSLDN1) is the preferred solution for large strain/distortion models.

• User-specified Upper Limit on Number of Iterations: The solution may fail and give this message if SOL 400 reaches the user-specified upper limit on the number of (Newton-Raphson) iterations within a load increment. The default is set to 10. One of the first things to try, if you think the problem is otherwise well posed (e.g., no other problems are known) would be to increase the value (specified on NLSTEP) to a value of 20.

What is happening is that SOL400 keeps iterating and tries to converge to a solution for that increment. If that does not happen, SOL 400 will cut back the load (by half) and re-solve that increment. Sometimes this is not enough to get convergence, and it will exit with a UFM.

Check the residual values in the ADVDBG section of the .f06 file to confirm this.

Minimum Time Step Size Exceeded/Time-Step Size Too Small: This exit message indicates that SOL 400 cuts back to a time-step size too small for the analysis to continue. The load stepping algorithm has a cut-back feature where the load step is automatically reduced (when the time step is decreased, the factor is calculated internally based upon the minimum time step).

When an increment runs into these problems, it automatically cuts the load-step size and resolve that increment. If the problem persists, it cut-back the load-step again. This happens until the limit of the number of cut-backs is reached. This can result in a very small time step. In such a case, SOL 400 stops the analysis with a UFM.

• Allowable Number of Bisections Exceeded: SOL 400's automatic load stepping scheme is set up such that the applied load in an increment scales up (or down) depending on how easy (or difficult) the solution was in the previous increment.

The degree of difficulty is determined based on the NDESIR parameter of NLSTEP: desired number of iterations per increment (default = 4). SOL 400 will scale down the step size until it reaches a lower limit on the step size (default = 0.001% of total time step) and then exit with a UFM worded similar to the above. This is an indication to you that the analysis encountered some difficulty at that stage.



#### CHAPTER 11 197 Trouble Shooting







Q1: Is SOL 400 only for nonlinear analysis, or can I also do linear analysis?

SOL 400 can do linear, nonlinear, modal, buckling, and transient structural analysis along with a host of other analysis types. SOL 400 is designed to be the only solution sequence you need. For a complete list, see ANALYSIS case control entry documentation in the *Quick Reference Guide*.

Q2: Will I get the same results from a SOL 400 linear analysis as I do from a SOL 101 analysis?

SOL 400 has additional capabilities that require controls not available in SOL 101. This means the additional controls for a SOL 400 analysis must be set in the same way as they are for a SOL 101 analysis.

Having said that, SOL 400 control defaults are set in such a way that you should get the same results from a SOL 400 analysis that you do from a SOL 101 analysis.

Q3: How different is SOL 400 from SOL 600?

In SOL 600, MSC Nastran preprocesses the data and calls the Marc solver. In SOL 400, all MSC Nastran infrastructure is available, while in SOL 600 only selected MSC Nastran infrastructure capabilities have been integrated. This is why MSC recommends that SOL 400 be the default solution method for solving nonlinear problems.

An extensive finite element library for building your simulation model, and a set of solution procedures for the nonlinear analysis, which can handle very large matrix equations, are available in both solution sequences of MSC Nastran Implicit Nonlinear.

Q4: Will I get the same results from a SOL 400 nonlinear analysis as I do from a SOL 106 static or SOL 129 transient nonlinear analysis?

There are additional capabilities and additional controls in SOL 400 that must be set in the same way as they are for SOL 106 or SOL 129 in order to get the same results. Also, SOL 400 has a complete set of large strain elements that make it unlikely that you will get the same solution unless you are performing a small deformation analysis.

However, if you set your analysis up to use the same elements as the SOL 106 or 129 solution, and you limit the conditions applied to the model you should get the same results as a SOL 106 or 129 analysis.

Q5: Will I get the same results from a SOL 400 linear, modal, buckling, or nonlinear analysis as I will from a SOL 101, 103, 105, and 106 analysis?

SOL 400 encapsulates those solution sequences and should be able to reproduce their results. Getting the same answers requires equivalence in element types, analysis control settings, and procedures.

Q6: Is it possible to use the same model for linear and nonlinear analysis. If so what is the best way to add the additional input required for the nonlinear solution to an existing linear model?

