

# MSC.Nastran 2004

Release Guide

**Corporate**

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

**Europe**

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

**Asia Pacific**

MSC.Software Japan Ltd.  
Entsuji-Gadelius Building  
2-39, Akasaka 5-chome  
Minato-ku, Tokyo 107-0052, Japan  
Telephone: (81) (3) 3505 0266  
Fax: (81) (3) 3505 0914

**Worldwide Web**

[www.mscsoftware.com](http://www.mscsoftware.com)

**Disclaimer**

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

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

User Documentation: Copyright © 2003 MSC.Software Corporation. Printed in U.S.A. All Rights Reserved.

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

MSC, the MSC.Software logo, Simulating Reality, and Patran are registered trademarks of the MSC.Software Corporation in the United States and/or other countries. MSC., MSC.Dytran, MSC.Marc, MSC.Laminate Modeler, MSC.Nastran, MSC.visualNastran, and MSC.Patran are trademarks of MSC.Software Corporation.

NASTRAN is a registered trademark of NASA. PAMCRASH is a trademark or registered trademark of ESI Group. SAMCEF is a trademark or registered trademark of Samtech SA. LS-DYNA is a trademark or registered trademark of Livermore Software Technology Corporation. ANSYS is a registered trademark of SAS IP, Inc., a wholly owned subsidiary of ANSYS Inc. ABAQUS is a registered trademark of ABAQUS Inc. All other brand names, product names or trademarks belong to their respective owners.

NA\*V2004\*Z\*Z\*Z\*DC-REL

# C O N T E N T S

## MSC.Nastran 2004 Release Guide

### Preface

- List of MSC.Nastran Books, xx
- Technical Support, xxi
- Internet Resources, xxiv
- Permission to Copy and Distribute MSC Documentation, xxv

## 1

### Introduction

- MSC.Nastran 2004 Overview, 2

## 2

### Design Optimization

- Eigenvector Sensitivity and Optimization, 14
  - Introduction, 14
  - Benefits, 14
  - Theory, 14
  - Inputs, 18
  - Outputs, 20
  - Guidelines and Limitations, 20
  - Example, 20
- Support of Adjoint Sensitivity in Superelement Analysis, 25
  - Input, 26
  - Examples, 26
  - Results Discussion, 27
- Adjoint Methods as Part of the Analysis, 28
  - Introduction, 28
  - Benefits, 28
  - Inputs, 28
  - Guidelines, 29
  - Example, 29
- Distributed Memory Parallel Processing, 31
  - Introduction, 31
  - Internal Parallel for FRRD1 Module, 31
  - Input, 31
  - Output, 32
  - Example, 32

- ❑ Frequency Domain Parallel Lanczos Method, 32
  - ❑ Input, 33
  - ❑ Output, 33
  - ❑ Example, 34
  - ❑ DOF Domain Decomposition Method, 34
  - ❑ Input, 34
  - ❑ Output, 35
  - ❑ Examples, 36
  - ❑ Guidelines, 36
- New DRESP1 Response Types, 37
  - ❑ Introduction, 37
  - ❑ Benefits, 37
  - ❑ Theory, 37
  - ❑ Input, 38
  - ❑ Output, 39
  - ❑ Guidelines and Limitations, 40
  - ❑ Examples, 40
- Performance Enhancements for Large Scale NVH Optimization, 46
  - ❑ Introduction, 46
  - ❑ Benefits, 46
  - ❑ Inputs, 47
  - ❑ Outputs, 47
  - ❑ Guidelines and Limitations, 48
  - ❑ Example, 48
- Additional Control Print for Design Response and Constraint, 51
  - ❑ Introduction, 51
  - ❑ Input, 51
  - ❑ Guidelines and Limitations, 51
  - ❑ Example, 53
- Support of External Response in SOL 200, 55
  - ❑ Introduction, 55
  - ❑ Benefits, 55
  - ❑ Inputs, 56
  - ❑ Outputs, 62
  - ❑ Example, 64
- Improved BEAM Processing, 74
- Frequency-Dependent Constraints, 77
  - ❑ Introduction, 77
  - ❑ Benefits, 77
  - ❑ Input, 77
  - ❑ Outputs, 77
  - ❑ Guidelines and Limitations, 77
  - ❑ Example, 77

# 3

## Dynamics

- Results/Discussion, 78
- Generation of a Complete Updated Bulk Data File in SOL 200 File, 79
  - Introduction, 79
  - Example - newbulk2.dat, 80
- Responses Spanning Across Subcases or Superelements, 83
  - Introduction, 83
  - Benefits, 83
  - Input, 83
  - Example - rspan.dat, 83
- Miscellaneous Optimization Enhancements, 86
  - Multiple Frequency Sets in a Frequency Response Analysis, 86
  - Correction in the Use of OFREQ, 86
  - Enhanced Optimization Algorithm, 87
  - Elimination of Constraints on Properties that are a Linear Function of a Single Design Variable, 88
  - New DESVAR Case Control command to Select a Subset of Design Variables, 89
  - Old Design Sensitivity and Optimization, 90
- Enhanced Methods for Specifying Delay and Phase Angle in Dynamics, 92
- Select Modes to be Included for Response Analysis, 93
  - Introduction, 93
  - Benefits, 93
  - Inputs, 93
  - Examples, 94
  - Example 2, 94
  - Example 3, 94
  - Example 4, 94
  - Example 5, 95
- Linear Transient Analysis Restarts (SOLs 109 and 112), 96
  - Introduction, 96
  - Inputs, 96
  - Guidelines and Limitations, 96
- Enhancements to Initial Condition Usage in Linear Transient Analysis, 97
  - Introduction, 97
  - Input, 98
- Residual Vector Enhancements, 100

- ❑ Calculation of Residual Vectors in MSC.Nastran, 100
  - ❑ Enhancements to Degrees of Freedom Specification for Residual Vector Processing, 103
- Enforced Motion Enhancements, 104
  - ❑ Introduction, 104
  - ❑ Benefits, 104
- Random Response Analysis, 106
  - ❑ Introduction, 106
  - ❑ Cross-Power Spectral Density and Cross-Correlation Functions, 106
  - ❑ Numerical Integration Using Log-Log Approximation, 110
  - ❑ Theory, 110
  - ❑ Inputs, 112
  - ❑ Cumulative Root Mean Square (CRMS), 112
  - ❑ Theory, 112
  - ❑ Inputs, 112
  - ❑ Output, 112
- Modal Kinetic and Strain Energy Output, 114
  - ❑ Introduction, 114
  - ❑ Benefits, 114
  - ❑ Theory, 114
  - ❑ Inputs, 114
  - ❑ Outputs, 116
  - ❑ Guidelines and Limitations, 116
  - ❑ Example, 117
- Enhancements to Structural Damping for Shell and Bush Elements, 118
  - ❑ Introduction, 118
  - ❑ Benefits, 118
  - ❑ Theory, 118
  - ❑ Inputs, 119
  - ❑ Outputs, 120
  - ❑ Guidelines and Limitations, 120
  - ❑ Example, 120
- Modal Contribution Fraction (MCFRACTION), 122
  - ❑ Introduction, 122
  - ❑ Benefits, 122
  - ❑ Theory, 122
  - ❑ Inputs, 124
  - ❑ Outputs, 124
  - ❑ Guidelines and Limitations, 126
  - ❑ Example, 127
- Enhancements to Modal Dynamic Analysis Processing, 132

- ❑ Increased Accuracy from GKAM Module, 132
  - ❑ Increased Accuracy from FRRD1 Module, 132
  - ❑ Eigenvector Matrix Output from GKAM Module, 132
  - ❑ New Eigenvalue Table Output from GKAM Module, 133
  - ❑ New Partitioning Vector Output from GKAM Module, 133
- MAX/MIN/RMS Methodology in Data Recovery, 134
  - ❑ Introduction, 134
  - ❑ Benefits, 134
  - ❑ Theory, 135
  - ❑ Inputs, 135
  - ❑ Case Control, 135
  - ❑ Outputs, 137
  - ❑ Guidelines and Limitations, 139
  - ❑ Examples, 140
- Massless Mechanism Identification and Control Enhancements, 145
  - ❑ Input, 146
  - ❑ New Parameters for MSC.Nastran 2004, 147
- General Enhancements for Orthogonalizing Complex Vectors, 149
  - ❑ Summary, 149
  - ❑ SUBDMAP CALL, 150
  - ❑ Results of Testing, 153
  - ❑ Limitations, 154
- Complex Eigenvalue Analysis Updates, 155
- Real Eigenvalue Analysis, 156
  - ❑ Reduction Method Performance, 156
  - ❑ Fatal/Warning Message Option, 156
  - ❑ New Eigenvalue Matrix Output From the Real Eigenvalue Analysis (READ) Module, 156
- Fast Direct Frequency Response Option, 157
  - ❑ Theory, 157
  - ❑ Inputs, 157
- Inertia Relief with Auto-Suport, 159
  - ❑ Enhancement, 159
  - ❑ Review, 159
  - ❑ General Formulation, 159
  - ❑ Inrel = -2, 160
  - ❑ Inrel = -1, 162
  - ❑ Internal Loads and Accuracy, 163
  - ❑ Guidelines for Inertia Relief, 164
- New C-Set and R-Set Reduction, 166

# 4

## Rotordynamics

- Introduction, 172
  - Benefits, 172
  - Basic Theory and Methods, 173
  - Basic Equations, 173
  - Whirl Modes and Critical Speeds, 174
  - Multiple Rotors and the Reference Rotor, 175
  - Input, 176
  - Case Control, 176
  - Parameters, 183
  - New Case Control Options, 183
- Examples, 186
  - Example 1a: Rotor Whirl Frequencies, 188
  - Example 1b: Rotor Critical Speeds, 188
  - Example 2: Steady-State Rotor Response due to Unbalance Forces, 189
  - Example 3: Response to Rotor Imbalance During Start-up, 190
- Equations Used in Analyses, 192
  - Frequency Response, 192
  - Complex Modes, 193
  - Nonlinear Transient Response, 194

# 5

## Elements, Loads, and Model Checkout Tools

- CBAR/CBEAM Elements, 202
  - Introduction, 202
  - Benefits, 202
  - Theory, 202
  - Inputs, 203
  - Outputs, 204
  - Guidelines and Limitations, 205
  - Example, 205
- Nonstructural Mass Enhancements, 207
  - Introduction, 207
  - Benefits, 207
  - Theory, 207
  - Inputs, 208
  - Outputs, 208
  - Guidelines and Limitations, 208
  - Example, 209
- Enhanced Element Summary Printout (ELSUM), 213
  - Introduction, 213



- Benefits, 213
  - Theory, 213
  - Inputs, 214
  - Outputs, 214
  - Guidelines and Limitations, 217
  - Example, 217
- Equilibrium Checks, 218
  - Introduction, 218
  - Benefits, 218
  - Theory, 218
  - Inputs, 219
  - Outputs, 219
  - Guidelines and Limitations, 220
- New Shell Elements Corner Thickness Option, 221
  - Shell Elements Corner Thickness Specification, 221
  - Introduction, 221
  - Benefits, 221
  - Theory, 221
  - Inputs, 221
  - Outputs, 221
  - Guidelines and Limitations, 221
  - Example, 222
- Strength Ratio Output for Laminated Composites, 225
  - Introduction, 225
  - Theory, 225
  - User Interface, 230
- Additional Laminate Options for the PCOMP Entry, 231
  - Introduction, 231
  - Benefits, 231
  - Theory, 231
  - Input, 233
  - Remarks, 234
  - Output, 234
  - Guidelines and Limitations, 235
  - Examples (PSTIFF1, PSOFT1, PMEM1, PSMEAR, and PSMCORE), 235
  - Input File, 235
  - Results, 236
- Temperature-Dependent Composites, 237
  - Introduction, 237
  - Benefits, 237
  - Theory, 237
  - Input, 239

# 6

## External Superelements

# 7

## Coupled Fluid- Structure Analysis

- Output, 240
  - Guidelines and Limitations, 240
  - Examples, 240
- Rigid Element Enhancements, 243
  - Introduction, 243
  - Methods for Rigid Elements, 243
  - Theory, 245
  - Input, 253
  - Limitations in SOL 400, 257
  - Examples, 257
- QUADR and TRIAR Elements, 263
  - Introduction, 263
  - Benefits, 263
  - Theory, 264
  - Shell Models and Shell Normal, 267
  - Mass Properties, 268
  - Consistent Surface and Edge Loads, 269
  - User Interfaces, 270
  - Examples, 273
- CWELD Element Enhancements, 279
  - Introduction, 279
  - Benefits, 279
  - Theory, 281
- New K6ROT Default, 292
  - Introduction, 292
  - Benefits, 292
  - Theory, 292
  - Guidelines and Recommendations, 293
  - Examples, 293
- Enhancements to External Superelements, 296
- New Body In White Method, 300
  - Introduction, 300
  - Example, 301
  - Example, 301
  - New BW Coupling Algorithm, 301

# 8

## Nonlinear Analysis

- Example (acoustic1.dat), 303
- Direct Input of Interface [A] Matrix, 313
  - Examples, 313
- Acoustic Modal Participation Factors, 314
  - Introduction, 314
  - Case Control Command Summary, 315
  - Modal Participation Theory, 319
  - Modal Participation Example (acoustic2.dat), 321
- Acoustic XY Plots, Random, and Restarts, 332
  - XY Plots, 332
  - Random Analysis, 333
  - Random, XY Plot, Restart Example, 333
- Acoustic Source Change, 340
  - Simple Acoustic Source (ACSRCE) Equation, 340
- Efficiency Enhancement in Virtual Mass, 342
  - Introduction, 342
  - Input, 342
  - Theory and Output, 342
  - Upward Compatibility Issues, 342
- Modifications to the Computation of Fluid and Structural Modes in SOL 103, 343
- MSC.Nastran Implicit Nonlinear -- SOL 600, 346
  - Introduction, 346
  - What is SOL 600 and How Does it Work, 347
  - Example: Two Pipes Crushing (Contact & Large Strain):, 356
- Prerelease of the General Nonlinear Solution Sequence SOL 400, 362
  - Introduction, 362
  - Benefits, 362
  - Limitations for the Current Release, 363
  - Analysis Steps, 363
  - Linear Static Analysis and Nonlinear Static Analysis, 365
  - Vector Operations and Convergence Criteria, 365
  - Solution Algorithm and Simulation of SOL 106, 366
  - Lagrange Rigid Elements and the Right Rotation Method, 367
  - Nonlinear Iteration Summary Table, 367
  - Restart, 369
  - Output, 371
  - User's Interfaces, 372

# 9

## Numerical Enhancements

- Examples, 373
- Performance Improvements in SOL 129, 378
  - Introduction, 378
  - Benefits, 378
  - Theory, 379
  - Load-Dependent Time Step Adjustment, 380
  - Adaptive Time Stepping for NOLIN and NLRGAP, 381
  - Initial Time Step Adjustment, 383
  - Enhancements For Automatic Time Stepping Due to Error Corrections, 385
- Buckling Analysis, 387
  - Introduction, 387
  - Linear Buckling Analysis with Preloads, 387
  - Theory, 387
  - Example - prld\_buck.dat, 388
  - Nonlinear Buckling Analysis with Symmetric Stiffness Matrix, 390
  - Nonlinear Buckling Analysis with Unsymmetric Stiffness Matrix, 391
  - Example, 392
  - An Example of Cylinder Buckling, 394
- Introduction to Parallel Numeric Enhancements, 400
- Domain Decomposition, 403
  - Introduction, 403
- New Distributed Memory Parallel Methods, 404
  - Introduction, 404
  - Hierarchic Distributed Parallel Lanczos Method, 404
  - DOF Based Domain Decomposition, 405
- Inputs, 407
- Method Selection Guidelines, 411
  - Static Analysis, 411
  - Normal Modes Analysis, 411
  - Modal Dynamic Analysis (SOLs 111 and 112), 411
  - Direct Frequency Response (SOL 108), 412
- Limitations, 413
- Examples, 415
  - Example 1 – SOL 101, 415

# 10

## Heat Transfer

- ❑ Example 2 – SOL 103 with Frequency Domain Decomposition, 416
  - ❑ Example 3 – SOL 103 with Geometric Domain Decomposition, 416
  - ❑ Example 4 – SOL 103 with the Hierarchical Method, 417
  - ❑ Example 5 – SOL 111 with ACMS, 417
  - ❑ Example 6 – SOL 108, 418
- Parallel Performance Data, 420
- Other Numeric Enhancements, 421
  - ❑ Sparse Cholesky Solution, 421
  - ❑ New MPYAD Method 1 Storage 3 Option, 421
- Advanced Sorting Routines for VIEW Factor Shadowing Algorithm, 424
  - ❑ Introduction, 424
  - ❑ Benefits, 424
- Axisymmetric View Factors for Gray-Diffuse Surface Character, 428
  - ❑ Introduction, 428
  - ❑ Benefits, 428
  - ❑ Theory, 428
  - ❑ Example, 429
  - ❑ Results/Description, 430
- Thermostat Control with Deadband Applied to a Heat Source, 435
  - ❑ Introduction, 435
  - ❑ Examples, 436
  - ❑ Input File, 437
  - ❑ Example 2 (contrlt2.dat), 440
  - ❑ Boundary conditions:, 440
  - ❑ Model Description, 441
  - ❑ Results/Discussion, 442

# 11

## MSC.Nastran/MSC. Adams Interface

- ADAMS/Flex Support, 446
  - ❑ Introduction, 446
  - ❑ Benefits, 446
  - ❑ Theory, 447
  - ❑ Mass Invariant Calculations, 448
  - ❑ Inputs, 451
  - ❑ Output, 454

# 12

## Aeroelasticity

- Guidelines and Limitations, 455
- Examples, 458
- , 490

# 13

## Miscellaneous Enhancements

- Correction of a Machine Dependency in the ZONA51 Supersonic Aerodynamic Method for Planar and Nonplanar Configurations, 492
  - Introduction, 492
  - Benefits/Improvements, 493
- Enhancements to MATMOD Module Option 1, 498
  - Introduction, 498
  - Input, 498
  - Examples, 499
- Enhancements to MATMOD Module Option 2, 501
  - Introduction, 501
  - Input, 501
- New PARAML Module Options For Modifying Table Data Blocks Via DMAP, 503
  - Introduction, 503
  - Benefits, 503
  - Input, 503
  - Category 1 - Replace Any Entry in Any Record of a Table, 504
  - Option TABREPI, 505
  - Option TABREPC, 505
  - Option TABREP2C, 505
  - Category 2 - Insert an Entry in Any Record of a Table or Insert a New Record After Any Record of a Table, 506
  - Option TABINSR, 506
  - Option TABINSI, 506
  - Option TABINSC, 507
  - Option TABINS2C, 507
  - Category 3 - Delete Any Number of Entries in Any Record of a Table, Delete the Trailing Portion of Any Record of a Table or Delete an Entire Record of a Table, 508
  - Option TABDEL, 508
- Option to Reduce Post Processing Data (POST Case Control Command), 509
  - Introduction, 509
  - Benefits and Features, 509

- Inputs, 511
  - Output, 512
  - Guidelines and Limitations, 512
  - Example, 513
  - Model Description, 514
  - Input File, 514
- Additional Describers for the ASSIGN Statement, 517
  - Introduction, 517
  - Input, 517
  - Example, 518
- Option to Print the CD Value in the F06 File, 519
  - Introduction, 519
  - Inputs, 519
  - Outputs, 519
  - Guidelines and Limitations, 519
- Enhancements in the Specification of Parameters, 521
- Enhancement for the DMAP String-based Editor, 524
  - Introduction, 524
  - Benefits, 524
  - Limitations, 524
  - Examples, 524
- Consistent Parameter Checking, 526
- Punch Output Enhancement, 527
  - Introduction, 527
  - Inputs, 528
  - Example, 532
- Scale Factor for X2GG and X2PP Matrices, 552
- Direct Input of Structural Element Damping and Area Matrices, 553
- MSC.Access, DDLADD Revisions, and XDB Defects Correction, 554
- Enhanced Free Field Format, 555
  - Example 1 - Entry with or without user continuation mnemonics., 555
  - Example 2 - Entry with Automatic Continuation., 555
  - Example 3 - Entry with continuation mnemonics are included with the data., 556
  - Example 4 - Entry with single field continuations., 556
  - Example 5 - Entry with mixed single field, double field continuations., 556
  - Example 6 - Entry continued by terminating the parent entry with a comma., 557
- Dynamic Element Forces, 559

# 14

## Upward Compatibility

- Sparse Data Recovery, 561
  - User Interface, 561
  - Changes in Output Due to Sparse Data Recovery, 561
  - Limitations:, 562
  - DMAP Alters:, 563
- Results or Output Changes, 566
- Modifications to OUTPUT2 (INPUTT2) Transport Formats, 570
- Summary of DMAP Module Changes from MSC.Nastran 2001 to MSC.Nastran 2004, 574
  - DMAP Module Changes, 575
  - ASDR Module, 576
  - ASG Module, 577
  - DDRMM Module, 578
  - DCMP Module, 579
  - DPD Module, 580
  - DSADJ Module, 581
  - DSAL Module, 582
  - DSVG1 Module, 583
  - EFFMASS Module, 584
  - INPUTT2 Module, 585
  - MODACC Module, 586
  - MODEPOUT Module, 587
  - MTRXIN Module, 588
  - OFP Module, 590
  - OUTPRT Module, 591
  - ORTHOG Module, 592
  - OUTPUT2 Module, 593
  - SDR2 Module, 594
  - XYTRAN Module, 596
- Summary of Data Block Changes from MSC.Nastran 2001 to MSC.Nastran 2004, 597
- Removal of Old Features, 618
- MSC.Nastran Error List, 619



# A

## Statements, Commands, Entries, and Parameters

- New and Modified Executive Control Statements for MSC.Nastran 2004, 622
  - New Executive Control Statements for MSC.Nastran 2004, 622
  - New Executive Control Statement for MSC.Nastran Implicit Nonlinear (SOL 600) in MSC.Nastran 2004, 622
  - Executive Control Statements for MSC.Nastran 2004 with Major Modifications, 622
  - Executive Control Statements for MSC.Nastran 2004 with Minor Modifications, 622
- New and Modified Case Control Commands for MSC.Nastran 2004, 623
  - New Case Control Commands for MSC.Nastran 2004, 623
  - Case Control Commands for MSC.Nastran 2004 with Major Modifications, 623
  - Case Control Commands for MSC.Nastran 2004 with Minor Modifications, 624
- New and Modified Parameters for MSC.Nastran 2004, 625
  - New Parameters for MSC.Nastran 2004, 625
  - Parameters for MSC.Nastran 2004 with Major Modifications, 626
- New and Modified Bulk Data Entries for MSC.Nastran 2004, 627
  - New Bulk Data Entries for MSC.Nastran 2004, 627
  - Bulk Data Entries for MSC.Nastran 2004 with Major Modifications, 628
  - Bulk Data Entries for MSC.Nastran 2004 with Minor Modifications, 628

# B

## References

## INDEX

- References, 630

MSC.Nastran Release Guide , 633



---

# Preface

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

## List of MSC.Nastran Books

Below is a list of some of the MSC.Nastran documents. You may order any of these documents from the MSC.Software BooksMart site at [www.engineering-e.com](http://www.engineering-e.com).