Yes. SOL 400 has been designed to allow you use the same model for linear and nonlinear analysis. To accomplish this, do the following:

- a. Take the linear model.
- b. Read it into a graphical pre-and post processor like Patran.



- c. Add the additional model attributes required such as contact.
- d. Make sure to specify the proper nonlinear analysis control parameters.
- e. Let the graphical pre- postprocessor write out the correct input.

Q7: Is it possible to convert an existing MSC Nastran model into a SOL 400 model? If so what is the best way to do this?

Yes, you can convert an existing MSC Nastran model into a SOL 400 model. To accomplish this, do the following:

- a. Take the existing MSC Nastran model.
- b. Read it into a graphical pre-and post processor like Patran or SimXpert.
- c. Add the additional model attributes required such as contact or nonlinear materials properties.
- d. Specify the proper nonlinear analysis control parameters.
- e. Let the graphical pre- postprocessor write out the correct input.

See Chapter 6: Setting Up, Monitoring, and Debugging the Analysis for an example of the process required to convert a SOL 101 model to a SOL 400 model.

Q8: Is it possible to take an existing Abaqus model and convert it into a SOL 400 model? If so what is the best way to do this? Will I get the same answer?

Starting with the 2014 release, a translator is can be used to convert an Abaqus input file into an MSC Nastran input file. Prior to that, do the following:

- a. Read the file into Patran.
- b. Change the preference to MSC Nastran.
- c. Add the additional data.

Getting the same answer depends on many factors:

- a. Compatibility of the element types selected
- b. Analysis capabilities and algorithms (such as contact algorithms)
- c. Analysis procedures
- d. Selected controls

In particular, characterization of contact interactions may affect the results.

Q9: Does Patran/SimXpert support SOL 400?

While Patran/SimXpery do not support 100% of SOL 400 capabilities, they provide complete support of the most commonly used features of SOL 400 including nonlinear materials, contact, composites, and multi stepping/perturbation analysis. MSC is continuing to develop them to support new SOL 400 capabilities.

Q10:If I am new to SOL 400 what's the best way to learn it?

There are a few ways of learning:



- The MSC documentation system has a complete set of solved SOL 400 problems in a document called the *MSC Nastran Demonstration Problems*. The input files from these solved problems are included in the documentation system.
- Patran has a set of demo problems that can be found in the **Analysis** menu, under the **Run a Demo** tab. After running these demos, Patran can be used to interrogate the model and investigate how the models are set up.
- The SimCompanion website has a knowledge base with answers to commonly asked questions, as well as links to the MSC training courses on SOL 400.

Q11:Where can I find example input files for SOL 400 demo problems?

Example problems can be found in the *MSC Nastran Demonstration Problems Manual*. These example problems also include input files. There is a test suite of example input files in the MSC Nastran test problem library. Also, the **Run a Demo** problems in Patran will leave the input files that were used to run the example.

Q12:What kind of results are supported in the op2, xdb, HDF5, and/or dball/master?

While you can request SOL 400 output in any valid MSC Nastran output format, only the MASTER/DBALL and new OP2 (PARAM, POST, 1) and HDF5 formats will contain all of the results data blocks of SOL 400.

Specifically, the nonlinear stress/strain and contact results data blocks will be missing if any other format is requested. Thus, MSC strongly recommends that you use MASTER/DBALL, new OP2, or HDF5 output formats for SOL 400 solutions.

Q13:What do I do if I run the analysis and do not get a solution? What if it does not converge?

Check the solution files (\*.f06, \*.log, \*.f04) to look for error messages. Typically there will be a message telling why the solution did not converge.

There are a couple of answers to this question depending on why the analysis did not get a solution. Reasons why an analysis does not result in a solution can be grouped into two categories.

- The first group relates to having a correctly formatted input file. These tend to be the most common, and also the easiest to fix.
- The second group relates to having what might be called a *well-posed problem*.

This means that our problem does not violate any of the laws of physics, and that there actually is a valid numerical solution to the problem we are trying to solve. A typical example of this type of error would be trying to come up with a static solution to a problem that is not statically determinate. There are methods used to determine if a model is properly constrained, such as running a modal analysis to look for unconstrained rigid body modes.

Q14:How do I know if I need to do a nonlinear analysis?

While this is not an exact answer as loading conditions will affect this, you should do a nonlinear analysis in one of the following conditions:

- If your strains are approaching 5%
- The deflection of any node in your model approaches 5% of the smallest dimension.



Q15:How do I know if linear contact will give me the right answer, or if I need to do a nonlinear analysis?

The term *linear contact* is an oxymoron. The very nature of contact problems are that finite deformation of the nodes must be tracked to determine if the notes are in contact or not in contact. This deformation is almost always large enough to invalidate linear contact solutions. The best use for linear contact is to use it to connect dissimilar meshes or full assembly modeling where no separation occurs.

Q16:What is the difference between linear buckling analysis and nonlinear buckling analysis?

A linear buckling analysis is based on the un-deformed configuration of the structure.

A nonlinear buckling analysis is based on a deformed shape of the structure. The deformation and stresses make the structure stiffer (if in tension) and cause the natural frequencies to increase. Hence, it is often necessary to do a nonlinear buckling analysis to get accurate mode shapes and frequencies.

Q17: What is a stress stiffened model analysis?

Before extracting the eigenvalues from the structural system, the model may be pre-loaded prior to the modal extraction. Initial stress effects are then included in the stiffness which tend to raise the eigenvalues. This effect tends to affect the eigenvalues more than the eigenvectors (mode shapes).

Q18:Will RBE's rotate with the rest of the model?

If a large displacement, nonlinear analysis is performed, then yes, the RBEs rotates with the model. In a linear analysis, the displacement and rotation of the nodes is assumed to be infinitesimally small.

Q19:What are Marc elements? When would I use them? How do I activate them?

Most MSC Nastran elements, also called Advanced nonlinear elements or large strain elements were originally formulated for linear analysis. At some point, some of those original elements were modified for use in nonlinear analysis (SOLs 106 and 129), but certainly not all of them.

- Instead of developing new elements and capabilities from scratch for SOL 400, MSC Nastran decided to copy the well-proven, robust, mature large deformation/large strain analysis procedures and element formulations from Marc.
- Instead of developing new nonlinear element designations, which would have increased work to convert a linear SOL 101 model to a SOL 400 nonlinear model, MSC Nastran decided to use the same element designations and allow you to control the formulation used through secondary property entries such as PBEMN1, PSHLN1, PSHNL2, and PSLDN1.

While this method gives you direct control over the element formulation, SOL 400 automatically selects the proper formulation for the problem. This means that if the problem is a large deformation, large strain problem, SOL 400 automatically uses the large strain element formulation.

Unless you want to directly control the formulation (such as use reduced integration or some other special formulation), it is best to allow MSC Nastran to select the formulation used. This also means that no additional input is required.

Q20:Do shell and beam offsets rotate with the model?

Yes, if MDLPRM, OFFDEF, LROFF is set in the deck when parameter LGDISP>=0.

Q21:Will CBAR elements rotate with the model?



Yes, if CBAR is converted to a CBEAM.

Q22:Will CGAP elements rotate with the model?

No. Contact bodies should always be used for contact involving large deformations.

Q23:Will CBEAM pin flags rotate with the model?

No.

Q24:What is the difference between the Stress Tensor and the Nonlinear Stress Tensor quantities I see listed in my Patran results?

There is no difference. The label *stress tensor* was used for linear analysis, and the label *nonlinear stress tensor* for nonlinear analysis.

Q25:Which stress quantity does MSC Nastran use when it looks up the stress/strain curve I provided?

For small strain elements (traditional MSC Nastran elements), engineering stress/strain is used, but for large strain nonlinear element (new Marc elements), Cauchy stress is used.

Q26:Will the force I apply rotate with the model?

These are referred to as *follower forces* and they will be included if: 1) a nonlinear analysis procedure is used; 2) a FORCE1 or FORCE2 entry is used for loading; and 3) and appropriate PARAM,LGDISP value is used. Only *follower forces* applied by FORCE1, FORCE2, MOMENT1, MOMENT2 will rotate with model. Forces applied by RFORCE will rotate according to the specified angular velocity/acceleration rules.

Q27:What if I don't want the force to rotate with the model?

Use the static force FORCE bulk data entry.

Q28:Will the pressure I apply to a surface rotate with the model?

Only follower forces applied by PLOAD, PLOAD2, and PLOAD4 will rotate with model.