### Installation and Release Guides

- ☐ Installation and Operations Guide
- ☐ Release Guide

### Reference Books

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

### User's Guides

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

## Technical Support

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

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

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

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

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

### Phone and Fax

#### **United States**

Telephone: (800) 732-7284

Fax: (714) 784-4343

#### **Frimley, Camberley**

#### **Surrey, United Kingdom**

Telephone: (44) (1276) 67 10 00

Fax: (44) (1276) 69 11 11

#### **Munich, Germany**

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

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

#### **Tokyo, Japan**

Telephone: (81) (3) 3505 02 66

Fax: (81) (3) 3505 09 14

#### **Rome, Italy**

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

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

#### **Paris, France**

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

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

#### **Moscow, Russia**

Telephone: (7) (095) 236 6177

Fax: (7) (095) 236 9762

#### **Gouda, The Netherlands**

Telephone: (31) (18) 2543700

Fax: (31) (18) 2543707

#### **Madrid, Spain**

Telephone: (34) (91) 5560919

Fax: (34) (91) 5567280

### Email

Send a detailed description of the problem to the email address below that corresponds to the product you are using. You should receive an acknowledgement

that your message was received, followed by an email from one of our Technical Support Engineers.

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

## Training

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

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

The MSC Institute of Technology is located at:

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

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

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

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

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

## Internet Resources

### **MSC.Software ([www.mscsoftware.com](http://www.mscsoftware.com))**

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

### **Simulation Center ([simulate.engineering-e.com](http://simulate.engineering-e.com))**

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

### **Engineering-e.com ([www.engineering-e.com](http://www.engineering-e.com))**

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

### **CATIASOURCE ([plm.mscsoftware.com](http://plm.mscsoftware.com))**

Your SOURCE for Total Product Lifecycle Management Solutions.



## Permission to Copy and Distribute MSC Documentation

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

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

---

**Please complete and mail to MSC for approval:**

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

Name: \_\_\_\_\_

Title: \_\_\_\_\_

Company: \_\_\_\_\_

Address: \_\_\_\_\_

\_\_\_\_\_

Telephone: \_\_\_\_\_ Email: \_\_\_\_\_

Signature: \_\_\_\_\_ Date: \_\_\_\_\_

---

Please do not write below this line.

**APPROVED: MSC.Software Corporation**

Name: \_\_\_\_\_

Title: \_\_\_\_\_

Signature: \_\_\_\_\_ Date: \_\_\_\_\_

Fold here

Place  
Stamp  
Here

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

Fold here

CHAPTER

**1**

# Introduction

■ MSC.Nastran 2004 Overview

## 1.1 MSC.Nastran 2004 Overview

MSC.Nastran 2004 is the most significant and comprehensive version of MSC.Nastran released by MSC.Software. Many major enhancements have been incorporated into this release. They are the support of rotor dynamics; ability to run MSC.Marc inside MSC.Nastran and bring the results back into MSC.Nastran; improved fluid/structure interface calculation by introducing a new body and white method; direct creation of an MSC.Adams MNF file; performance enhancements in optimization including the support of distributed parallel processing (DMP), enhanced support of adjoint method in NVH analysis, and support of eigenvector sensitivity; introduction of two new DMP methods and a more consistent DMP interface; improved rigid elements that support large displacement; enhanced view module calculation and a new thermostat control element in heat transfer; automated external superelement analysis; plus many users' requested enhancements. This section presents a short overview of the most significant features introduced in MSC.Nastran 2004. A more detailed description is available within each section.

**Chapter 1, Introduction.** Gives an overview of new features in MSC.Nastran 2004.

**Chapter 2, Design Optimization.** For MSC.Nastran 2004, major enhancements in optimization include:

- **Eigenvector Sensitivity and Optimization.** Eigenvector sensitivity and optimization can now be requested in SOL 200. This capability provides a foundation for test/analysis correlation in MSC.Nastran.
- **Support of Adjoint Sensitivity in Superelement Analysis.** The adjoint sensitivity analysis method can now be used for superelements in addition to the residual structure.
- **Adjoint Methods as Part of the Analysis.** The adjoint method can now be used in the analysis phase of SOL 200 in frequency response analysis. This capability has been available in the optimization phase since MSC.Nastran Version 70.
- **Distributed Memory Parallel Processing.** Three DMP techniques are now available in SOL 200: Internal parallel for FRRD1 module, frequency domain parallel Lanczos method, and degrees-of-freedom (DOF) domain parallel Lanczos method.
- **New DRESP1 Response Types.** Six new response types (grid point force in two different formats, total element strain energies, psd displacement, psdf velocity, and psdf accelerations) have been added in MSC.Nastran 2004.

- **Performance Enhancements for Large Scale NVH Optimization.** Two major performance enhancements have been added to SOL 200: Support of sparse data recovery in both the analysis and design sensitivity analysis phase, improve data management and program flow in the DSA module.
- **Additional Control Print for Design Response and Constraint.** Previously the P2 parameters on the DOPTPRM entry controls an all-or-nothing type printout. Seven new parameters have been added to provide a finer control of the printout.
- **Support of External Response in SOL 200.** A new entry, DRESP3, using a client/server interface has been added to SOL 200. This feature allows MSC.Nastran to interface with user-defined response quantities using an external program.
- **Improved BEAM Processing.** The internal handling in the design of the PBEAML and PBEAM entries has been streamlined for more efficient design of these properties.
- **Frequency-Dependent Constraints.** Frequency-dependent constraints can now be specified using the TABLEDi entry.
- **Generation of a Complete Updated Bulk Data File in SOL 200 File.** A new option on the ECHO Case Control command allows the user to request a full updated SOL 200 input file for subsequent analysis.
- **Responses Spanning Across Subcases or Superelements.** Different responses from different subcases or superelements can now be combined.
- **Miscellaneous Optimization Enhancements.** Other miscellaneous optimization enhancements include the support of multiple frequency set, support of OFREQ, the use of the latest DOT program, and the option of picking selective design variables.

**Chapter 3. Dynamics.** The following features related to dynamic analysis have been incorporated into MSC.Nastran 2004.

- **Enhanced Methods for Specifying Delay and Phase Angle in Dynamics.** The delay and dphase value(s) can be specified directly on the TLOADi, RLOADi, and ACSRCE Bulk Data entries without referencing a corresponding DELAY or DPHASE entry.
- **Select Modes to be Included for Response Analysis.** A new MODESELECT Case Control is now available to include/exclude certain modes in the dynamic response calculations.

- **Linear Transient Analysis Restarts (SOLs 109 and 112).** A continuation restart from a previous time step can now be performed in SOLs 109 and 112.
- **Enhancements to Initial Condition Usage in Linear Transient Analysis.** Initial condition can now be specified in a modal transient analysis. Furthermore, a static solution can also be used as an initial condition to a linear transient analysis (SOLs 109 and 112.)
- **Residual Vector Enhancements.** Residual vectors are turned on by default in MSC.Nastran 2004. Furthermore, two new Bulk Data entries have been added to create residual vectors due to unit load.
- **Enforced Motion Enhancements.**
- **Random Response Analysis.** Cross-power spectral density, cross-correlation functions, cumulative RMS, log-log approximation of RMS, N0, and CRMS are now supported. Furthermore, you can request PSDF, AUTO, RMS, N0, and CRMS output both in print and punch format without using the xyplot and xypunch commands.
- **Modal Kinetic and Strain Energy Output.** Modal kinetic and strain energy output are now available in SOLs 111, 112, 146, and 200.
- **Enhancements to Structural Damping for Shell and Bush Elements.** Different structural damping can be specified for each of the six degrees of freedom for the CBUSH element. Furthermore, each material field MID1, MID2, MID3, and MID4 on the PSHELL entry can reference different structural damping.
- **Modal Contribution Fraction (MCFRACTION).** Modal participation factors are now available in SOLs 110, 111, and 112.
- **Enhancements to Modal Dynamic Analysis Processing.** Several enhancements have been added to the GKAM module. Machine precision, instead of single precision is used in the generation of modal matrix. If the output eigenvectors are the same as the input eigenvectors, GKAM will no longer generate these redundant eigenvectors. A partitioning vector is generated if a selective set of mode is used. This partitioning vector can be used in subsequent user's dmap.
- **MAX/MIN/RMS Methodology in Data Recovery.** A new sorting option is now available to sort grid point or element data in modal transient response analysis.
- **Massless Mechanism Identification and Control Enhancements.** An improved massless mechanism logic has been added to better distinguish the difference between rigid body modes and massless mechanism.

- **General Enhancements for Orthogonalizing Complex Vectors.** A general new subdmap (ONORM) is delivered in MSC.Nastran 2004 for orthogonalizing vector, including complex vectors.
- **Complex Eigenvalue Analysis Updates.** The number of complex eigenvalues printed will now be based on the ND field request for both the CLAN and QZHESS methods. Furthermore, a couple of minor enhancements have been added to the QZHESS method.
- **Real Eigenvalue Analysis.** The dynamic matrix generation phase for HOU, MHOU, GIV, and MGIV methods now take advantage of the new sparse Cholesky factor. UWM 5407 can now be set to a fatal message by setting system cell 317 to 1.
- **Fast Direct Frequency Response Option.** A fast direct frequency response option has been added for models that consist of solid elements.

#### Chapter 4. Rotordynamics.

- **Introduction.** This section provides an overview of the new rotordynamic capabilities introduced in MSC.Nastran 2004.
- **Examples.** This section contains examples showing various analysis methods using the rotordynamic capabilities.
- **Equations Used in Analyses.** Various equations used in rotordynamics are summarized in this section.

#### Chapter 5. Elements, Loads, and Model Checkout Tools.

- **CBAR/CBEAM Elements.** Offsets for the CBAR/CBEAM elements can now be defined in the element coordinate system in addition to the previously available displacement coordinate system.
- **Nonstructural Mass Enhancements.** Five new options for including non-structural mass by elements lists or specific property lists have been added to MSC.Nastran 2004. These new options are Case Control selectable.
- **Enhanced Element Summary Printout (ELSUM).** Additional options to print the element summary based on element ID and/or property ID have been added to the ELSUM command.
- **Equilibrium Checks.** Equilibrium check can now be performed using the equilibrium Case Control command. This command sums the resultants of the applied loads, spcforces, and mpcforces.

- **New Shell Elements Corner Thickness Option.** A new flag has been added to the shell element for specifying the corner values in terms of fraction rather than actual thicknesses.
- **Strength Ratio Output for Laminated Composites.** A new strength ratio calculation--in addition to the current failure index calculation--is now available for laminated composites.
- **Additional Laminate Options for the PCOMP Entry.** Several new laminate options have been added to the PCOMP entry. This is primarily used as an approximation in the preliminary design where the exact configuration is not known.
- **Temperature-Dependent Composites.** Temperature-dependent composite is now available in SOL 106. Furthermore, thermal strain can optionally be calculated using a more accurate integral approach rather than a secant approach.
- **Rigid Element Enhancements.** A new Lagrange rigid element is now available in SOLs 1010, 103, 105, and 400. This element supports large rotation.
- **QUADR and TRIAR Elements.** An improved QUADR/TRIAR element with drilling degree-of-freedom has been added to MSC.Nastran 2004.
- **CWELD Element Enhancements.** The CWELD element has been enhanced to connect more than a single element per sheet and to handle connections other than surface patches.
- **New K6ROT Default.** The default for the K6ROT is now 100. for all solution sequences.

## Chapter 6. External Superelements.

- **Enhancements to External Superelements.** Several enhancements have been added to the processing of external superelements. This includes a more intuitive interface in the generation of the external superelement, a more automated assembly method, plus the ability to perform data recovery in the assembly run.

## Chapter 7. Coupled Fluid-Structure Analysis.

- **New Body In White Method.** A new body in white fluid/structure interface matrix calculation method has been added in MSC.Nastran 2004.
- **Direct Input of Interface [A] Matrix.** A new A2GG Case Control command allows the user to add the A matrix directly to the model.



- **Acoustic Modal Participation Factors.** A new FLSPOUT Case Control command allows the user to request fluid, structural, and panel participation factors without the use of parameters.
- **Acoustic XY Plots, Random, and Restarts.** Fluid structure, and panel participation factors can be plotted using the MSC.Nastran xyplot.
- **Acoustic Source Change.** The logic for the acoustics source strength is returned to the pre-MSC.Nastran 2001 logic. The scaling for power and source strength should be performed on the TABLEDi and DAREA entry, respectively.
- **Efficiency Enhancement in Virtual Mass.** Efficiency in the virtual mass calculation that were previously available in the vma alter has been incorporated into standard MSC.Nastran.

## Chapter 8. Nonlinear Analysis.

- **MSC.Nastran Implicit Nonlinear -- SOL 600.** A new solution sequence, SOL 600, has been added to MSC.Nastran 2004 that allows the user to optionally execute MSC.Marc from inside MSC.Nastran. The user can request output in either MSC.Marc format or MSC.Nastran format.
- **Prerelease of the General Nonlinear Solution Sequence SOL 400.** A general solution sequence, SOL 400, has been added that supports the Lagrange rigid element plus the ability to perform multiple independent loadings in a single nonlinear run. This solution sequence is a pre-release capability.
- **Performance Improvements in SOL 129.** The adaptive time steps in SOL 129 now accounts for the loading history.
- **Buckling Analysis.** A discussion of various techniques for linear and nonlinear buckling analysis.

## Chapter 9. Numeric Enhancements.

- **Introduction to Parallel Numeric Enhancements.** This section provides a history and overview of the parallel methods available in MSC.Nastran.
- **Domain Decomposition.** This section provides a brief description of the geometric and frequency domain decomposition methods.
- **New Distributed Memory Parallel Methods.** Two new domain decomposition methods are added in MSC.Nastran 2004--DOF based domain decomposition and hierarchical distributed parallel Lanczos method. The DOF method partitions the model after the constraints are eliminated.

- **Inputs.** The DOMAINSOLVER Case Control command has been modified to provide a more consistent user's interface for the different methods.
- **Method Selection Guidelines.** General guidelines for selecting various domain decomposition methods for different solution sequences are provided in this section.
- **Limitations.** This section summarizes the limitations for various distributed parallel methods for different solution sequences.
- **Examples.** This section contains various examples showing the domain decomposition methods for various solution sequences.
- **Parallel Performance Data.** Performance data for 1.8 million degrees-of-freedom model using various distributed parallel methods are shown in this section.
- **Other Numeric Enhancements.** Two other numeric enhancements have been added to MSC.Nastran 2004. They are the Sparse Cholesky Solution and MPYAD Method 1 Storage 3 Option for dense matrix.

## Chapter 10. Heat Transfer.

- **Advanced Sorting Routines for VIEW Factor Shadowing Algorithm.** Advanced sorting routines have been added to the VIEW factor shadowing algorithm.
- **Axisymmetric View Factors for Gray-Diffuse Surface Character.** The adaptive view factor modules have been enhanced for computing view factors for axisymmetric geometries.
- **Thermostat Control with Deadband Applied to a Heat Source.** A new thermostat control with dead band has been added to MSC.Nastran.

## Chapter 11. MSC.Nastran/MSC.Adams Interface

- **ADAMS/Flex Support.** A new ADAMSMNF Case Control command is now available for requesting the MNF file directly in MSC.Nastran for MSC.Adams Flex.

## Chapter 12. Aeroelasticity.

- **Correction of a Machine Dependency in the ZONA51 Supersonic Aerodynamic Method for Planar and Nonplanar Configurations.** It has been found that the algorithm used in ZONA51 that computes the intersection between the downstream and upstream Mach cone from a sending control point to a receiving control point may produce slightly different results between machines. A new algorithm has been implemented in ZONA51 to remove this sensitivity.

## Chapter 13. Miscellaneous Enhancements.

- **Enhancements to MATMOD Module Option 1.** Option 1 of the MATMOD module has been enhanced to allow for extraction of multiple columns in a single operation.
- **Enhancements to MATMOD Module Option 2.** Option 2 of the MATMOD module has been enhanced to allow for filtering matrix based on algebraic value in addition to absolute value.
- **New PARAML Module Options For Modifying Table Data Blocks Via DMAP.** Three new options have been added to the PARAML module for replacing, inserting, or deleting word(s) or records of a table.
- **Option to Reduce Post Processing Data (POST Case Control Command).** A new POST Case Control command has been added to control the amount of output being sent to the plot file.
- **Additional Describers for the ASSIGN Statement.** Two new describers, RECL and SIZE, have been added to the ASSIGN statement.
- **Option to Print the CD Value in the F06 File.** A CD value can now be printed in the .f06 file for grid point related output.
- **Enhancements in the Specification of Parameters.** Parameters can now be specified in the rc file.
- **Enhancement for the DMAP String-based Editor.** An option to allow for interactive editing of user's dmap is available in MSC.Nastran 2004 for unix and linux machines.
- **Consistent Parameter Checking.** A more consistent parameter checking has been added in MSC.Nastran 2004.
- **Punch Output Enhancement.** A new punch option has been added to system cell 210 that will eliminate punching out of the column numbers (columns 73-80).