For details, see *follower forces* notes under the PLOAD and PLOAD4 entries in the *MSC Nastran Quick Reference Guide*.

Q29:Is it possible to create the input such that the pressure I apply to a surface will not rotate with the model?

Yes. With PLOAD4, if the load direction is given, the pressure applied will be fixed in the given direction and not rotate with model. See remark 2 of PLOAD4 in the *MSC Nastran Quick Reference Guide*.

It says: 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. 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.

Q30:If I am familiar with running linear MSC Nastran analyses but have never run a nonlinear SOL 400 analysis, how can I learn what I need to do differently?



It is 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. Copy an example problem to your local directory, so you can see the files created, and you don't inadvertently create files in the delivery directory.

Q31:How can I see a plot of load vs deflection for the rigid body that loaded my model?

Only available for load controlled rigid bodies, you can plot the results of the control grid of the rigid body.

Q32:How can I monitor the nonlinear solution to see if it is progressing toward convergence?

Monitor the jobname.sts file with a text editor that automatically updates. As each load increment converges a new line with relevant information will appear in the .sts file.

For details, see Chapter 6: Setting Up, Monitoring, and Debugging the Analysis.

Q33:Is it possible to see the results of any of the intermediate loading steps before the analysis is complete?

Yes. It is possible to have MSC Nastran write out the results of converged increments to intermediate OP2 files. See the INTERM option of the OUTCTRL parameter under the NLOPRM entry in the MSC Nastran Quick Reference Guide.

Q34:How can I find out what the normal and frictional forces or stresses are between the contact bodies?

Use the BOUTPUT case control output request and these values will be in the MASTER/DBALL or new OP2 files along with the other output requested.

Q35:Is an RBE spider the best way to fix a surface of my structure?

It depends on the constraints on the surface. If all the grids on the surface have the same behavior in some direction, an RBE is easy to use. It is also easy to *glue* the area to a rigid surface and use the rigid surface controls. You may also use SPC/SPC1.

Q36:What are alternatives to an RBE spider for displacement control of part of my structure?

SPC/SPC1/SPCD/SPCR may be combined together for any complicated displacement control. *Glueing* the area to a rigid surface and using the rigid surface controls is also easy.

Q37:Will I get the same answer if I apply my loads and constraints using a glued-on rigid surface as I would using an RBE spider or displacement constraint?

Yes, if both loading methods are done correctly you will get the same results.

Q38:How do a simulate a complex sequence of loading, such as the process of an engine head going through the sequence of bolt tightening, then heating up?



Break the loading up into discrete loadcases. For more information on this process, see Chapter 3: Nonlinearity and Analysis Types in this manual for more information on this topic.

Q40:What is the simplest way to include contact in my model?

If you create contact bodies and run a SOL 400 nonlinear analysis, the contact interaction is included by default, but does not include friction between the components. Patran has a tool under **Tools** - **Modeling** that automatically creates contact bodies based on groups, materials, properties, etc.)

To include friction, specify the friction model and friction coefficients. You can specify a global model value, a value for each contact body, or for each contact pair—the most localized value, in the most general to most specific order just provided, will be used.

Q41:What Thermal Solutions are supported in SOL 400?

SOL 400 supports steady state and transient thermal analysis procedures (ANALYSIS = HSTAT and HTRAN), updated to include thermal contact and coupled thermal-structural interaction, plus the Sinda RC network approach. Chapter 3: Nonlinearity and Analysis Types has a description of the SOL 400 thermal analysis capabilities, plus a description of the coupled multi-physics capabilities that are supported.

Q42:How do the RCNS and RCNT RC Network analysis options in SOL 400 work?

These RC analysis options run a subset of the Sinda Solver. Standalone Sinda is a modern equivalent of traditional Resistor-Capacitor type solver. It is also called *Finite Difference Lumped Parameter Network Solver*. Simulation can include various aspects of conduction, convection, radiation and with optional Fortran access predefined or custom loading function can be specified.

Using Patran with Sinda preference, or in this case from MSC Nastran, an RC network is generated from traditional shaped elements. MSC Nastran implementation does not include the Fortran access nor standard/custom loading function. Loading must be manually defined but the same effect can be achieved. Additionally, MSC Nastran implementation does include automated radiation refinement techniques available in Sinda.