- **Scale Factor for X2GG and X2PP Matrices.** Scale factors can be added to the X2GG and X2PP type matrices.
- **Direct Input of Structural Element Damping and Area Matrices.** Two new DMIG input--K42GG and A2GG--have been added. The K42GG and A2GG are for structural damping and fluid structure interface matrix [A].
- **MSC.Access, DDLADD Revisions, and XDB Defects Correction.** A mode keyword has been added to the DDLADD to allow for the use of “replace” instead of the default “insert”.
- **Enhanced Free Field Format.** The free field format has been enhanced to allow for including more than 10 fields on a single line.
- **Dynamic Element Forces.**

## Chapter 14. Upward Compatibility

- **Results or Output Changes.** This section contains a summary of output changes in MSC.Nastran 2004.
- **Modifications to OUTPUT2 (INPUTT2) Transport Formats.** This section contains changes to the OUTPUT2 and INPUTT2 formats. This is especially useful with user’s program reading output2 files.
- **Summary of DMAP Module Changes from MSC.Nastran 2001 to MSC.Nastran 2004.** This section contains a list of the DMAP changes between MSC.Nastran 2001 and MSC.Nastran 2004.
- **Summary of Data Block Changes from MSC.Nastran 2001 to MSC.Nastran 2004.** This section contains a list of the Data Block changes between MSC.Nastran 2001 and MSC.Nastran 2004.
- **Removal of Old Features.** This section contains a list of old features that are removed from the code.
- **MSC.Nastran Error List.** This section contains the information regarding the location of the MSC.Nastran error list.

**Appendix A, Statements, Commands, Entries, and Parameters.** This section contains a list of Executive Control statements, Case Control commands, Bulk Data entries, and Parameters that are either new, or have been changed in MSC.Nastran 2004.

- **New and Modified Executive Control Statements for MSC.Nastran 2004.** “[New and Modified Executive Control Statements for MSC.Nastran 2004](#)” on page 622 contains a list of new or modified Executive Control statements.

- **New and Modified Case Control Commands for MSC.Nastran 2004.** “[New and Modified Case Control Commands for MSC.Nastran 2004](#)” on page 623 contains a list of new or modified Case Control commands.
- **New and Modified Parameters for MSC.Nastran 2004.** “[New and Modified Bulk Data Entries for MSC.Nastran 2004](#)” on page 627 contains a list of new or modified Bulk Data entries.
- **New and Modified Bulk Data Entries for MSC.Nastran 2004.** “[New and Modified Parameters for MSC.Nastran 2004](#)” on page 625 contains a list of new or modified Parameters.

## Appendix B. References

- **References.** This section contains a list of useful references.

## Appendix C. Index



## Design Optimization

- Eigenvector Sensitivity and Optimization
- Support of Adjoint Sensitivity in Superelement Analysis
- Adjoint Methods as Part of the Analysis
- Distributed Memory Parallel Processing
- New DRESP1 Response Types
- Performance Enhancements for Large Scale NVH Optimization
- Additional Control Print for Design Response and Constraint
- Support of External Response in SOL 200
- Improved BEAM Processing
- Frequency-Dependent Constraints
- Generation of a Complete Updated Bulk Data File in SOL 200 File
- Responses Spanning Across Subcases or Superelements
- Miscellaneous Optimization Enhancements

## 2.1 Eigenvector Sensitivity and Optimization

### Introduction

Six eigenvector response types have been added to SOL 200 for optimization. They are displacement, stress, strain, element force, element strain energy, and total strain energy. They can be requested by the Bulk Data entry, DRESP1. It uses the existing eigenvector sensitivity solver (Nelson's method) that has been used in SOL 103 with the old design sensitivity capability. Furthermore, the subspace iteration method has been added to account for cases with repeated roots. Eigenvalue sensitivity and optimization has been supported in MSC.Nastran SOL 200 since Version 66. A rudimentary eigenvector sensitivity capability could be activated using PARAM,EIGD,YES in SOL 103. This capability did not, however, extend to optimization. MSC.Nastran 2004 introduces robust eigenvector sensitivity, that includes the effects of rigid element, repeated roots, and superelements.

### Benefits

The addition of eigenvector sensitivity and optimization provides a foundation for incorporating Test/Analysis Correlation into MSC.Nastran. This enables redesign tasks that seek to match prescribed mode shapes from a vibration test. Mode tracking ensures that the constraints on eigenvectors will not be affected by the modes switching.

### Theory

Two different eigenvector sensitivity and optimization methods are introduced in MSC.Nastran 2004. They are the Nelson's method and the Subspace Iteration method.

#### Nelson's Method

The eigenvector sensitivity is based on Nelson's method (Reference 3.) The eigenvalue equation can be expressed as:

$$([K] - \lambda_i[M])\{\phi_i\} = \{0\} \quad \text{Eq. 2-1}$$

Perturbing [Eq. 2-1](#), we arrive at:

$$([K] - \lambda_i[M])\{\Delta\phi_i\} = -([\Delta K] - \lambda_i[\Delta M] - \Delta\lambda_i[M])\{\phi_i\} \quad \text{Eq. 2-2}$$

Because the left-hand-side matrix is singular, the following substitution:

$$\{\Delta\phi_i\} = \{V_i\} + C_{ii}\{\phi_i\} \quad \text{Eq. 2-3}$$



is used, where  $\{V_i\}$  is an arbitrary vector that may contain a contribution from  $\{\phi_i\}$  and hence, may not be orthogonal to  $\{\phi_i\}$ .

Without loss of generality, we can set one of the terms in  $V_i$  to 0.0 and then partition out the corresponding row and column and the rows associated with this term from the right hand side. The term selected correspond to the row of the original eigenvector that is largest in magnitude. If we denote this reduced system with a  $(\bar{\cdot})$  over the terms, we have an n-1 system of:

$$([\bar{K}] - \lambda_i[\bar{M}])\{\bar{V}_i\} = -([\Delta K] - \lambda_i[\Delta M] - \Delta\lambda_i[M])\{\phi_i\} \quad \text{Eq. 2-4}$$

The  $\bar{V}$  term can be expanded to  $V$  by placing a zero in the location that has been partitioned out. The algorithm takes two different paths depending on the normalization method chosen.

For NORM=MAX option, the maximum term is invariant at 1.0 and this is the term used to partition the matrix equations. Therefore, 0.0 is the correct sensitivity for this term and  $V$  provides the sensitivity of the eigenvector.

For the NORM=MASS option, a first order perturbation of the mass normalization equation yields:

$$2\{\phi_i\}^T[M]\{\Delta\phi_i\} + \{\phi_i\}^T[\Delta M]\{\phi_i\} = 0 \quad \text{Eq. 2-5}$$

Substituting [Eq. 2-5](#) into [Eq. 2-3](#) yields:

$$C_{ii} = -\frac{1}{2}(2\{\phi_i\}^T[M]\{V_i\} + \{\phi_i\}^T[\Delta M]\{\phi_i\}) \quad \text{Eq. 2-6}$$

The total sensitivity of the eigenvector can therefore be determined from [Eq. 2-3](#).

## Subspace Iteration Method

Nelson's method assumes that the eigen equation is rank 1 deficient. This assumption is not valid if there are repeated roots. MSC.Software has developed an alternative procedure for this situation based on the Subspace Iteration method. See Reference [4](#). for a complete derivation and application of this method in eigenvalue sensitivity analysis.

Consider the following system equations with a shift  $\lambda_s$ :

$$[K_0 + M_0 \cdot \lambda_s] \cdot [\Phi_0] = [M_0][\Phi_0][\lambda_0 + \lambda_s] \quad \text{Eq. 2-7}$$

Next, define the perturbed system by introducing  $p$ , where  $[\delta K] = K_1 \cdot p$  and  $[\delta M] = M_1 \cdot p$ , and we arrive at the following equations:

$$[(K_0 + M_0 \cdot \lambda_s) + (K_1 + M_1 \cdot \lambda_s) \cdot p][\Phi] = [M_0 + M_1 \cdot p][\Phi][\lambda_0 + \lambda_s] \quad \text{Eq. 2-8}$$

For simplification let

$$K_{os} = K_0 + M_0 \cdot \lambda_s$$

$$K_{1s} = K_1 + M_1 \cdot \lambda_s$$

$$\Lambda = \lambda_0 + \lambda_s$$

We establish the following equations in order to solve for the orthogonal basis  $\Psi$  in Subspace Iteration method:

$$[K_{os} + K_{1s} \cdot p][\Psi] = [M_0 + M_1 \cdot p][\Phi][\Lambda] \quad \text{Eq. 2-9}$$

For efficiency, introduce the following approximation:

$$\Psi = \Psi_0 + \Psi_1 p + \Psi_2 p^2 + \Psi_3 p^3 + \Lambda \quad \text{Eq. 2-10}$$

Using the following transformation,  $\Phi = \Psi \phi$  the perturbed system equations in subspace become:

$$k\phi = m\phi\lambda \quad \text{Eq. 2-11}$$

where:

$$k = \Psi^T [K_o + K_1 p] \Psi$$

$$m = \Psi^T [M_o + M_1 p] \Psi$$

The solution leads to an updated  $\Phi$ , which can be used to compute the next  $\Psi$ . The iteration process will be continued until the following convergence criterion is met.

$$\frac{|\lambda_i^{(k+1)} - \lambda_i^{(k)}|}{\lambda_i^{(k)}} \leq tol \quad \text{Eq. 2-12}$$

The following mode tracking procedure:

$$[C] = [\Phi_0]^T [M][\Phi] \quad \text{Eq. 2-13}$$

is performed in order to account for possible mode switching.

Finally the eigenvalue and eigenvector sensitivities are obtained using the following finite difference equations:

$$\frac{\partial \lambda_i}{\partial p} = \frac{\lambda_i - \lambda_{0i}}{p} \quad , \quad \frac{\partial \Phi_i}{\partial p} = \frac{\Phi_i - \Phi_{0i}}{p} \quad \text{Eq. 2-14}$$

The actual application to eigenvector sensitivities perturbs the stiffness and mass properties for each design variable and computes the new eigenvalues and eigenvectors. This can produce a value for the eigenvalue sensitivity as well, but this method is not utilized in MSC.Nastran.

### Rigid Element Effects with Nelson's Method

With the existence of rigid elements, the  $G_m$  matrix is not invariant, and extra terms need to be added into the pseudo load vector and data recovery formulation.

Consider the computation of pseudo load vectors, starting from the g-size system equations of normal modes analysis:

$$\left\{ \begin{bmatrix} G_m^T & I \end{bmatrix} ([K_{gg}] - \lambda_i [M_{gg}]) \begin{bmatrix} G_m \\ I \end{bmatrix} \right\} \left\{ \phi_n^i \right\} = \{0\} \quad \text{Eq. 2-15}$$

Perturbing [Eq. 2-15](#) and rearranging terms, we arrive at the following equation:

$$\begin{aligned} ([K_{nn}] - \lambda_i [M_{nn}]) \left\{ \Delta \phi_n^i \right\} = & - \begin{bmatrix} G_m^T & I \end{bmatrix} ([\Delta K_{gg}] - \lambda_i [\Delta M_{gg}] - \Delta \lambda_i [M_{gg}]) \begin{bmatrix} G_m \\ I \end{bmatrix} \left\{ \phi_n^i \right\} \\ & - \begin{bmatrix} \Delta G_m^T & 0 \end{bmatrix} ([K_{gg}] - \lambda_i [M_{gg}]) \begin{bmatrix} G_m \\ I \end{bmatrix} \left\{ \phi_n^i \right\} \\ & - \begin{bmatrix} G_m^T & I \end{bmatrix} ([K_{gg}] - \lambda_i [M_{gg}]) \begin{bmatrix} \Delta G_m \\ 0 \end{bmatrix} \left\{ \phi_n^i \right\} \end{aligned} \quad \text{Eq. 2-16}$$

The second and third terms on the right-hand side are additional terms for the rigid element sensitivity analysis.

The following equation:

$$\{\Delta \phi_m\} = [G_m] \{\Delta \phi_n\} + [\Delta G_m] \{\phi_n\} \quad \text{Eq. 2-17}$$

is used for the data recovery of the m-set eigenvector sensitivity.

## Superelements with Nelson's Method

For eigenvalue response, the sensitivity of each superelement is computed by:

$$\frac{d\lambda_k}{dx} = \frac{\phi_k^T \left( \frac{dK}{dx} - \lambda_k \frac{dM}{dx} \right) \phi_k}{\phi_k^T M \phi_k} \quad \text{Eq. 2-18}$$

For the whole structure, the sensitivity can be summed up as:

$$\frac{d\lambda_k}{dx} = \sum_{j=1}^{NSE} \frac{d\lambda_k^j}{dx} \quad \text{Eq. 2-19}$$

As for eigenvector sensitivity, the pseudo load can be treated the same way as in static analysis. Starting from equation:

$$\begin{bmatrix} \bar{K}_{aa} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \begin{Bmatrix} U_a \\ U_o \end{Bmatrix} = \begin{Bmatrix} \bar{P}_a \\ P_o \end{Bmatrix} \quad \text{Eq. 2-20}$$

the O-set DOF can be expressed as:

$$\{U_o\} = [K_{oo}^{-1}]\{P_o\} - [K_{oo}^{-1}][K_{oa}]\{U_a\} = \{U_o^o\} + [G_{oa}]\{U_a\} \quad \text{Eq. 2-21}$$

and the equations are reduced to a-set as:

$$[K_{aa}]\{U_a\} = \{P_a\} \quad \text{Eq. 2-22}$$

where:

$$[K_{aa}] = [\bar{K}_{aa} + G_{oa}^T K_{oa}]$$

$$\{P_a\} = \{\bar{P}_a\} + [G_{oa}^T]\{P_o\}$$

## Inputs

### Case Control

Eigenvector sensitivity is invoked in SOL 200 using the ANALYSIS=MODES Case Control command similar to the eigenvalue sensitivity analysis.

## Bulk Data

The existing DRESP1 Bulk Data interface has been enhanced, making use of the displacement, stress, strain, and element force input format for static analysis.

The DRESP1 entry has the following format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	-etc.-							

For static analysis, the field ATTB must be blank. For eigenvector response, field ATTB is an integer (greater than 0), that represents the mode number.

For normal modes analysis, the RTYPE, ATTA, ATTB, and ATTi fields have the following format.

Response Type	Response Attributes		
RTYPE	ATTA	ATTB	ATTi
DISP	Displacement Component	MODE ID	Grid Id
STRAIN	Strain Item Code	MODE ID	Property ID (PID)
STRESS	Stress Item Code	MODE ID	Property ID (PID)
FORCE	Force Item Code	MODE ID	Property ID (PID)
ESE	Strain Energy Item Code	MODE ID	Property ID (PID)
TOTSE		MODE ID	Property ID (PID)

## Parameter

The new DPHFLG parameter specifies the eigenvector sensitivity and optimization method.

PARAM,DPHFLG,0 (default) selects the Nelson's method.

PARAM,DPHFLG,1 selects the Subspace Iteration method.

MSC.Nastran does not automatically identify repeated roots in the context of eigenvector sensitivity analysis. This method should only be selected when repeated roots (i.e., identical eigenvalues) are expected in the problem because there is a significant performance penalty in the use of the Subspace Iteration method relative to Nelson's method.

## Outputs

There is no significant change for the output format. For eigenvector responses, the output will loop over different modes, similar to the frequency excitation loop in frequency response analysis.

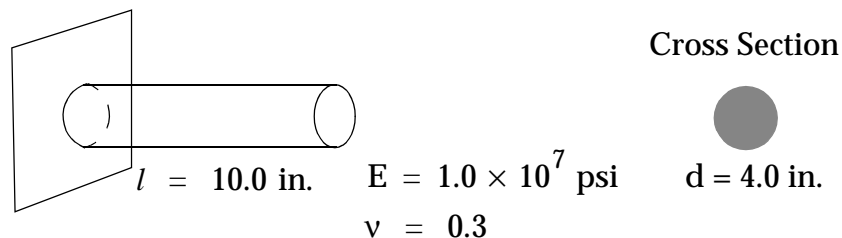
## Guidelines and Limitations

1. Due to the nature of the mathematical operations involved, the runtime when using eigenvector sensitivity can be substantially higher as compared to eigenvalue sensitivity analysis.
2. Subspace Iteration method is introduced for treating repeated roots. It is activated by PARAM,DPHFLG,1. The default value of 0 requests the Nelson's method.
3. Superelements are not supported with subspace iteration.
4. Eigenvector solutions are relative, and a sign change can occur between different runs of the same model when the MASS normalization method is chosen. This can create difficulties in the test/analysis correlation task. Therefore, it is recommended that the MAX normalization method be used.
5. The POINT normalization method is not supported for eigenvector sensitivity analysis.

## Example

(degvtn01.dat)

The following simple example problem demonstrates the capability of eigenvector response in optimization. The goal of this analysis is to match the analytical mode shapes to the prescribed mode shapes—in this case, the test modes.



## Optimization Statement

- Objective Function: Minimize the RMS error between the first analytical mode shape (bending mode) and the first test bending mode.

- **Constraint:** The RMS error between the third analytical mode shape (extension mode) and the corresponding third test mode must be less than 0.002.

Table 2-1 Test Data

Grid	MODE 1		MODE 3	
	Coordinate	Value	Coordinate	Value
3	T3	0.01431	T1	0.1204
6	T3	0.1741	T1	0.5431
9	T3	0.6381	T1	0.9216
Frequency (Hz)	61.912		578.28	

- **Design Variables:** A single design variable, the radius of the 3 elements closest to the root of the beam, is used for this problem.

Model Description

The model consists of a cantilever beam with 10 bar elements consisting of circular cross sections. The radius of the three elements closest to the fixed end is used as the design variable. The other seven elements near the free end have the same invariant radius.

Not all degrees-of-freedom are prescribed as test data. The transverse displacements in z-direction for mode 1 (bending mode) and the longitudinal displacements for mode 3 (extension mode) are chosen. To simplify the problem, only the displacements at grid points 3, 6, and 9 are used. Since the NORMalization method is MAX, it doesn't make sense to use the displacements at the tip, which will always be 1.0 in this case.

Results/Discussion

In a practical design environment, constraining the natural frequencies is recommended, however it is not included here. The job was executed in MSC.Nastran SOL 200. Convergence was achieved in three cycles. The design cycle history from the output file is as follows:

\*\*\*\*\*  
SUMMARY OF DESIGN CYCLE HISTORY  
\*\*\*\*\*

(HARD CONVERGENCE ACHIEVED)

(SOFT CONVERGENCE ACHIEVED)

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED4  
NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS3

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		1.975708E-01		1.248395E+02
1	1.323163E-01	3.400791E-03	3.790751E+01	1.687198E+00
2	7.284324E-05	5.526642E-05	3.180379E-01	N/A
3	5.526642E-05	5.526642E-05	0.000000E+00	N/A

DESIGN VARIABLE HISTORY

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL	:	1	:	2	:	3	:
1	101	RAD1	2.0000E+00	:	4.0000E+00	:	3.9211E+00	:	3.9211E+00	:

The test/analysis mode shape correlation is as follows:

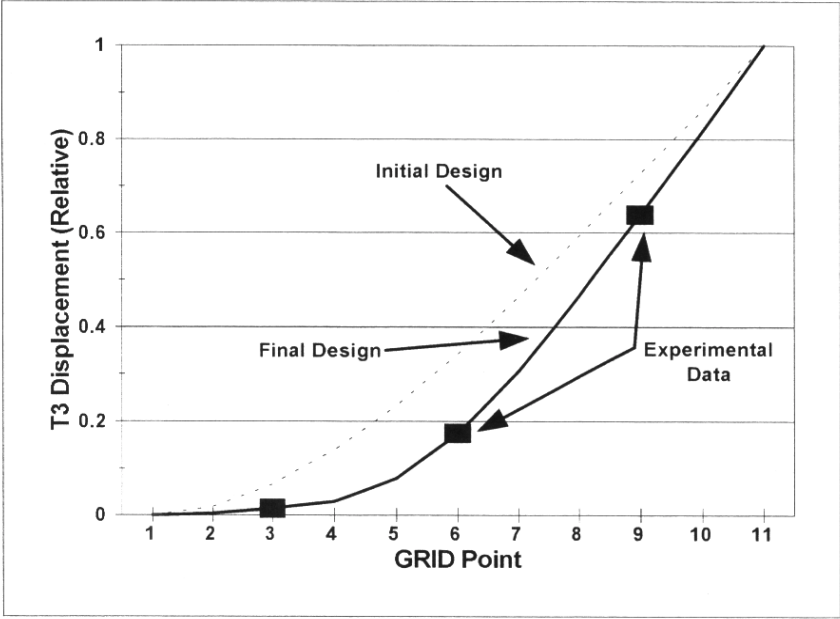


Figure 2-1 Mode 1 (First Bending Mode)



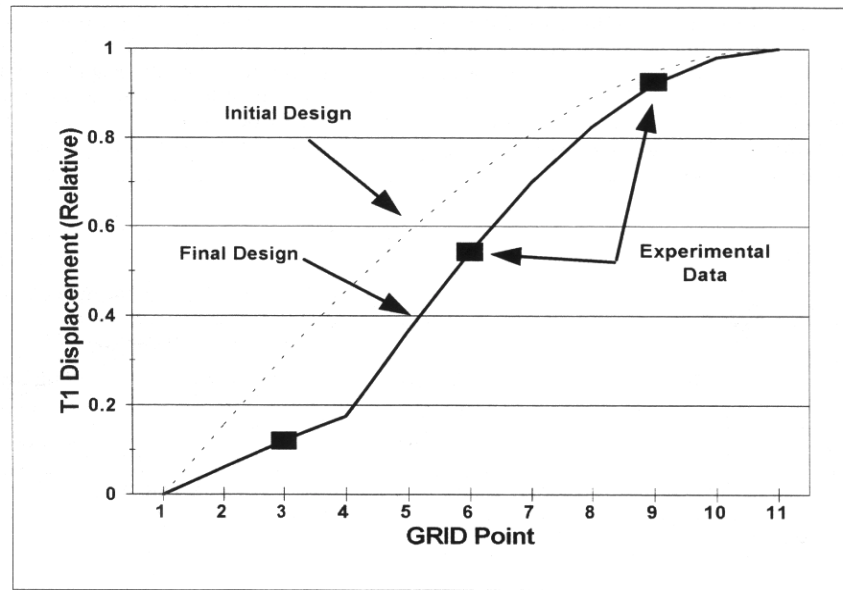


Figure 2-2 Mode 3 (First Extension Mode)

## Input File

```
ID MSC, DEGVN01 $ NEW FOR V2004 LWO 2/1/02
TIME 100
DIAG 6,8
SOL 200
CEND
$
TITLE = EIGENVECTOR SENSITIVITY AND OPTIMIZATION
SUBTITLE = BEAM FROM UAI USER'S GUIDE PROBLEM 25-6
$
$ INITIAL DESIGN: AROOT=4.0, ATIP=4.0
$ OBJECTIVE FUNCTION IS TO MINIMIZE THE DIFF. BETWEEN
$ COMPUTED MODES AND ACTUAL MODES
$
$
$
SPC = 1
$
SUBCASE 1001
  ANALYSIS = MODES
  DISP = ALL
  METHOD = 1
  DESOBJ(MIN)=500
  DESSUB =600
$
BEGIN BULK
PARAM GRDPNT 0
PARAM NORM 1
GRID 1 0. 0. 0.
= *(1) = *(3.0) =
=(9)
$
CBAR 1 1 1 2 0. 1. 0.
```

```

=          *(1)      =          *(1)      *(1)      =          =
=(1)
CBAR      4          2          4          5          0.          1.          0.
=          *(1)      =          *(1)      *(1)      =          =
=(5)
$
PBARL    1          1                      ROD
          2.
PBARL    2          1                      ROD
          2.

$
MAT1     1          3.+7                      .3          .01
$
SPC1     1          123456  1
SPC1     1          26      1          THRU      11
$
EIGR     1          GIV                      4
          MAX
$. . . . . 2. . . . . 3. . . . . 4. . . . . 5. . . . . 6. . . . . 7. . . . . 8. . . . . 9. . . . . 0
DOPTPRM  DESMAX    20          IPRINT    3          P1          1          P2          31
$
$. . . . . 2. . . . . 3. . . . . 4. . . . . 5. . . . . 6. . . . . 7. . . . . 8. . . . . 9. . . . . 0
DESVAR   101        RAD1      2.0        0.1        10.0
DVPREL1  201        PBARL     1          12
          101        1.0

$
$. . . . . 2. . . . . 3. . . . . 4. . . . . 5. . . . . 6. . . . . 7. . . . . 8. . . . . 9. . . . . 0
DRESP2   500        DIFM1     550
          DRESP1    511        512        513
DEQATN   550        F(MD13,MD16,MD19) = SQRT((1.432e-2-MD13)**2+
                                          (1.741e-1-MD16)**2+
                                          (6.381e-1-MD19)**2)

DRESP1   511        MDISP13 DISP          3          1          3
DRESP1   512        MDISP16 DISP          3          1          6
DRESP1   513        MDISP19 DISP          3          1          9
$
$. . . . . 2. . . . . 3. . . . . 4. . . . . 5. . . . . 6. . . . . 7. . . . . 8. . . . . 9. . . . . 0
DCONSTR  600        700          0.002
DRESP2   700        DIFM3     750
          DRESP1    514        515        516
DEQATN   750        F(MD33,MD36,MD39) = SQRT((1.204e-1-MD33)**2+
                                          (5.431e-1-MD36)**2+
                                          (9.216e-1-MD39)**2)

DRESP1   514        MDISP33 DISP          1          3          3
DRESP1   515        MDISP36 DISP          1          3          6
DRESP1   516        MDISP39 DISP          1          3          9
$
ENDDATA

```

## 2.2 Support of Adjoint Sensitivity in Superelement Analysis

The adjoint load method for sensitivity was implemented in MSC.Nastran Version 70 for models consisting of residual structure only. The adjoint load method has been broadened further in MSC.Nastran 2004 to cover models that contain multiple superelements.

### Theory

For grid point response, the sensitivity of a response with respect to design variables,  $x$ , can be expressed as follows:

$$\frac{dr}{dx} = \frac{\partial f^T}{\partial u} \frac{\partial u}{\partial x} \quad \text{Eq. 2-23}$$

where:

$$r = f(u)$$

The adjoint load method solves a solution vector of the form:

$$[\bar{K}]^T \{\lambda\} = \{\partial f / \partial u\} \quad \text{Eq. 2-24}$$

Note that  $[\bar{K}]$  is simply  $[K]$ , the system stiffness matrix, in static analysis. For frequency response:

$$[\bar{K}] = -\omega^2 [M] + i\omega [B] + [K]$$

where  $[M]$  and  $[B]$  are system mass and damping matrices, respectively.  $\{\partial f / \partial u\}$  is referred to as an adjoint load vector since it can be thought of as an additional loading condition for the system. For a response at a specific DOF of a grid, the adjoint load vector is just a  $G$ -size vector with 1.0 at the specific DOF. If responses at different DOFs are involved,  $\{\partial f / \partial u\}$  is simply multi-column adjoint load vectors with each column containing a unique DOF. The  $\{\lambda\}$  is known as adjoint solution vectors. Note that **Eq. 2-24** is very similar to the system equation with  $\{\partial f / \partial u\}$  as load vectors,  $\{P\}$ , and  $\{\lambda\}$  as solution vectors,  $\{u\}$ . Therefore, the  $\{\lambda\}$  can be obtained by going through the same steps in regular solution process. After  $\{\lambda\}$  is available, the sensitivity can be computed as follows:

$$\frac{dr}{dx} = -\{\lambda\}^T [\Delta \bar{K}] \{u\} \quad \text{Eq. 2-25}$$

In the case of multiple superelements, the basic theory of adjoint load method is the same. The generation of adjoint load vectors,  $\{\partial f / \partial u\}$ , is now performed at the superelement level which allows design responses to be anywhere in the model. Since the adjoint load vectors are treated just like any mechanical load vector, they are processed in the same order from tips, to collectors, to residual structure as standard loads. Similarly, the adjoint solution vectors,  $\{\lambda\}$ , for each superelement are recovered using the same steps as in the recovery of regular solution vectors.

Input

There is no input required to perform adjoint analysis with superelements. Instead, the method is selected automatically whenever it is merited based on the statistics of the problem. In general, it is expected to have a performance and disk space advantage over previous versions if the SOL 200 file has grid-related constraints and multiple superelements. The capability can be deselected by setting PARAM AUTOADJ to NO.

Examples

The results of two small existing tpl files (s6339 and s34902b) and a larger modified file (los\_200x) are compared. All three files are used for modal frequency response analysis with multiple superelements.

Model Description

The statistics and results are shown in the following table:

Test Files	s34902b	s6339	los_200x
Number of Design Variables	4	10	13
Number of constraints/frequency	1	1	1
Number of design cycles	1	2	10
Total number of FRRD1 columns (direct)	40	710	1313
CPU time (direct)	10.3	8.9	2086.
Total number of FRRD1 columns (adjoint)	10	71	101
CPU time (adjoint)	10.6	7.0	376.

## Results Discussion

Since s34702b and s6339 are small models, the performance data shown are not meaningful. They do, however, illustrate the use of adjoint sensitivity analysis with superelements.

For s6339, the solution is decoupled. Therefore, the FRRD1 call does not contribute to the performance gain for adjoint load method. Instead, the performance gain can be attributed to the load reduction and solution recovery process for much smaller number of columns in load and solution vectors.

The los\_200x example has about 2300 grid points grouped into 4 Superelements, namely 10, 20, 30 and 0 (residual). This is a superelement variation of the visual sensor example discussed on pp. 58-61 of *MSC.Nastran Version 70 Release Guide* that documented the dramatic gains in disk space and CPU time using the adjoint method.

From the results shown on the previous page, the los.200x model increased speed a more than five-fold when using the adjoint method versus the direct method. This is due mainly to the reduction from 1313 to 101 columns in FRRD1 going from the direct to the adjoint method.

## 2.3 Adjoint Methods as Part of the Analysis

### Introduction

The adjoint method has been available in design sensitivity since Version 70 of MSC.Nastran. However, in certain situations, using the adjoint solution vectors as part of the standard frequency response analysis can result in large performance benefits. This new feature is now available in MSC.Nastran 2004.

### Benefits

Prior to the release of MSC.Nastran 2004, the adjoint solution vectors of the adjoint load method for sensitivity computation were obtained as part of the sensitivity analysis. The solution is carried out in the FRRD1 module and includes a DECOMP (decomposition of real or complex matrix for each forcing frequency) and FBS (forward/backward operation to obtain solution vector for each forcing frequency). For large frequency analysis tasks using the direct method (ANALYSIS=DFREQ), these FRRD1 operations can dominate the overall CPU time and time is saved by solving for the adjoint solution vectors during the original analysis. This enhancement has been added to MSC.Nastran 2004. The sensitivity FRRD1 has been removed such that duplicate DECOMPs are no longer performed and the time spent in FBS is shifted to the first FRRD1 call.

### Inputs

A new parameter, PARAM SOLADJC, is introduced to control the algorithm for the adjoint solution as part of the analysis. The adjoint load vectors will be solved as part of the analysis if the following conditions are met:

1. PARAM SOLADJC is  $\geq 0$  (Default for SOLADJC is 0).
2. The ANALYSIS type is DFREQ.
3. All of the DRESP1 entries are grid responses.
4. The number of DOFs referenced on the DRESP1  $<$  (number of independent design variables + the number of type-2 properties + the number of spawned beam library properties)  $\times$  (the number of frequency steps).
5. For multiple subcases, each subcase has the same set of excitation frequencies.
6. PARAM AUTOADJ is set to YES (default).

## Guidelines

1. Note that failure to meet any of the first five conditions listed previously does not mean the adjoint load method is turned off. It simply means the adjoint load vectors will not be solved as part of the analysis. The adjoint load method may still be selected after the number of responses is reduced by the constraint deletion step of the optimization job.
2. The overhead of DECOMP for a MFREQ job may be insignificant if the problem is decoupled and/or the modal coordinates are small. The DFREQ optimization job makes the best use of the algorithm for adjoint solutions as part of the analysis, but this feature is not available for MFREQ jobs.
3. The algorithm for adjoint load as part of analysis cannot be guaranteed to provide performance benefits. In order to solve the adjoint load as part of analysis, the adjoint load vectors must be available prior to the constraint deletion step in the sensitivity computation. Therefore, all the grid point related responses referenced as constraints must be considered during the generation of adjoint load vectors, which creates extra columns in the solution matrix. However, because the column increase is usually small and since the FBS for a few extra columns will not generally take nearly as much time as DECOMP, the algorithm for adjoint solution as part of analysis still exhibits performance improvement in most DFREQ cases. Where this feature is not anticipated to be beneficial, PARAM SOLADJC can be set to an integer value less than zero.

## Example

A DFREQ job with approximately half a million DOFs in the solution set is used to compare the performance between the new and old path for the adjoint load method.

## Model Description

Here, the old path (ADJPATH1) is the original path and the new path (ADJPATH2) is the new algorithm implemented in MSC.Nastran 2004 for adjoint solution as part of analysis. Note that both ADJPATH1 and ADJPATH2 are available in MSC.Nastran 2004 using PARAM SOLADJC. Other statistics for the job are:

1. 10 forcing frequencies.
2. 5 SUBCASEs.
3. 9 DOFs referenced on DRESP1.
4. 102 independent design variables.

Results/Discussion

The job ran for 3 optimization cycles. Some key performance data are listed in the following table:

	Number of DECOMP in Analysis Phase (total/Cycle)	Number of Columns Produced in FBS in Analysis Phase (per cycle)	Number of DECOMP DSA Phase (Total/Cycle)	Number of Columns Produced in FBS in DS Phase (per cycle)	Total CPU
ADJPATH1	30/10	50	30/10	60	18446
ADJPATH2	30/10	140	0/0	0	10529

Even with the penalty for generating adjoint load vectors prior to the constraint deletion (140 columns for FBS in ADJPATH2 versus 110 columns for FBS in ADJPATH1), the algorithm for adjoint solution as part of analysis required 8000 less CPU sec, a performance gain of 43% reduction. The major contributing factor is the reduction in the number of DECOMPs from 60 to 30. The performance data presented here should not be extrapolated as a general case for DFREQ. The performance improvements depend on the following factors:

1. The ratio of CPU time spent in DECOMP and total CPU time for one cycle. As the ratio goes higher, the better the performance gains.
2. The number of constraints deleted in constraint deletion process. The fewer constraints that are deleted, the better the performance gains.



## 2.4 Distributed Memory Parallel Processing

### Introduction

MSC.Nastran 2004 introduces three DMP techniques for SOL 200: internal parallel for the FRRD1 module, frequency domain parallel Lanczos method, and a degrees-of-freedom (DOF) domain parallel Lanczos method. Distributed memory parallel processing (DMP) was first introduced in MSC.Nastran Version 70.7 for direct frequency response. Although DMP has been extended to other solution disciplines in MSC.Nastran, SOL 200 has not previously used this feature. All computing nodes for a DMP job are turned on only in the analysis phase of the design cycle in SOL 200. During the design sensitivity and optimization phase of a design cycle, the master node is the only active computing node and all other nodes are idle. It is important to note “[Adjoint Methods as Part of the Analysis](#)” on page 28 removes the requirement for significant FRRD1 calls in the sensitivity phase. For more information about new DMP options in MSC.Nastran 2004, see “[New Distributed Memory Parallel Methods](#)” on page 404.

### Internal Parallel for FRRD1 Module

FRRD1 is the solution generator for frequency response analysis. For each forcing frequency, FRRD1 goes through an identical process to produce solution vector(s). Therefore, the work assigned to each processor is equal to the total number of forcing frequencies (NF) divided by the number of processors (NP). If the results of NF divided by NP is not integer, one or more processors will need to pick up one extra forcing frequency. Hence, the maximum load imbalance among all processors in the “internal parallel for FRRD1 module” is the solution time for one forcing frequency. Although this DMP technique is supported in direct frequency (DFREQ) and modal frequency (MFREQ), it more significantly impacts DFREQ jobs.

### Input

To request the number of processors (NP) for a frequency response SOL 200 job, add ‘dmp=NP’ to the job submittal line. Similar to running DMP on non optimization jobs, the Message Passing Interface (MPI) software from your hardware vendor must be properly installed for most machines prior to running an MSC.Nastran DMP job.

---

**Note:** Nodes may be selected for the DMP job. The defaults for the hosts is machine dependent. See “hosts” in “[Keywords and Environment Variables](#)” in Appendix B of the *MSC.Nastran 2004 Installation and Operations Guide* for more details.

---

## Output

The .f06 file for the DMP run using internal parallel for FRRD1 module will look almost identical to a .f06 file from a scalar run (single processor). The output that is unique to the DMP run is shown as follows:

```
*** SYSTEM INFORMATION MESSAGE 6902 (DPERFM)
PERFORMANCE SUMMARY TABLE FOR DISTRIBUTED MEMORY FREQUENCY RESPONSE
NUMBER OF FREQUENCY DOMAINS = 4
NUMBER OF FREQUENCIES = 12
PROCESSOR      # FREQ.      CPU (SEC)      ELAPSED (SEC)
-----
1. pancakesp.scm.na.mscsoft      3      1184.32      1248.49
2. coffeesp.scm.na.mscsoftw      3      1191.58      1214.30
3. jamsp.scm.na.mscsoftware      3      1176.62      1191.29
4. toastsp.scm.na.mscsoftwa      3      1149.80      1189.75

MASTER NODE ELAPSED TIME (SEC) = 1318.36
```

## Example

A vehicle model with 500,000 DOFs in the analysis set is selected. This job is a DFREQ job and has 12 forcing frequencies. A performance study using an IBM SP3 produced the following results.

Number of Processors	Total Elapsed Time	Speed-up	Elapsed Time in FRRD1	Overhead for DMP in FRRD1
1	252 min. 9 sec.	1	208 min. 4 sec.	0
2	139 min. 48 sec.	1.8	105 min. 44 sec.	9 min. 25 sec.
4	101 min. 29 sec.	2.5	67 min. 16 sec.	16 min. 6 sec.

Overhead for DMP in FRRD1 occurs at the beginning and the end of FRRD1. Note that the overhead can be greatly influenced by the workload of the system and network traffic. The overhead is shown here to depict that the overhead can become significant and the efficiency gain from DMP decreases as the number of processors increases. In this case, the FRRD1 module dominates the overall computing requirements, resulting in a significant payoff from running this module in the parallel mode.

## Frequency Domain Parallel Lanczos Method

The frequency domain parallel Lanczos method was first introduced for SOL 103 in Version 70.7. In MSC.Nastran 2004, this technique is also available in SOL 200. Because SOL 200 supports multi-disciplinary optimization, frequency domain

parallel Lanczos in SOL 200 is available for ANALYSIS=MODES and other analyses that use the modal formulation (ANALYSIS=MFREQ, MTRANS, MCEIG or FLUTTER).

Input

1. The following Executive Control Statement must be added to the beginning of the Executive Control Section.
- DOMAINSOLVER MODES (PARTOPT=FREQ)
- An example is shown below:
- NASTRAN bufsize=16385 \$
- domainsolver modes (partopt=freq) \$
- SOL 200
2. Furthermore, the number of processors (NP) must be specified using the ‘dmp=NP’ keyword on the submittal line.
3. The Lanczos method must be used.

**Note:** It is possible to select nodes for the DMP job. The defaults for the hosts is machine dependent. See “hosts” under “[Keywords and Environment Variables](#)” in Appendix B of the *MSC.Nastran 2004 Installation and Operations Guide* for more details.

Output

The .f06 file for a DMP run using the frequency domain parallel Lanczos method 8 is almost identical to that from a scalar run (single processor). The output from the master node, unique to DMP run, is shown below when ‘dmp=4’ was used.

FREQUENCY DOMAIN PARALLEL LANCZOS METHOD				
68	EIGENVALUES FOUND IN DISTRIBUTED SEGMENT # 1			
68	EIGENVALUES FOUND IN DISTRIBUTED SEGMENT # 2			
78	EIGENVALUES FOUND IN DISTRIBUTED SEGMENT # 3			
63	EIGENVALUES FOUND IN DISTRIBUTED SEGMENT # 4			
EIGENVECTOR COLLECTION TIME: 327.1400146 SECONDS				
TABLE OF SHIFTS FOR SEGMENT 1: (LNNRIGL)				
SHIFT #	SHIFT VALUE	FREQUENCY, CYCLES	# EIGENVALUES BELOW	# NEW EIGENVALUES FOUND
1.	-3.9478416E+01	-9.9999994E-01	0	0
2.	3.1974559E+05	8.9995834E+01	68	0
3.	8.2762062E+01	1.4478906E+00	6	68

### Example

A vehicle model with a 500,000 DOFs in the analysis set is used for this example. This is an MFREQ job with 12 forcing frequencies. A performance study on an IBM SP3 produced the following results:

Number of Processors	Total Elapsed Time	Speed-up	Elapsed Time in READ Module
1	245 min. 27 sec.	1	178 min. 57 sec.
2	217 min. 45 sec.	1.13	133 min. 16 sec.
4	158 min. 43 sec.	1.55	79 min. 35 sec.

In this case, the READ module is less dominant in the overall total job elapsed time, but more processors significantly increase speed.

### DOF Domain Decomposition Method

The DOF domain parallel Lanczos method is a new technique introduced in MSC.Nastran 2004 for extracting eigenvalues in parallel mode. SOL 200 supports the degree-of-freedom domain method for MODES and analyses with modal formulation.

### Input

1. The following Executive Control statement must be added to the beginning of the Executive Control Section.  
  
DOMAINSOLVER MODES (PARTOPT=DOF) \$  
  
An example is shown:  
  
NASTRAN bufsize=16385 \$  
domainsolver modes (partopt=dof) \$  
SOL 200
2. Furthermore, the number of processors (NP) must be specified using the ‘dmp=NP’ keyword on the submittal line.
3. The Lanczos eigenvalue method must be used.

---

**Note:** It is possible to select nodes for the DMP job. The defaults for the hosts are machine dependent. See “hosts” under “[Keywords and Environment Variables](#)” in Appendix B of the *MSC.Nastran 2004 Installation and Operations Guide* for more details.

---

## Output

The .f06 file for a DMP run using the DOF domain parallel Lanczos method looks almost identical to a .f06 file from a scalar run (single processor). The output from master node that is unique to DMP run is shown as follows when ‘dmp=4’ was used.

```
*** USER INFORMATION MESSAGE 7465 (PREOUT)
STATISTICS FROM AUTOMATIC MATRIX PARTITIONER
THE NUMBER OF ROWS IN THE MATRIX IS      521489
THE NUMBER OF DOMAINS CREATED USING EXTREME IS      4
THE NUMBER OF DEGREES OF FREEDOM (DOF) IN THE GLOBAL BOUNDARY IS      6733
DOMAIN ID      # INTERNAL DOF      # EXTERNAL DOF
-----
      1      131383      2407
      2      130772      3645
      3      123928      3318
      4      128673      2962

.
.
.

MATRIX DOMAIN PARALLEL LANCZOS METHOD
```

## Examples

A vehicle model with 500,000 DOFs in the analysis set is used for this example. This is an MFREQ job and has 12 forcing frequencies. A performance study on an IBM SP3 produced the following results:

Number of Processors	Elapsed Time	Speed-up	Elapsed Time in READ Module
1	245 min. 27 sec.	1	178 min. 57 sec.
2	228 min. 42 sec.	1.07	149 min. 40 sec.
4	188 min. 49 sec.	1.30	95 min. 3 sec.

## Guidelines

The same guidelines for non-optimization DMP runs apply to DMP in optimization as DMP is only available in the analysis phase. See “[Numerical Enhancements](#)” on page 399 for general recommendations. DMP also supports multi-disciplinary optimization.

## 2.5 New DRESP1 Response Types

### Introduction

Six new response types are available for use in MSC.Nastran 2004 in addition to the six eigenvector responses that have already been discussed in Section 2.1. The new responses provide grid point forces, total strain energy, and grid point power spectral density responses.

### Benefits

Grid point force responses will typically be applied in conjunction with other responses and design quantities to produce a type-2 or type-3 response that can be used in the design of shell structures in general and fuselage structures in particular.

The total energy response is particularly applicable to topology optimization tasks.

The power spectral density (PSD) responses augment the RMS responses that were first available in MSC.Nastran 2001. The availability of the PSD response allows the user to tailor the response at individual frequencies. This can be used with the new “**Frequency-Dependent Constraints**” on page 77, to place a target limit on an output PSD response.

### Theory

In random analysis of MSC.Nastran, the power spectral density (PSD) of a structural response,  $H_{jx}$ , is from Eq. 6-49 in the *MSC.Nastran Advanced Dynamic Analysis User's Guide*:

$$S_j(\omega) = \sum_a \sum_b H_{ja}(\omega) \cdot H_{jb}^*(\omega) \cdot S_{ab}(\omega) \quad \text{Eq. 2-26}$$

where:

$a, b$  = two loading conditions (subcases)

$S_{ab}(\omega)$  = input spectral density

$*$  = complex conjugate of the response

The design sensitivity is the derivative of the PSD response,  $S_j(\omega)$ , with respect to each design variable,  $x$ . Taking the derivative of  $S_j(\omega)$ , the sensitivity for PSD response is

$$\Delta S_j^x(\omega) = \sum_a \sum_b (H_{ja}(\omega) \cdot \Delta H_{jb}^{x*}(\omega) + \Delta H_{ja}^x(\omega) \cdot H_{jb}^*(\omega)) \cdot S_{ab}(\omega) \quad \text{Eq. 2-27}$$

where a  $\Delta H$  is generated for each design variable,  $x$ . Sensitivities obtained from the previous equations can be used in optimization. With PSD responses, engineers can fine tune the vehicle response of a CAE model to a specific pattern that provides analysts with a digital proving ground capability.

Input

The existing DRESP1 entry is used to identify the six new response types. See the *MSC.Nastran Quick Reference Guide* for a complete description of the DRESP1 entry. The format for the new responses is shown in [Table 2-2](#).

Table 2-2

Response Type (RTYPE)	PTYPE	Response Attributes		
		ATTA (Integer > 0)	ATTB (Integer > 0) or Real > 0.0	ATTi (Integer > 0)
GPFORCE Remark 1, 3	Grid ID	Component	Blank	Element ID
GPFORCP Remark 2, 3	Blank	Grid ID	Blank	Orient ID
TOTSE Remark 4	Blank	Blank	Mode Number	Blank
PSDDISP	RANDPS ID	Component	Frequency (Blank, Real > 0 or Character)	Grid ID
PSDVELO	RANDPS ID	Component	Frequency (Blank, Real > 0 or Character)	Grid ID
PSDACCL	RANDPS ID	Component	Frequency (Blank, Real > 0 or Character)	Grid ID

Remarks

1. The RTYPE=GPFORCE entry is used to define grid point forces of linear elements with the user specifying PTYPE=Grid Point ID, ATTA=load component, and ATTi=Element ID. Multiple components are supported for the ATTA field and multiple Element IDs are supported for the ATTi input.



2. Parameter PARAM, NOELOC, +1 will cause the output of the sum of the forces parallel to the edges of adjacent elements. The RTYPE=GPFORCP entry is used to define the element elastic forces with user's given ATTA=Grid Point ID and ATTi=Orient Point ID. Multiple Orient IDs are supported for the ATTi input.
3. Case Control command, GPFORCE, and/or the parameter, PARAM, NOELOC, 1 are not required to be present for the DRESP1 response.
4. For static analysis with RTYPE=TOTSE, the field ATTB must be blank. For eigenvector response, field ATTB is an integer > 0 that specifies the mode number.
5. The use of the PTYPE field for specifying the RANDPS ID on the PSDxxxx responses differs from the RMSxxxx responses that use the ATTB field.  
The formatted print of all response types is supported through the DSAPRT Case Control command.
6. The ATTB field of the PSDxxxx plays the same role as it does to FRXXXX response. A blank value creates a response at all frequencies, a positive real number creates a single response at the excitation frequency closest to the value, and character input invokes a mathematical function that results in a scalar quantity that is computed based on the response at all frequencies.

## Output

Representative DRESP1 responses are shown in [Figure 2-3](#), [Figure 2-4](#) and [Figure 2-5](#) for Grid Point Force, Total Strain Energy and PSD Responses, respectively.

----- GRID POINT FORCE BALANCE -----								
INTERNAL ID	DRESP1 ID	RESPONSE LABEL	GRID ID	ELEMENT ID.	COMPONENT NO.	LOWER BOUND	VALUE	UPPER BOUND
2	3	GPFORCE	20201	2	3	2.5000E+04	2.5562E+00	N/A
3	3	GPFORCE	20201	2	4	2.5000E+04	-5.5577E-02	N/A
4	3	GPFORCE	20201	25	3	2.5000E+04	2.7135E+01	N/A
----- SUMMATION OF ELEMENT FORCES ON ADJACENT ELEMENTS -----								
INTERNAL ID	DRESP1 ID	RESPONSE LABEL	GRID ID	ORIENT ID	LOWER BOUND	VALUE	UPPER BOUND	
22	1	GPFORCP	20201	20101	2.5000E+04	2.6957E+01	N/A	
23	2	GPFORCP	20101	20201	2.5000E+04	1.5071E+02	N/A	
24	4	GPFORCP	10200	10100	2.5000E+04	5.4318E+01	N/A	

**Figure 2-3 Grid Point Force Responses**

The design response output for the total strain energy response is shown in [Figure 2-4](#).

----- EIGENVECTOR TOTAL STRAIN ENERGIES -----						
INTERNAL ID	DRESP1 ID	RESPONSE LABEL	MODE	LOWER BOUND	VALUE	UPPER BOUND
4	4	TSEL	1	N/A	1.3398E+04	1.5000E+00

Figure 2-4 Total Strain Energy Response

----- PSD DISPLACEMENT RESPONSES -----								
INTERNAL ID	DRESP1 ID	RESPONSE LABEL	GRID ID	COMPONENT NO.	FREQUENCY	LOWER BOUND	VALUE	UPPER BOUND
3	101	MING5T2	5	2	8.1418E-01	N/A	2.1945E+00	N/A

Figure 2-5 PSD Displacement Response

Guidelines and Limitations

- 1. The grid point force design responses are supported only for linear static analysis. The total strain energy is only supported for linear static and real eigenvector analyses.
- 2. Only sizing design variables are supported for the cases of RTYPE=GPFORCE/GPFORCP and TOTSE.
- 3. The DESSUB or DESOBJ request that invokes a PSDXXXX should always be placed in the first subcase that is invoked by the associated RANDPS ID.
- 4. Multiple RANDPS IDs can be referenced in a single design task.
- 5. The frequency dependent constraint limits discussed in Section 2.12 can be used to readily specify a target for a PSD response.
- 6. A direct method in SOL 200 is used for the sensitivity computation of total strain energy responses. In some cases, the total strain energy responses can be equivalent to structural compliance in a design optimization model. The structural compliance can be defined by a DRESP2 entry and its sensitivity is calculated by the adjoint method whenever only RTYPE=DISP is present. For a problem with a large number of design variables, the adjoint method is much more efficient than the direct method. Thus, the DISP response is recommended as the preferred alternative to the TOTSE response.

Examples

Three small examples show the use of each of the three new response types.

## Example 1

**Listing 2-1** shows some input data entries to illustrate the use of grid point force responses. The output from a DSAPRT Case Control command is shown in **Figure 2-6** for the grid point force responses.

**Listing 2-1 Input File with Grid Point Force Responses (gpfds1.dat)**

```
$DRESP1, ID, LABEL, RTYPE, PTYPE, REGION, ATTA, ATTB, ATT1, +
$, ATT2, ...
$
DRESP1, 1, GPFORCP, GPFORCP, , , 20201, , , 20101
DRESP1, 2, GPFORCP, GPFORCP, , , 20101, , , 20201
DRESP1, 3, GPFORCE, GPFORCE, 20201, , 34, , , 2,
, 31, 25, 26
DRESP1, 4, GPFORCP, GPFORCP, , , 10200, , , 10100
, 20100, 10300, 10201
DRESP1, 5, GPFORCE, GPFORCE, 10203, , 123, , , 24,
, 7, 23, 11
```

*****									
* DESIGN SENSITIVITY MATRIX OUTPUT *									
* RESPONSE SENSITIVITY COEFFICIENTS *									
*****									
-----									
DRESP1 ID=	5	RESPONSE TYPE= GPFORCE		ELEM ID= 11		COMP NO= 1		GRID= 10203	
SUBCASE	RESP VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE
1	-7.9501E+02	1	TP	-7.6904E+02	2	TW	5.4016E+02	3	BA
2	2.0985E+03	1	TP	2.0996E+03	2	TW	-4.5166E+02	3	BA
-----									
DRESP1 ID=	4	RESPONSE TYPE= GPFORCP		GRID ID= 10200		ORIE ID= 10100		SEID= 0	
SUBCASE	RESP VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE
1	5.4318E+01	1	TP	-7.8328E+01	2	TW	5.9319E+01	3	BA
2	-1.3818E+02	1	TP	-3.4485E+02	2	TW	1.5793E+02	3	BA
-----									

**Figure 2-6 Formatted Sensitivity Output with Grid Point Force Responses**

## Example 2

### Model Description

To illustrate how total strain energy response is defined, consider a ten-bar truss problem shown in **Figure 2-7** subject to a vertical load. The design goal is to minimize the total strain energy (i.e. maximum global structural stiffness) with an allowable mass change that must be within 50% of the design domain. Here, the density approach is used to carry out topology optimization. A detailed description of topology optimization using SOL 200 material design sensitivity capability can be found in the *MSC.Nastran Version 70.7 Release Guide*.

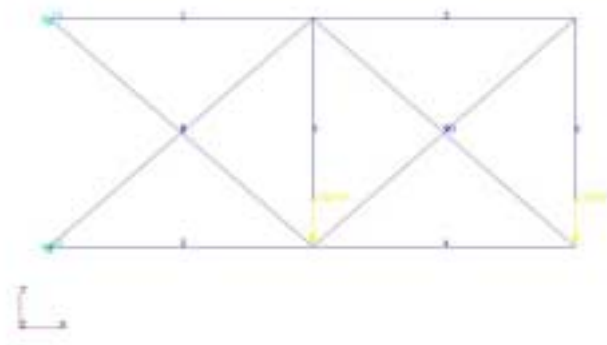


Figure 2-7 Ten-bar Truss Subjected to Two Loads Acting Simultaneously

Theory

In the density approach, the design variable is the normalized density  $x = \rho/\rho_0$  for each design element, where  $\rho$  and  $\rho_0$  are the intermediate and physical densities. In addition, the intermediate Young’s modulus  $E$  is related to a design variable by  $E = X^\eta \times E_0$  where  $E_0$  is the physical Young’s modulus. The exponent  $\eta$  is a penalty value. In SOL 200, the DVMREL1 entry is used to relate intermediate density to a design variable. The DVMREL2 and DEQATN entries are used to relate an intermediate Young’s modulus to design variables.

Inputs

Entries required for a design element are listed in Listing 2-2. Notice that the DEQATN entry can be referenced repeatedly by all the design elements.

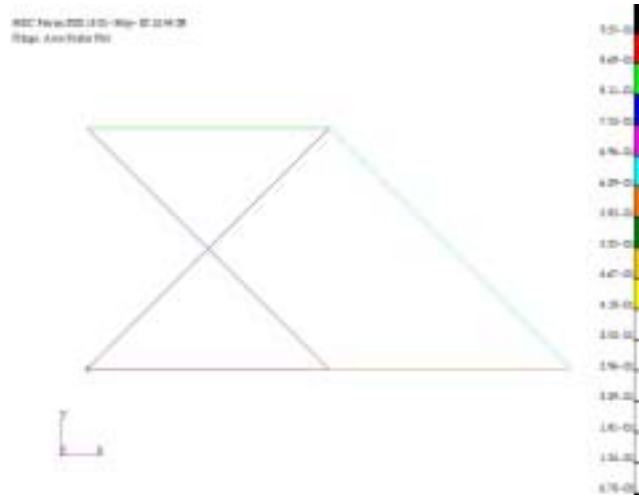
Listing 2-2 Selected Design Data for mintse10.dat

CROD	1	1	5	3	
PROD	1	1	5.0		
MAT1	1	2.07+5		.3	7.93-6
DESVAR	1	DENSITY	.5	0.0	1.0
DVMREL1	1	MAT1	1	RHO	
	1	7.93-6			
DVMREL2	1	MAT1	1	E	200
		DESVAR	1		

```
DEQATN    200    YOUNG(X) = 2.07e+5 * x **2
DRESP1     1    TOTSE    TOTSE
```

### Results/Description

The optimization job converges after 18 design cycles. **Figure 2-8** shows the optimum layout design that is the same as the best topology results presented in the literature. It is seen that 4 of the truss members are not needed.



**Figure 2-8 Optimal Topology**

### Example 3 - d200psd21.dat

#### Model Description

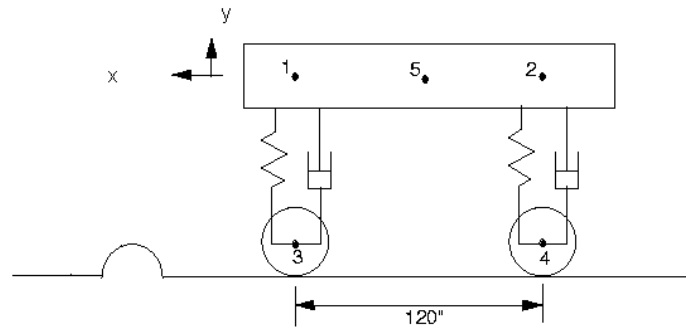
An extremely simplified car model, see **Figure 2-9**, is subjected to random loading that is uncorrelated. The goal is to minimize the maximum PSD displacement of the vehicle CG while constraining the stress. Design variables are the stiffness and damping values (spring and shock absorber) of the suspension system. Numerical values for the design limits are:

$$100. < K < 1000.$$

$$1.0 < B < 10.$$

$$|\sigma| \leq 4500.$$

Eq. 2-28



**Figure 2-9 Simplified Car Model**

### Inputs

An abbreviated input file showing only the relevant design optimization entries is given in [Listing 2-3](#). This case requires two subcases – one for the front wheel and one for the back. A unit enforced displacement is applied to each wheel using the enforced motion method. Constraints are applied to the stresses resulting from these unit displacements in the individual subcases while the maximum PSD response is calculated by selecting the maximum PSD response among all forcing frequencies.

### Listing 2-3 Design Optimization Input File (partial)

```
TITLE = SIMPLE CAR WITH RANDOM INPUT
SPC = 100
FREQUENCY = 130
STRESS(PHASE) = ALL
DISP(PHASE) = ALL
DESOBJ= 101
ANALYSIS = DFREQ
DESSUB = 800
$
$ THIS RANDOM CASE CONTROL CALLOUT IS NEEDED ONLY
$ FOR THE XYPLOT REQUEST WHEN USING OPTIMIZATION
$
RANDOM = 1000
SUBCASE 1
  DLOAD = 111
SUBCASE 2
  DLOAD = 112
$
OUTPUT (XYPLOT)
XTITLE = FREQUENCY (HZ)
YTITLE = DISP PSD AT GRID PT 5
XYPUNCH DISP PSDF /5(T2)
$
BEGIN BULK
.
. ANALYSIS MODEL NOT SHOWN
```

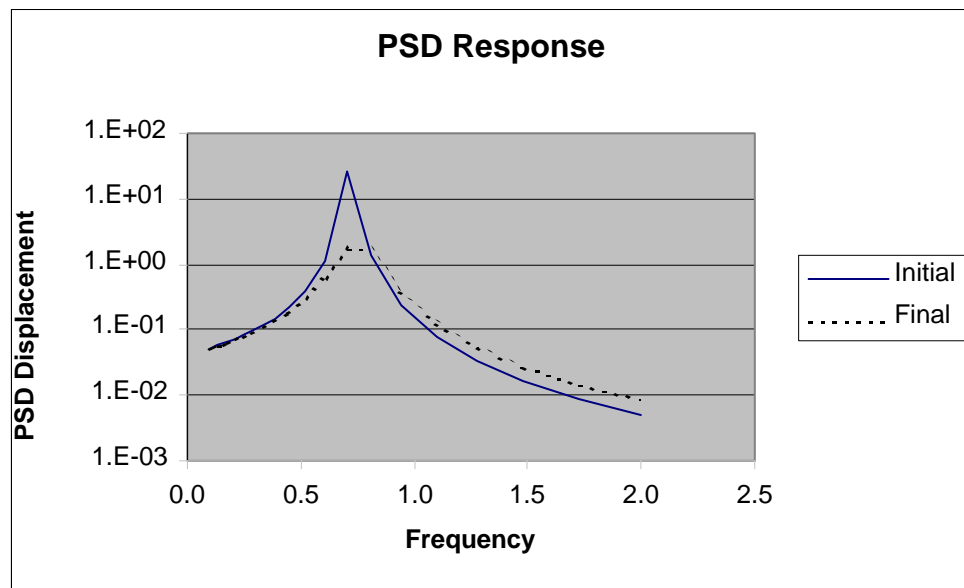
```

.
$
$ -----
$
$   DESIGN MODEL
$
$
DESVAR,1,K2,1.,0.5,5.
DESVAR,2,B2,1.,0.5,5.
$
DVPREL1,101,PBUSH,1000,K2,100.
,1,200.
DVPREL1,201,PBUSH,1000,B2,1.
,2,2.0
$
DRESP1,101,MING5T2,PSDDISP,1000,,2,MAX,5
$
DCONSTR,800,801,,4500.
DCONSTR,800,802,,4500.
DRESP1,801,E5PTC,FRSTRE,PBAR,,12,,11
DRESP1,802,E5PTE,FRSTRE,PBAR,,14,,11
$
DOPTPRM DESMAX 20 P1      1      P2      14      METHOD      3
$
ENDDATA

```

## Results/Description

**Figure 2-10** shows the initial and final PSD plots. The maximum PSD response for the CG has been reduced from 26.6 to 1.71 in 6 design cycles. The final design variables put the stiffness at 223 and the damping at 10.



**Figure 2-10 Initial and Final PSD Plots for the Simplified Car Model**

## 2.6 Performance Enhancements for Large Scale NVH Optimization

### Introduction

Major automobile manufacturers have a growing need for efficient design optimization capabilities. Many of these models are high fidelity FE models (millions of DOFs) to perform complex dynamic optimization tasks such as NVH optimization. NVH optimization was first made available in Version 70 of MSC.Nastran when the adjoint method was introduced. The capability has been further improved by taking advantage of the vectorization for the triple matrix product in DSADJ in MSC.Nastran 2001. In MSC.Nastran 2004, two major performance improvements have been added to SOL 200. Specifically, they are:

1. Support sparse data recovery (SDR) in the analysis phase and the design sensitivity analysis (DSA) phase. The key idea behind this enhancement is to use the sparse data of a design model and an analysis model to improve the SOL 200's performance. The sparsity of a design model is measured by how many elements/grids are designed or constrained relative to the original model. Similarly, the sparsity of the analysis model is measured by how many elements/grids are affected by output requests. A design model that consists of only a fraction of the original model, say 20% of a model with multi-million DOFS, avoids the expensive operation to recover the full g-size solution vectors. Subsequently, the reduced solution matrix can be used in the DSA phase by modules DSADJ and DSVG1 for efficient computations.
2. The DSA process is expedited by improving the data management and program flow in module DSADJ. Inside module DSADJ, the triple matrix product for each designed element is a numerically intensive operation. To augment the existing algorithm that holds the full g-size solution vectors in the memory (method I), the new approach (method II) only holds the active components of the solution vectors.

In addition, initial studies have been made into the applicability of ACMS (Automated Component Modal Synthesis) technology in SOL 200. This subsection provides guidelines and limitations for the application of ACMS in SOL 200.

### Benefits

This enhancement allows users to run large scale NVH jobs with substantial improvement in runtime plus the reduction in memory and I/O requirements. The example shown in [Table 2-3](#) illustrates that the run time for a typical NVH



optimization job can be reduced to hours from days, with the added benefits of disk space reduction. In addition, the newly added DMP support in SOL 200 (see “**Distributed Memory Parallel Processing**” on page 31) provides additional performance improvement to key modules (e.g., READ). Finally, although the performance enhancements were originally implemented for the NVH optimization tasks, other large scale optimization tasks will also benefit from this enhancement.

## Inputs

Sparse Data Recovery is available by default. The following two parameters can be used to disable it:

1. PARAM,SPARSEDR,(YES)/NO

Controls whether Spare Data Recovery (SDR) should be performed in the analysis phase. Its default is to perform SDR. This parameter is recognized in SOLs 103, 108, 111, 112, and 200. The default for SPARSEDR is YES.

2. PARAM,SPARSEDM,(YES)/NO

This parameter controls whether the SDR should be performed in the design sensitivity analysis phase of SOL 200. Its default is to perform SDR.

The ADJMETH parameter be set to 1 when disk space is critical.

PARAM,ADJMETH,(0)/1/2

Default = 0. By default, MSC.Nastran automatically selects the better of the two methods. ADJMETH=2 only holds the active solution vectors (new method). ADJMETH=1 holds the full g-size solution vector (old method) and can be used when disk space is an issue.

ACMS in SOL 200 requires the following modifications to a nonsuperelement file:

1. In the executive file, add  
DOMAINSOLVER ACMS \$  
and
2. In the Bulk Data, add  
PARAM,AUTOQSET,NO

## Outputs

The performance enhancements do not affect accuracy or the format of various results generated from the design optimization process. However, a new system information message (SFM 7603) is printed to indicate which method is being selected and how it is selected. The following is a sample message printed out in the .f04 file:

```
*** SYSTEM INFORMATION MESSAGE 7603 (DMKCRD)
METHOD II IS SELECTED BY THE PROGRAM TO PERFORM TRIPLE MATRIX PRODUCT
```

Guidelines and Limitations

- 1. The sparse data recovery is supported for the following disciplines: analysis=modes, dfreq, mfreq, and mtran.
- 2. The performance improvement is directly affected by combinations such as the size of the problem, the sparsity of the design model, and the available memory space.
- 3. The sparser the design model, the higher the speed up. Similarly, if the output request of the analysis results involves only a few dofs of the analysis model, the job will also run faster. For a large scale NVH optimization task, it is generally recommended that the output request be limited.
- 4. Module DSADJ selects the processing method that is deemed most efficient. When disk space is an issue, PARAM ADJMETH can be set to 1 to use less disk space, at the expense of a performance penalty.
- 5. The benefits of ACMS are most evident for large models (> 1 million DOFs) with hundreds of extracted modes.
- 6. ACMS benefits erode if the design model extends over a large portion (> 5%) of the finite element model.
- 7. ACMS in SOL 200 may perform best if a single tree (DOMAIN SOLVER ACMS TREE=SINGLE) is specified.
- 8. ACMS in SOL 200 is only operational for DMP=1 (default).

Example

Industry Example of an NVH Model

This is a full vehicle NVH optimization task. The model statistics are given in [Table 2-3](#).

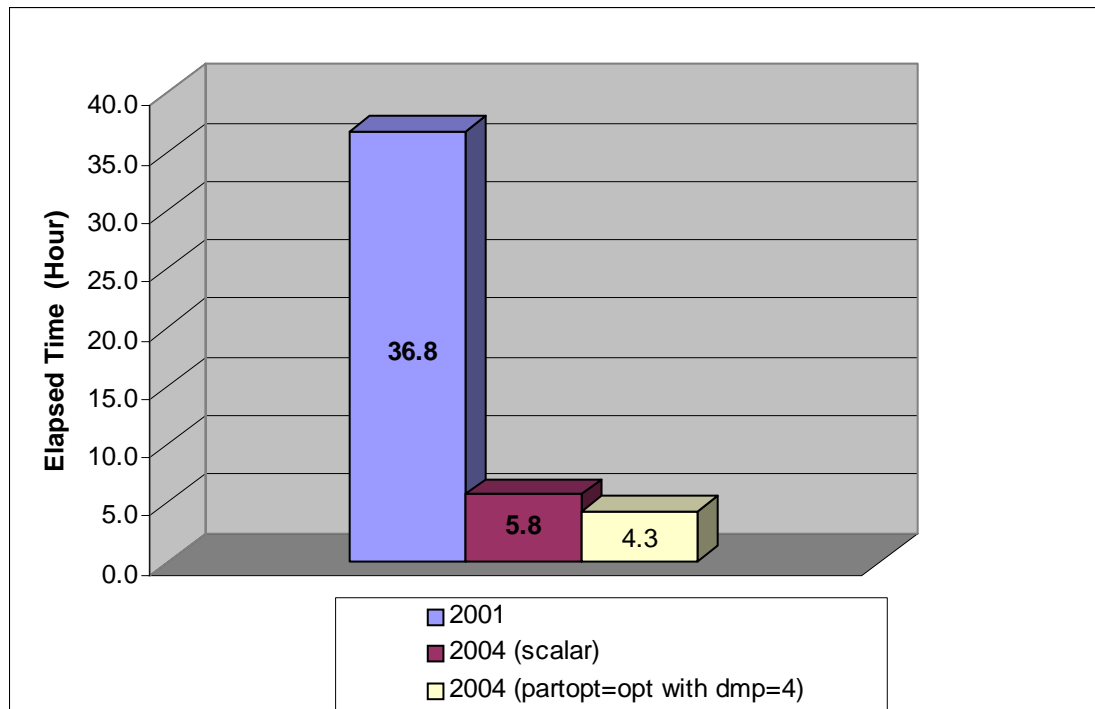
Table 2-3 Model Statistics for Industry Example 1

Objective: minimize weight
Constraints: limits on selected acceleration components
A full detailed vehicle model (2.2 Million dofs)
162 excitation frequencies per load case with seven load cases
251 modes

**Table 2-3 Model Statistics for Industry Example 1**

128 design variables and 1431 retained responses
Design set = 0.53 Million dofs (25% of the original model)

The job is run on an IBM-AIX SP2 machine with and without DMP option. **Figure 2-11** shows the total improvement for one design cycle among three runs. It is clear that MSC.Nastran 2004 reduced the turnaround time from days to hours. In addition, comparing two 2004 runs, the run with DMP option further reduced the elapsed time by 25% for one design cycle. This additional time saving is due to the 50% performance gain on the Lanczos Eigenvalue Extraction method using partopt=dof and dmp=4.

**Figure 2-11 Total CPU for One Design Cycle**

The next two figures report the breakdown of the performance data. **Figure 2-12** compares the CPU time spent on the solution data recovery in submap DISPRS while **Figure 2-13** compares the CPU time spent in module DSADJ. For this model, the data recovery and adjoint method show a performance improvement of 20 and 8 fold, respectively.

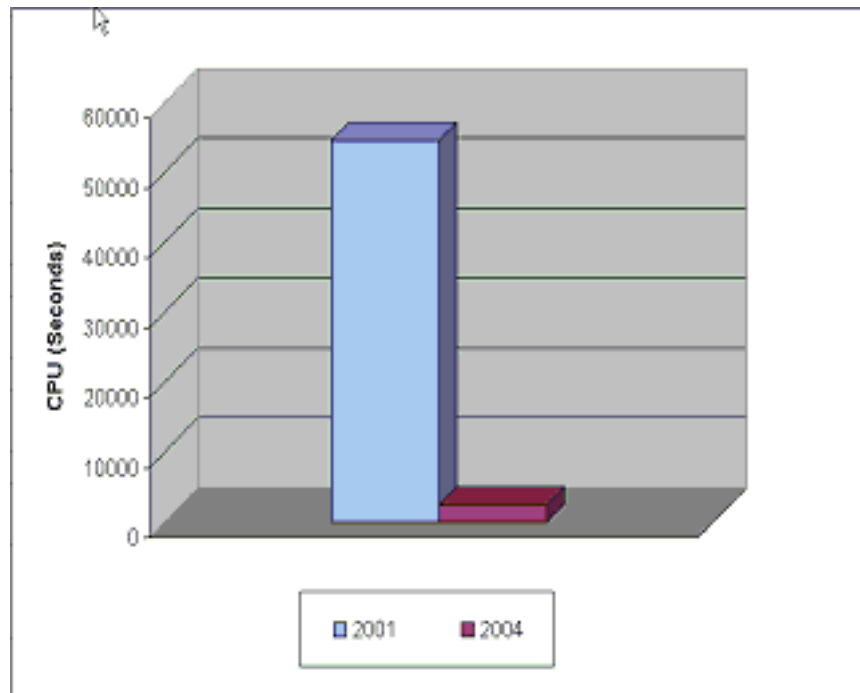


Figure 2-12 Total CPU for Data Recovery

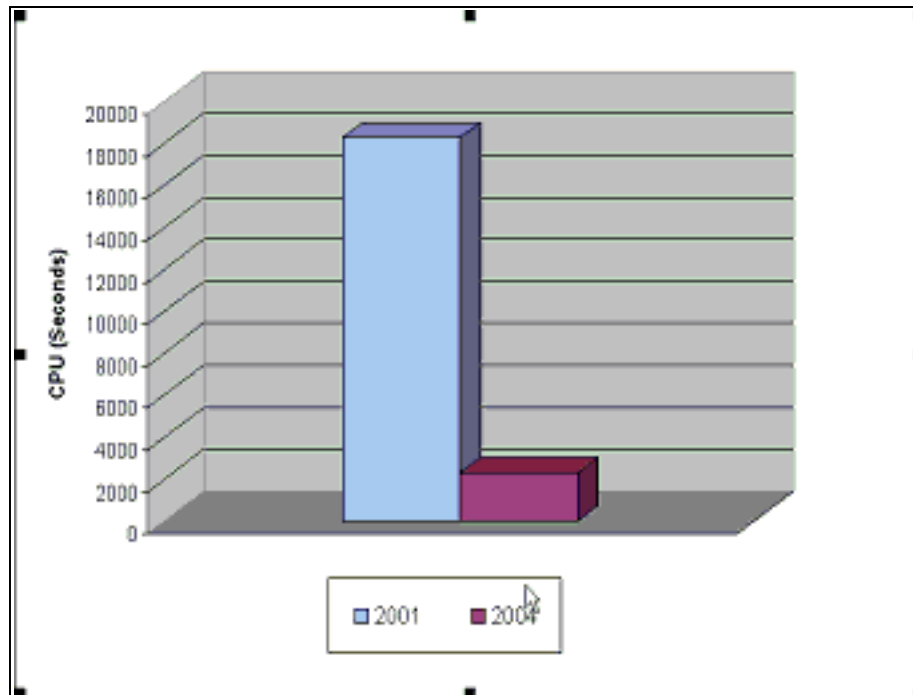


Figure 2-13 Total CPU for DSADJ

## 2.7 Additional Control Print for Design Response and Constraint

### Introduction

MSC.Nastran 2004 has added parameters that allow the user to control the quantity of design response and constraint prints that are produced. The design sensitivity and optimization capability in MSC.Nastran produces results for design constraints and responses that can be printed to the .f06 file using the P2 parameter on the DOPTPRM entry. Previously, this was an “all-or-nothing” process in which the print parameters were either on or off, resulting in large volumes of output for large models with hundreds and thousands of design responses and constraints.

### Input

Eight new parameters have been added to the DOPTPRM entry. The first seven parameters control constraint prints while the eighth parameter controls the response prints.

Name	Description
P2CR	Maximum number of Constraints on Responses to be printed
P2CDDV	Maximum number of Constraints on Dependent Design Variables to be printed
P2CP	Maximum number of Constraints on Properties to be printed
P2CC	Maximum number of Constraints on Connectivity properties to be printed
P2CM	Maximum number of Constraints on Material properties to be printed
P2CBL	Maximum number of Constraints on Beam Library dimensions to be printed
P2CALL	Maximum number of Constraints of all categories to be printed
P2RSET	SET1 ID for which a SET of Response IDs is defined to be printed.

### Guidelines and Limitations

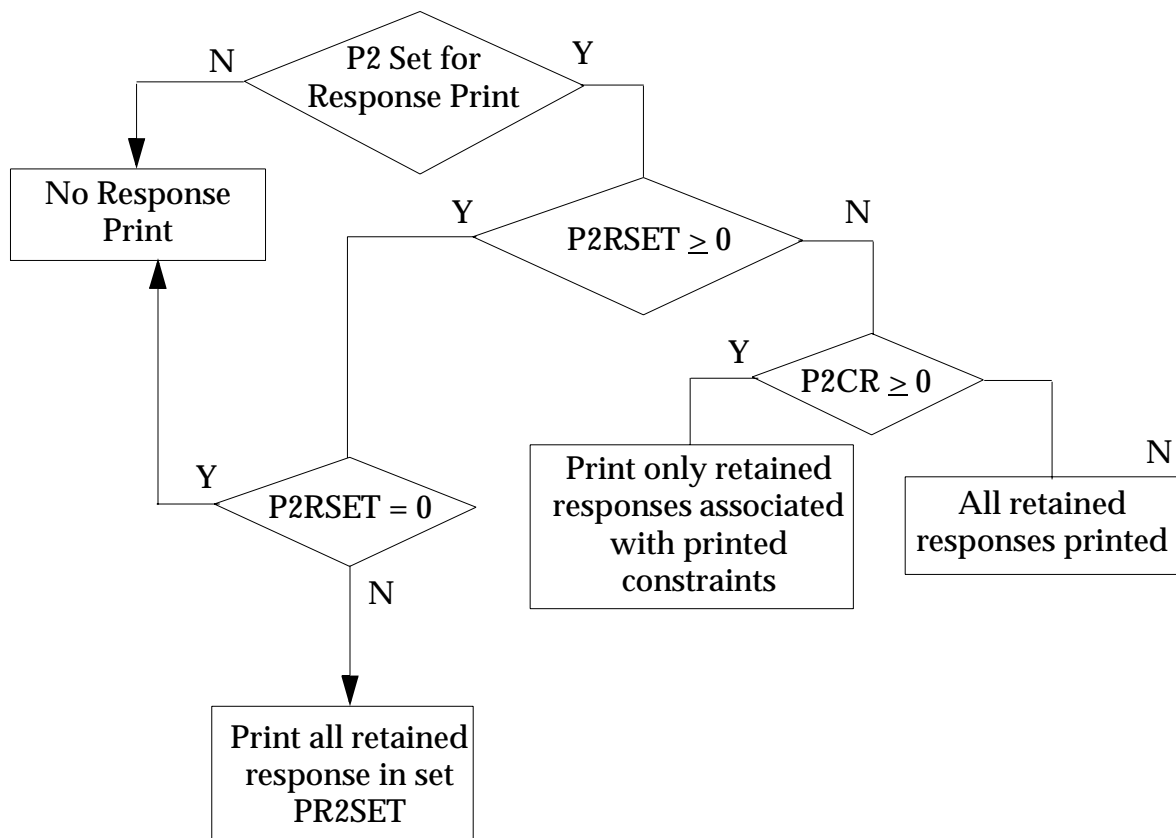
1. It is still necessary to set P2 to a value that enables printing of the constraints and/or responses.
2. If none of the above eight parameters is included on the DOPTPRM entry, the print behavior is unchanged from earlier releases of MSC.Nastran.

- 3. P2CALL can be used to provide a default for any or all of the categories. For example, P2CALL = 10 will print up to 10 of each type of constraint. If there are fewer than ten constraints of a given type available, only the available number will be printed. Note that P2CALL applies to each type of constraint individually. No attempt is made to find and only print out the most critical constraints across all types of these constraints.
- 4. The remaining P2Cx parameters allow you to selectively specify the number of constraints printed for that constraint type. If the number specified is fewer than the number available, a sort is performed to find the most critical P2Cx parameters and only these are printed. If the number specified exceeds the number of constraints available, no sorting is performed and all the constraints are printed in the same order of increasing internal constraint ID as is done currently.
- 5. Some users have gained insight into the performance of the optimization algorithm by setting the DOPTPRM IPRINT parameter to a value that shows the constraint values during the optimization process. These constraint values are in the internal constraint order associated with the constraint evaluation and is not affected by the print order produced with the P2Cx parameter. The internal constraint ID is still printed when the P2Cx parameter is used, but the constraint are unlikely to be in ascending order of this ID.
- 6. P2RSET can be used to print a selected set of responses. P2RSET refers to a SET1 Bulk Data entry that contains a list of DRESPx IDs to be printed. Only retained responses (i.e., those that survived the constraint screening process) are printed.

Table 2-4 Controls for Print Response

P2CR	P2RSET	Constraints on Response	Responses
Used to create constraint set A	Yes with set B	Set A	Set B (P2CR has no affect)
No (means ALL)	Yes with set B	All	Set B
Used to create constraint set A	No	Set A	Print responses associated with set A
No (means ALL)	No	All	All

The print of responses can now be affected by three DOPTPRM parameters: P2, P2CR and P2RSET. **Figure 2-14** depicts how the three parameters affect the print while scenarios for different paths are shown in **Table 2-4**.



**Figure 2-14** Flowchart Showing How P2, P2RSET and P2CR Affect the Print of Retained Design

## Example

A brief example demonstrates the reduced print available with the new parameters. A case was run with the following DOPTPRM entry:

```

$
DOPTPRM    APRCOD    1    DELP    0.15    P1    1    DESMAX    2
           P2        15    IPRINT    0    P2CR    3    P2CP    3
$

```

Only the three most critical constraints on responses and the three most critical constraints on properties are requested to be printed. For the initial design, this produced the following printout of the constraints and responses:

----- DESIGN CONSTRAINTS ON RESPONSES -----								
(MAXIMUM RESPONSE CONSTRAINTS MARKED WITH **)								
-----								
INTERNAL ID	DCONSTR ID	INTERNAL RESPONSE ID	RESPONSE TYPE	L/U FLAG	INTERNAL REGION ID	SUBCASE ID	VALUE	
-----								
13	10	41	FRACCL	UPPER	1086	1	2.8334E-01**	
12	10	40	FRACCL	UPPER	1085	1	2.1704E-01	
11	10	39	FRACCL	UPPER	1084	1	1.5688E-01	
-----								
----- CONSTRAINTS ON DESIGNED PROPERTIES -----								
-----								
INTERNAL ID	PROPERTY ID	PROPERTY NAME	L/U FLAG	CYCLE LIMIT	VALUE			
-----								
33	10	T	UPPER	9.2000E-02	-1.3043E-01			
24	1	T	UPPER	9.2000E-02	-1.3043E-01			
25	2	T	UPPER	9.2000E-02	-1.3043E-01			
0 SUBCASE 1								
-----								
R E S P O N S E S I N D E S I G N M O D E L								
-----								
(N/A - BOUND NOT ACTIVE OR AVAILABLE)								
I N I T I A L A N A L Y S I S S U B C A S E = 1								
-----								
----- FREQUENCY ACCELERATION RESPONSES -----								
-----								
INTERNAL ID	DRESP1 ID	RESPONSE LABEL	GRID ID	COMPONENT NO.	FREQUENCY	LOWER BOUND	VALUE	UPPER BOUND
-----								
39	1084	FA084T3	805	3	2.4600E+01	N/A	2.2286E+03	1.9264E+03
40	1085	FA085T3	805	3	2.4800E+01	N/A	2.3649E+03	1.9432E+03
41	1086	FA086T3	805	3	2.5000E+01	N/A	2.5153E+03	1.9600E+03

The three constraints on responses are constraints 13, 12 and 11 and they have been sorted so that the most critical constraint appears first. The three most critical property responses are 33, 24 and 25 and, in this case, these three constraints are identical. When identical constraints are found, the constraints that are printed are those moved to the top by the sort. Thus, there is no particular order in this case. Since the user has not specified P2RSET, the printed responses are those associated with the printed response constraints. The constraints identify responses 41, 40 and 39, but the responses themselves are printed in ascending order.

By way of contrast, if the P2CR and P2CP had not been used in this case, the print produced by P1=1, P2=15 would contain 13 constraints on responses, 20 constraints on properties, 41 type-1 responses, and one type-2 response. This example is for a small model, but it indicates the significant reduction in print with these new parameters. A side benefit is that it allows for quick determination of critical constraints, avoiding the need to scan through a long list.



## 2.8 Support of External Response in SOL 200

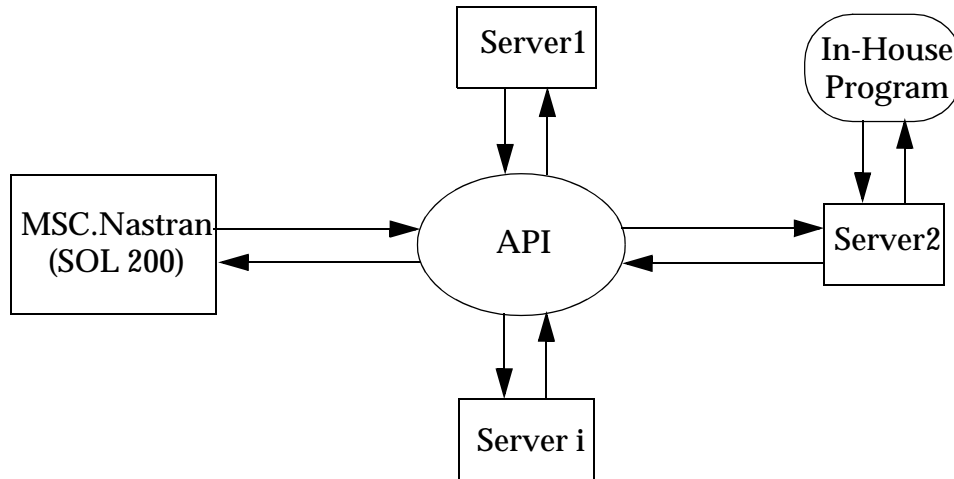
### Introduction

The design optimization capability (SOL 200) in MSC.Nastran has a powerful feature that allows creation a synthetic response using the DRESP2 Bulk Data entry. These synthetic responses, however, were limited to those directly available from MSC.Nastran. The new external response feature further extends the synthetic response by allowing custom defined responses using data supplied in user-supplied programs. Therefore, general and proprietary responses can be used either as an objective or as a constraint in an optimization design.

The external response feature is implemented in SOL 200 with the same client server technology used in supporting the user defined Beam Cross Section Library and P-element geometry evaluator capabilities in MSC.Nastran (Refs. 5., 6.). The design optimization modules in SOL 200 act as the client and user-supplied routines act as the server. Whenever the optimization module requires the value of an external response, it sends the request to the server. Upon receiving the request, the server invokes the server program to calculate the response and then returns the value to the client. The communication between the client and server programs is established through Application Programming Interface (API) routines. [Figure 2-15](#) shows a diagram of the external response capability.

### Benefits

The major benefit of this new capability is to open up the existing powerful design optimization capability in SOL 200 so that custom responses in the design process that were previously either difficult or impossible to formulate may now be included. The server executable is independent from the MSC.Nastran program, so that server programs may be customized easily without changes in MSC.Nastran. For example, it is possible to include a bolt fastener criteria in the full optimization process where the bolt response is created with the combination of design variables or property values and element forces from MSC.Nastran and client proprietary information for the fastener and its spacing. It is generally possible to optimize or constrain an external response created from the user proprietary data that either has direct dependence on the design variables available in SOL 200 or indirect dependence through other quantities available in SOL 200.



**Figure 2-15 Scheme of the External Response Capability**

## Inputs

A new Bulk Data entry, DRESP3 has been added and the DSCREEN entry has been extended to include screening of the DRESP3 constraint data. These will be discussed later in the section.

Since an external response is computed using a user-built server executable, defining an external response requires additional steps to preparing the regular Bulk Data entries. The purpose of these additional steps is to establish the connection between MSC.Nastran and the external response programs. Five steps are required in setting up the external response in a design task:

1. Create a DRESP3 entry to define the response.
2. Write FORTRAN or C routines and build a server executable based on the server template routines.
3. Create a CONNECT entry to define an external response group.
4. Create an evaluator connection file to associate the external response group with the server program.
5. Submit the MSC.Nastran job with the gmconn keyword referencing the evaluator connection file.

The descriptions for these steps are shown in the following sections.

## Step 1. Creating a DRESP3 Entry

A DRESP3 entry defines an external response from user-supplied routines. The response can be used as constraints or as an objective in a design analysis. DRESP3 constraints can be screened using the DSCREEN entry.

**Format:**

1	2	3	4	5	6	7	8	9	10
DRESP3	ID	LABEL	GROUP	TYPE	REGION				
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	etc.						
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	etc.						
	“DRESP1”	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	etc.						
	“DNODE”	G1	C1	G2	C2	G3	C3		
		G4	C4	etc.					
	“DVPREL1”	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	etc.					
	“DVCREL1”	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	“DVMREL1”	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	“DVPREL2	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	‘DCREL2”	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	“DVMREL2”	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	“DRESP2”	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						
	“USRDATA”	Character String Data							
		-etc.-							

Example:

1	2	3	4	5	6	7	8	9	10
DRESP3	1	LBUCK	TAILWNG	BUCK					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						
	USRDATA	Constants: 12345.6 789.0 99.							

Field	Contents
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character)
GROUP	Group name the external response type belongs to (Character). See Remark 2.
TYPE	External response type (Character). See Remark 3.
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	Flag indicating that the labels for the constants in a DTABLE entry follow. (Character)
LABLj	Label for a constant in the DTABLE entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag signifying that the following fields are grid points.
Gm	Grid point identification number. (Integer > 0)

Field	Contents
Cm	Degree-of-freedom number of grid point Gm. ( $1 \leq \text{Integer} \leq 3$ )
DVPREL1	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)
DVCREL1	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
DVMREL1	Flag indicating DVMREL1 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)
DVMPREL2	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
DVCREL2	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
DVMREL2	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number (Integer > 0)
DRESP2	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer > 0)
USRDATA	Flag indicating user input data (Character). See Remark 7.

#### Remarks:

1. DRESP3 entries may reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2 and DRESP2 entries. However, a DRESP3 entry cannot reference another DRESP3 entry.
2. The group name must be referenced by an FMS CONNECT entry.
3. Multiple types of external responses can be defined in one group. Each type name identifies a specific external response evaluated in the user-supplied routines.
4. DRESP3 entries must have unique identification numbers with respect to DRESP2 and DRESP1 entries.

5. The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2”, and “USRDATA” keywords on the continuation entries must appear in the order given above. Any of these words, along with the subsequent data associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.
6. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.
7. The data in the USRDATA field is character string based. It provides a convenient way to pass constants to the external response server routines. The maximum number of characters allowed is 32000.

**Screening DRESP3 Constraints.** The DSCREEN entry can be used if you want to retain the DRESP3 constraints using different screening data from the defaulted settings. For example, the following entry

```
DSCREEN  DRESP3  -0.1
```

will retain all the DRESP3 constraints that are greater than or equal to -0.1.

## Step 2. Build an External Response Server

First, you need to copy server template routines--r3sgrt, r3svald, and r3svals--to the local directory from the MSC.Nastran *install\_dir/msc2004/dr3srv* directory. Routine r3sgrt validates the response type names for an external response group specified on a DRESP3 entry. The NTYPES (the number of response types for a given external response group) and the TYPNAM arrays must be updated for your own applications. Routines r3svald and r3svals compute the desired response and return the value. For the short word machine (a typical workstation), use the r3svald version. For the long word machine, use the r3svals version.

Next, copy two script files, dr3srv and makefile, to the local directory. These two files are required to build external response programs. To build the server program, type ***msc2004 ./dr3srv build***. Correct any FORTRAN compilation errors until the server executable, dr3serv, is built successfully.

### Step 3. Creating a CONNECT Statement

The CONNECT statement, specified in the File Management System Section (FMS), defines the external response group. Multiple CONNECT statements may be used to define multiple groups.

**Format:**

```
CONNECT  DRESP3  Group Name  Evaluator Identifier
```

**Example:**

```
CONNECT  DRESP3  TESTGRP  EXTRESP
```

where DRESP3 is the keyword. The group name matches that specified on the DRESP3 entry and the evaluator identifier is the evaluator name that is referenced in the evaluator connection file (see next step).

### Step 4. Creating an Evaluator Connection File

The evaluator connection file defines the association between the external response group and the corresponding server program.

**Format:**

Evaluator name, the connection option, the path of the server program

**Example:**

where:

```
EXTRESP = ,-, /net/harkness/harkness/users/shz/dr3srv/dr3serv
```

EXTRESP is the evaluator name and the symbol, “-”, indicates that the pipe option is used (i.e., a server executable resides in any net-mounted computer). The path name points to the server executable.

### Step 5. Submit a MSC.Nastran Job with the GMCONN Keyword

The keyword gmconn is used on the command line to reference the name of the evaluator connection file. The connection file is processed at the MSC.Nastran initialization stage to establish a link between the MSC.Nastran program and the server program(s).

# Outputs

The new outputs for the retained DRESP3 responses are similar to those for the retained DRESP2 responses. They are printed in the .f06 file as shown below. The first is the output for the initial and final analyses. The second is the output printed at the end of each design cycle. Both are part of the design optimization output controlled by parameters P1 and P2 on the DOPTPRM entry.

**Listing 2-4 DRESP3 Response Output for the Initial and Final Analysis**

---- RETAINED DRESP3 RESPONSES ----							
INTERNAL ID	DRESP3 ID	RESPONSE LABEL	GROUP NAME	TYPE NAME	LOWER BOUND	VALUE	UPPER BOUND
1	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.4018E+00	1.0000E+00
2	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.4018E+00	1.0000E+00
3	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.4018E+00	1.0000E+00
4	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.4018E+00	1.0000E+00
5	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.4018E+00	1.0000E+00

**Listing 2-5 DRESP3 Response Output After Each Design Cycle**

----- RETAINED DRESP3 RESPONSES -----								
INTERNAL ID	DRESP3 ID	RESPONSE LABEL	GROUP NAME	TYPE NAME	LOWER BOUND	INPUT VALUE	OUTPUT VALUE	UPPER BOUND
1	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.0035E+00	1.0028E+00	1.0000E+00
2	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.0035E+00	1.0028E+00	1.0000E+00
3	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.0035E+00	1.0028E+00	1.0000E+00
4	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.0035E+00	1.0028E+00	1.0000E+00
5	31	LOCBUCK	MYBUCK	BUCKEJ	N/A	1.0035E+00	1.0028E+00	1.0000E+00

## Guidelines and Limitations

- 1. Each group of external responses can have multiple response types.
- 2. Each group of external responses requires one CONNECT entry.
- 3. The external response and external Beam Library can be mixed.
- 4. The number of characters in the USRDATA field on the DRESP3 entry cannot exceed 32000.
- 5. The finite difference approach is the only option to evaluate the gradients of the external responses.

## Improved TIMEOUT Scheme for the DRESP3 Capability

For the applications of the external responses (DRESP3), the API program uses the timeout scheme to ensure a sound connection between MSC.Nastran and the server program. For example, when a server is in the infinite loop, its run time will exceed the pre-specified timeout value. Consequently, the client program will respond accordingly by terminating the job with an appropriate message.



A DRESP3 (or DR3) server has two different modes. An active mode refers to the one in which the DR3 server is computing some response values while the MSC.Nastran client is waiting. An inactive mode is the one in which the server is standing by for the next call from the client. In general, the waiting period within an active session is quite small, while the waiting period for an inactive mode may be much longer; for example, waiting time for a standby server may take several hours for the large scale optimization tasks. Therefore, it may be advantageous to temporarily shut down the DR3 server for the inactive mode and to re-connect it when it is needed. The timeout scheme in MSC.Nastran 2001 is hard-coded, which enhances the existing scheme by performing the following tasks:

1. It provides a new option that allows the user to shut down the DR3 server when it is in the inactive mode and
2. Allows the user to specify timeout values:
  - timeout value for the MSC.Nastran client in an active mode
  - timeout value for the DR3 server in an inactive mode.

Three MSC.Nastran system cells are added for this enhancement.

MSC.Nastran system(382) Disconnects the external response server(s).  
= 0 (default. Keep the connection as in the existing scheme)  
= 1 (disconnect the server(s))

MSC.Nastran system(383) Sets timer for the external response server.  
= 0 (default. Use default timeout value: 100,000)  
> 0 (new timeout value)

MSC.Nastran system(384) Sets timer for the client (MSC.Nastran) communication with the DR3 server.  
= 0 (default. Use default timeout value: 10,000)  
> 0 (new timeout value)

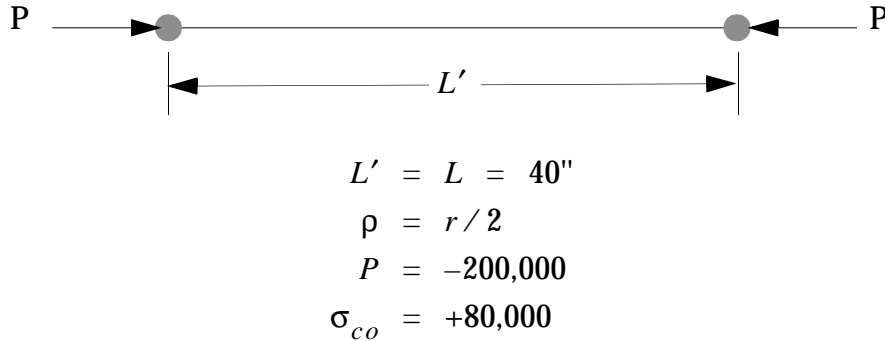
## Example

### External Response to Include Alternative Buckling Response

This example demonstrates how the DRESP3 Bulk Data entry can be applied to include a buckling criteria that is not available directly from MSC.Nastran. The DRESP3 Bulk Data entry is applied to a bar loaded longitudinally as shown in [Figure 2-16](#).

### Model Description

This particular design example is so simple that it may be solved using hand calculations. However, the contribution of this example is not in the complexity of the problem solved but in its illustration of the process that must be followed to apply the external response capability.



**Figure 2-16 Pinned-Pinned Column Buckling**

### Theory

The design task is to find the area of the rod that minimizes the weight while satisfying constraints on the longitudinal stress and on the buckling behavior. The stress constraint is

$$-80,000 \leq \sigma \leq 100,000$$

while the critical buckling stress is computed using a combination of Johnson and Euler buckling criteria, with the selection of the criterion to use based on the slenderness ratio of the rod (see Peery and Azar in [“References”](#) on page 406 of the *MSC.Nastran Design Sensitivity and Optimization User's Guide*).

$$\sigma_{cr} = \sigma_{co} \left[ 1 - \frac{\sigma_{co} (L' / \rho)^2}{4\pi^2 E} \right] \quad L' / \rho \leq \pi \sqrt{\frac{2E}{\sigma_{co}}} \quad \text{Eq. 2-29}$$

$$\sigma_{cr} = \frac{\pi^2 E}{(L' / \rho)^2} \quad L' / \rho \geq \pi \sqrt{\frac{2E}{\sigma_{co}}} \quad \text{Eq. 2-30}$$

where:

$\sigma_{cr}$  = critical buckling stress

$L'$  = effective length

$\rho$  = radius of gyration

$\sigma_{co}$  = empirically determined column yield stress

The fact that the failure criteria can branch based on a combination of structural properties and responses makes it an ideal application for the external response capability.

## Required Inputs

### Step 1 - Creating the DRESP3 entries

**Listing 2-6** shows the input file for this example. For the two subcases, the first subcase performs a static analysis and imposes constraints on the stress in the element and on the combined buckling failure criteria. The second subcase performs a buckling analysis and places a lower bound of 1.0 on the first buckling eigenvalue. This is redundant with the Euler buckling condition of the first subcase and therefore, provides a check on the external response calculation.

#### Listing 2-6 Input Data File for the DRESP3 Example (DSOUG13)

```
CONNECT DRESP3 TESTGRP EXTRESP
DIAG 8,15 $
SOL 200
CEND
TITLE      = BUCKLING TEST CASE WITH AN EXTERNAL RESPONSE - DSOUG13
SUBTITLE   = JOHNSON/EULER BUCKLING CASE
ECHO       = SORT
SPC        = 100
DESOBJ     = 20
SUBCASE 1
  DESSUB    = 1
  LABEL     = LOAD CONDITION 1
  LOAD      = 300
  ANALYSIS  = STATICS
  DISP      = ALL
  STRESS    = ALL
SUBCASE 2
  DESSUB    = 2
```

```

      ANALYSIS = BUCK
      METHOD    = 1
      DISP     = ALL
      LABEL    = BUCKLING FACTORS

$
BEGIN BULK
PARAM,POST,-1
$-----
$ ANALYSIS MODEL
$-----
$
$ GRID DATA
$      2      3      4      5      6      7      8      9      10
GRDSET
GRID 1      0.0  0.0  0.0
GRID 2      8.0  0.0  0.0
GRID 3     16.0  0.0  0.0
GRID 4     24.0  0.0  0.0
GRID 5     32.0  0.0  0.0
GRID 6     40.0  0.0  0.0
GRID 10     0.0  0.0 100.0      123456
$
$ ELEMENT AND MATERIAL DATA
CBAR 1      10      1      2      10
CBAR 2      10      2      3      10
CBAR 3      10      3      4      10
CBAR 4      10      4      5      10
CBAR 5      10      5      6      10
MAT1 1      3.0E7      0.33      0.1
$ PROPERTY DATA
PBARL 10      1      MSCBML0  ROD
      1.0
$ BOUNDARY CONDITION DATA
SPC1 100      1      1
SPC1 100      2      1      6
$ EXTERNAL LOADS DATA
FORCE 300      6      -2.0E5  1.0
$ BUCKLING ANALYSIS DATA
EIGRL 1      .05      4
$
$-----
$ DESIGN MODEL
$-----
$
DESVAR 1      RG      1.0      0.01  10.0
DOPTPRM P2      15      DESMAX 20      DELP      0.5      GMAX      0.01
      CONVDV 0.01  CONVPR 0.02  P1      1
DRESP1 20      W      WEIGHT
DRESP1 23      S1      STRESS  PBAR      7      10
DRESP1 24      S1      STRESS  PBAR      8      10
DRESP1 25      S1      STRESS  PBAR      6      10
DRESP3 32      JOHNSON TESTGRP EULJOH
      DESVAR 1
      DTABLE  L      E      SIGMAC

```

```

      DRESP1    25
$
DCONSTR    1    23              100000.
DCONSTR    1    24             -80000.
DCONSTR    1    32              1.0
DTABLE     E   30.0E6    L      40.0    SIGMAC    8.0E4
DVPREL1    10    PBARL    10      DIM1
           1    1.0
$
$  DESIGN FOR BUCKLING EIGENVALUE
DRESP1     1          BUCK1          LAMA          1
DCONSTR     2          1          1.0
PARAM      DSNOKD    1.0
ENDDATA

```

The listing shows the DRESP3 entry that provides the buckling response. The input includes the design variable value, constants that define  $E$ , the rod length,  $\sigma_{co}$ , and the axial stress in the rod. TESTGRP is specified on the DRESP3 entry as the external response GROUP and EULJOH as the TYPE.

## Step 2 - Building the External Response Server

Two subroutines perform the server task. The first subroutine (R35GRT) shown in [Listing 2-7](#). R3GRT specifies the types of responses available from the server. This is used to ensure that the user input TYPE is supported by the server. In this case, EULJOH is the only unsupported response. The second subroutine, R3VALD (shown in [Listing 2-9](#)), performs the response evaluation based on the user input arguments. (Note that the D suffix on this subroutine name is used when running on a short word; i.e., 32 bit machine and the subroutine should be named R3VALS when a long word machine is used). In this case, there are four real arguments being passed and in the order given on the DRESP3 entry. The output of the subroutine is the DR3VAL argument and is calculated using the formulas given in [Eq. 2-29](#) and [Eq. 2-30](#).

### Listing 2-7 The R3GRT Subroutines

```

SUBROUTINE R3SGRT(GRPID,TYPNAM,ERROR)
C -----
C
C      PURPOSE:  VERIFY THE EXTERNAL RESPONSE TYPE
C
C      GRPNAM:  INPUT INTEGER          - GROUP ID
C      TYPNAM   INPUT CHARACTER*8      - NAME OF EXTERNAL RESPONSE TYPE
C      ERROR    INPUT/OUTPUT INTEGER  -ERROR CODE FOR THE CALL.
C                                          > 0 = PROCESSING ERROR
C
C      METHOD
C      MATCH THE USER INPUT:  TYPNAM WITH THE LIST OF AVAILABLE
C      EXTERNAL RESPONSE TYPES. IF NO MATCH IS FOUND, SET ERROR CODE.
C

```

```

C   CALLED BY
C       R3CGRT
C
C   NOTE:
C       THE WRITER OF THIS ROUTINE IS RESPONSIBLE FOR SPECIFYING
C       NTYPES AND R3TYPE.
C -----
C
C   VARIABLES PASSED IN
C
C       INTEGER GRPID, ERROR
C       CHARACTER*8 TYPNAM
C
C   LOCAL VARIABLES
C
C       INTEGER NTYPES, BADTYP
C       PARAMETER(NTYPES=1)
C       CHARACTER*8 R3TYPE(NTYPES)
C
C       DATA BADTYP/7554/
C       DATA R3TYPE/'EULJOH  '/
C
C       ERROR = 0
C       DO 100 ITYPE = 1, NTYPES
C           IF (TYPNAM .EQ. R3TYPE(ITYPE)) THEN
C               GOTO 200
C           END IF
100    CONTINUE
C       ERROR = BADTYP
200    CONTINUE
C
C       RETURN
C       END

```

### Listing 2-8 The R3VALD Subroutines

```

SUBROUTINE R3SVALD(GRPID,TYPNAM,
.               NITEMS,ARGLIS,
.               NSIZE, ARGVAL,
.               NWRDA8,ARGCHR,
.               DR3VAL,ERROR)
C -----
C
C   PURPOSE: COMPUTE THE EXTERNAL RESPONSE
C
C   GRPID   INPUT INTEGER           - GROUP ID
C   TYPNAM  INPUT CHARACTER*8       - NAME OF EXTERNAL RESPONSE TYPE
C   NITEMS  INPUT INTEGER           - DIMENSION OF ARRAY ARGLIS
C   NSIZE   INPUT INTEGER           - DIMENSION OF ARRAY ARGVAL
C   NWRDA8  INPUT INTEGER           - DIMENSION OF CHARACTER ARRAY ARGCHR
C   ARGLIS  INPUT INTEGER           - ARRAY OF NO. OF ITEMS FOR EACH ARGUMENT
C   ARGVAL  INPUT DOUBLE            - ARRAY OF ALL ARGUMENT VALUES
C   ARGCHR  INPUT CHARACTER*8       - ARRAY OF CHARACTERS
C   DR3VAL  OUTPUT DOUBLE           - VALUE OF THE EXTERNAL RESPONSE
C   ERROR   INPUT/OUTPUT INTEGER    -ERROR CODE FOR THE CALL.

```

```
C                                0 = PRINT ERROR MESSAGES
C                                1 = DO NOT PRINT ERROR MESSAGES.
C
C  METHOD
C    A)SET UP VARIOUS PARAMETERS FROM THE ARGUMENT LIST
C    B)EVALUATE THE EXTERNAL RESPONSE BASED ON THE GIVEN TYPNAM
C    C)RETURN BADTYP ERROR IF TYPNAM IS NOT MATCHED HERE.
C
C  CALLED BY
C    SENDR3SVALD
C  -----
C
C  VARIABLES PASSED IN
C
C    CHARACTER*8 TYPNAM, ARGCHR(NWRDA8)
C    INTEGER GRPID, NITEMS, NSIZE, ARGLIS(NITEMS), ERROR, NWRDA8
C    DOUBLE PRECISION ARGVAL(NSIZE), DR3VAL
C
C  LOCAL VARIABLES
C
C    INTEGER BADTYP
C    DOUBLE PRECISION PI,  FAC, FACT, SLNDER
C    DOUBLE PRECISION R,L,E,SIGMA,SIGMAC, RGYRA
C
C    DATA BADTYP /7554/
C
C    THE USER-SUPPLIED EQUATION TO DEFINE THE EXTERNAL RESPONSE
C    SIGMA = DRESP1, R=DESVAR, L, E AND SIGMAC = DTABLE CONSTANTS
C
C    EULER   : EULER= -SIGMA * (L/ RGYRA ) **2 / (PI**2 * E)
C              RGYRA = R / 2.0
C
C    JOHNSON: JOHNSON = -SIGMA / (SIGMAC * FACTOR )
C              FACTOR = 1. - SIGMAC * (L/RGYRA)**2 /(4 * PI**2 * E)
C    ERROR = 0
C
C  SET UP PARAMETERS FOR VARIOUS ARGUMENT ITEMS
C
C    PI = 3.14159
C    PI2 = PI * PI
C    IF (TYPNAM .EQ. 'EULJOH ') THEN
C      R      = ARGVAL(1)
C      L      = ARGVAL(2)
C      E      = ARGVAL(3)
C      SIGMAC = ARGVAL(4)
C      SIGMA  = ARGVAL(5)
C      RGYRA  = R / 2.0
C      SLNDER = L / RGYRA
C      FACT   = PI * SQRT(2.0D0 * E / SIGMAC)
C      IF ( SLNDER .LE.FACT) THEN
C        JOHNSON CRITERION
C        FAC   = 1.0D0 - SIGMAC * (SLNDER) ** 2 /(4.0D0 * PI2 * E )
C        DR3VAL = -SIGMA / (SIGMAC * FAC)
C      ELSE
C        EULER CRITERION
C        DR3VAL = -SIGMA * SLNDER**2 / (PI2 * E)
C    ELSE
C      ERROR = BADTYP
```

```

END IF

RETURN
END

```

### Step 3 - CONNECT Statement

A CONNECT file management statement specifies a DRESP3 application, TESTGRP as the group, and EXTRESP as the external evaluator.

### Step 4

A final file 'eval connect' indicates where the server program resides. It has the form:

```
EXTRESP,-,/usr/ehj/dsoug/dr3srv/dr3serv
```

where EXTRESP is the evaluator name given in the CONNECT File Management statement and the '-' connect option in this case indicates the server exists on any network mounted computer. This is followed by the name of the file that contains the server.

### Step 5 - Submitting a MSC.Nastran job.

Now that all the files have been assembled, the job is submitted using the GMCONN option; for example:

```
nastran dsoug13 scr=yes gmconn=evalconnect
```

Selected results for this simple design task are presented in [Listing 2-9](#). The printout first gives results from the final analysis. The single design variable specified by the user as the radius of the ROD input on the PBARL has generated nine designed properties for the bar cross section. The "RESPONSE IN THE DESIGN MODEL" output indicates that the stresses in the bars are well below the compression allowable and that the final global buckling response is 1.0863. This value is above the 1.0 limit that has been placed on this response. Therefore, we can infer that the Johnson criterion is the critical design constraint. This is confirmed by the fact that all of the retained DRESP3 responses are at their upper bound limit. The design history summary indicates that the rod radius must increase from 1.0 to 1.1072 to satisfy this design requirement and that this was accomplished in two design cycles.



## Listing 2-9 Final Results in the Buckling Design Task (DSOUG13)

```
*****
*
*      F I N A L   A N A L Y S I S
*
*****
```

```
***** ANALYSIS RESULTS BASED ON THE FINAL DESIGN *****
```

```
----- DESIGN OBJECTIVE -----
```

INTERNAL RESPONSE ID	TYPE OF RESPONSE	LABEL	MINIMIZE OR MAXIMIZE	SUPERELEMENT ID	SUBCASE ID	VALUE
1	DRESP1	WEIGHT	MINIMIZE	0	0	1.5405E+01

```
----- DESIGN VARIABLES -----
```

INTERNAL ID	DESVAR ID	LABEL	LOWER BOUND	VALUE	UPPER BOUND
1	1	RG	1.0000E-02	1.1072E+00	1.0000E+01

```
----- DESIGNED PROPERTIES -----
```

PROPERTY TYPE	PROPERTY ID	PROPERTY NAME	TYPE OF PROPERTY	LOWER BOUND	VALUE	UPPER BOUND
PBARL	10	DIM1	DVPREL1	1.0000E-15	1.1072E+00	1.0000E+20
PBAR	10	C1	DVPREL1	-1.0000E+35	1.1072E+00	1.0000E+20
PBAR	10	D2	DVPREL1	-1.0000E+35	1.1072E+00	1.0000E+20
PBAR	10	E1	DVPREL1	-1.0000E+35	-1.1072E+00	1.0000E+20
PBAR	10	F2	DVPREL1	-1.0000E+35	-1.1072E+00	1.0000E+20
PBAR	10	A	SECPRO	1.0000E-15	3.8512E+00	1.0000E+20
PBAR	10	I1	SECPRO	1.0000E-15	1.1803E+00	1.0000E+20
PBAR	10	I2	SECPRO	1.0000E-15	1.1803E+00	1.0000E+20
PBAR	10	J	SECPRO	1.0000E-15	2.3605E+00	1.0000E+20

```
----- DESIGN CONSTRAINTS ON RESPONSES -----
```

```
(MAXIMUM RESPONSE CONSTRAINTS MARKED WITH **)
```

INTERNAL ID	DCONSTR ID	INTERNAL RESPONSE ID	RESPONSE TYPE	L/U FLAG	INTERNAL REGION ID	SUBCASE ID	VALUE
1	1	2	STRESS	LOWER	1	1	-3.5085E-01
2	1	3	STRESS	LOWER	1	1	-3.5085E-01
3	1	4	STRESS	LOWER	1	1	-3.5085E-01
4	1	5	STRESS	LOWER	1	1	-3.5085E-01
5	1	6	STRESS	LOWER	1	1	-3.5085E-01
6	1	1	EXTERNAL	UPPER	32	1	2.7899E-03**
7	1	2	EXTERNAL	UPPER	32	1	2.7899E-03**
8	1	3	EXTERNAL	UPPER	32	1	2.7899E-03**
9	1	4	EXTERNAL	UPPER	32	1	2.7899E-03**
10	1	5	EXTERNAL	UPPER	32	1	2.7899E-03**
11	2	12	LAMA	LOWER	0	2	-8.6330E-02

-----  
R E S P O N S E S   I N   D E S I G N   M O D E L

(N/A - BOUND NOT ACTIVE OR AVAILABLE)

----- WEIGHT RESPONSE -----

INTERNAL ID	DRESP1 ID	RESPONSE LABEL	ROW ID	COLUMN ID	LOWER BOUND	VALUE	UPPER BOUND
1	20	W	3	3	N/A	1.5405E+01	N/A

F I N A L   A N A L Y S I S   S U B C A S E =   1

----- STRESS RESPONSES -----

INTERNAL ID	DRESP1 ID	RESPONSE LABEL	ELEMENT ID	VIEW ELM ID	COMPONENT NO.	LOWER BOUND	VALUE	UPPER BOUND
2	24	S1	1		8	-8.0000E+04	-5.1932E+04	N/A
3	24	S1	2		8	-8.0000E+04	-5.1932E+04	N/A
4	24	S1	3		8	-8.0000E+04	-5.1932E+04	N/A
5	24	S1	4		8	-8.0000E+04	-5.1932E+04	N/A
6	24	S1	5		8	-8.0000E+04	-5.1932E+04	N/A
7	25	S1	1		6	N/A	-5.1932E+04	N/A
8	25	S1	2		6	N/A	-5.1932E+04	N/A
9	25	S1	3		6	N/A	-5.1932E+04	N/A
10	25	S1	4		6	N/A	-5.1932E+04	N/A
11	25	S1	5		6	N/A	-5.1932E+04	N/A

F I N A L   A N A L Y S I S   S U B C A S E =   2

----- BUCKLING LOAD RESPONSES -----

INTERNAL ID	DRESP1 ID	RESPONSE LABEL	MODE NO.	LOWER BOUND	VALUE	UPPER BOUND
12	1	BUCK1	1	1.0000E+00	1.0863E+00	N/A

---- RETAINED DRESP3 RESPONSES ----

INTERNAL ID	DRESP3 ID	RESPONSE LABEL	GROUP NAME	TYPE NAME	LOWER BOUND	VALUE	UPPER BOUND
1	32	JOHNSON	TESTGRP	EULJOH	N/A	1.0028E+00	1.0000E+00
2	32	JOHNSON	TESTGRP	EULJOH	N/A	1.0028E+00	1.0000E+00
3	32	JOHNSON	TESTGRP	EULJOH	N/A	1.0028E+00	1.0000E+00
4	32	JOHNSON	TESTGRP	EULJOH	N/A	1.0028E+00	1.0000E+00
5	32	JOHNSON	TESTGRP	EULJOH	N/A	1.0028E+00	1.0000E+00

SUBCASE 2

\*\*\*\*\*  
SUMMARY OF DESIGN CYCLE HISTORY  
\*\*\*\*\*

(HARD CONVERGENCE ACHIEVED)

(SOFT CONVERGENCE ACHIEVED)

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED 3  
NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS 2

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		1.256637E+01		4.017631E-01
1	1.539844E+01	1.539775E+01	4.471774E-05	3.486395E-03
2	1.540468E+01	1.540468E+01	1.238162E-07	2.789855E-03

0

SUBCASE 2

DESIGN VARIABLE HISTORY

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL	1	2	3	4	5
1	1	RG	1.0000E+00	1.1069E+00	1.1072E+00			

\*\*\* USER INFORMATION MESSAGE 6464 (DOM12E)  
RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 2.

## 2.9 Improved BEAM Processing

The internal handling in the design of the PBEAML and PBEAM has been streamlined to provide more efficient and robust design properties. No changes are required on the part of the user to benefit from these changes.

For the design of a constant section PBEAML or PBEAM, provide design model data to vary END A dimensions or properties. Prior to 2004, MSC.Nastran perturbed properties and dimensions for both end A and end B separately. As the design proceeded, end A and end B properties were linked together to maintain the constant section. This existing process unnecessarily increased the number of pseudo-loads, vectors in the design sensitivity analysis, and the number of property constraints in the optimizer.

The improved scheme recognizes that the properties for end A and end B will always be the same for a constant section beam. End A and B properties are therefore, perturbed simultaneously.

---

**Note:** Only end A properties are essential in forming the stiffness and mass matrix for a constant section beam. The simultaneous perturbation mainly benefits stress data recovery with correct stress data recovery point coordinates for the intermediate sections and end B.

---

The improved BEAM processing also covers the tapered beam case where PBEAM or PBEAML may have up to 9 intermediate sections and an end B. In versions prior to MSC.Nastran 2004, if end B DIMx of PBEAML or end B properties were designed, the last intermediate section that had the same property data was separately perturbed. The improved BEAM processing scheme considers the end B and the last intermediate section as inseparable and will perturb both if end B properties are designed.

The improved BEAM processing also addresses the case where END B or other intermediate stations are defined only for data recovery purposes and do not provide dimensions or properties for stations other than END A. For prior versions, this scenario was treated incorrectly in design optimization as a tapered beam, whereas it should have been treated as a constant section. For example, both of the following PBEAML cases are considered as a constant section in SOL 200 for MSC.Nastran 2004, but only the first was considered as a constant section in MSC.Nastran 2001.

Constant section PBEAML cases,

Case 1

1	2	3	4	5	6	7	8	9	10
PBEAML	2	10	BMLSERV	TUBE					
	10.5	8.6							

Case 2

1	2	3	4	5	6	7	8	9	10
PBEAML	2	10	BMLSERV	TUBE					
	10.5	8.6		YES	1.0				

Similarly, the following three cases of PBEAM are considered as constant section PBEAM. In the new scheme, properties for all sections will be perturbed simultaneously if end A properties are designed.

Constant section PBEAM cases,

Case 1

1	2	3	4	5	6	7	8	9	10
PBEAM	200	1	100.	1.+4	1.+4				+PBEAM1
+PBEAM1	2.0	1.0							

Case 2

1	2	3	4	5	6	7	8	9	10
PBEAM	201	1	100.	1.+4	1.+4				
	2.0	1.0							
	YESA	1.0							

Case 3

1	2	3	4	5	6	7	8	9	10
PBEAM	203	1	100.	1.+4	1.+4				
	2.0	1.0							

1	2	3	4	5	6	7	8	9	10
	YESA	0..5							
	YESA	1.0							

Because the improved BEAM processing scheme is considered as equivalent to current logic in handling constant section PBEAM/PBEAML and designing end B for a tapered beam, no significant difference in sensitivity and optimization results are expected. However, performance improvements should be observed as a result of the decrease in the number of perturbed properties. Accuracy enhancements may be an added benefit that results from the removal of redundant constraints on beam property values.

## 2.10 Frequency-Dependent Constraints

### Introduction

The specification of upper and lower bounds on responses has been extended so that the bounds can be specified using a TABLEDi Bulk Data entry.

### Benefits

Design requirements in frequency response applications, particularly NVH, often vary as a function of frequency. These limits can now be specified using the same TABLEDi format that is currently used for frequency or time-dependent loading. This greatly simplifies a task previously required individual specification of a limit at each frequency.

### Input

New options have been provided for the input response limits on the DCONSTR entry. See “[Statements, Commands, Entries, and Parameters](#)” on page 621 for the DCONSTR Bulk Data description. The allowable specifications input in fields 4 and 5 of this entry now allow integer or real number input. If the input is a real number, then it specifies the lower bound for field 4 and upper bound for field 5. If the input is an integer, it identifies a TABLEDi entry that provides the limits using a frequency-dependent table.

### Outputs

No changes have been made to outputs.

### Guidelines and Limitations

Frequency-dependent constraints are only applicable to DRESP1 response types that can be frequency-dependent; that is, those with RTYPE = FRxxxx or PDSxxxx or DRESP2 or DRESP3 responses that are frequency-dependent. If a frequency-dependent constraint is applied to any other response type, including transient responses, it produces a User Fatal Error.

### Example

This technique has been applied to an abridged example of input from a client (the actual example cannot be made available here) shows the use of the new feature.

FREQ1	4	6.0	0.1	60					
DRESP1	1133	ACC1033	PSDACCL	620	3	1033			
DCONSTR	101	1133		1133					
TABLED1	1133								
	0.0	0.0	6.0	1.0e-3	7.0	1.7e-3	8.0	1.7e-3	
	12.0	2.0e-4	endt						

Results/Discussion

Figure 2-17 shows a target imposed by the DCONSTR of the PSDACCL response and the initial and final PSD acceleration responses. The initial design violated the limit significantly while the final design is at or below the target at all frequencies.

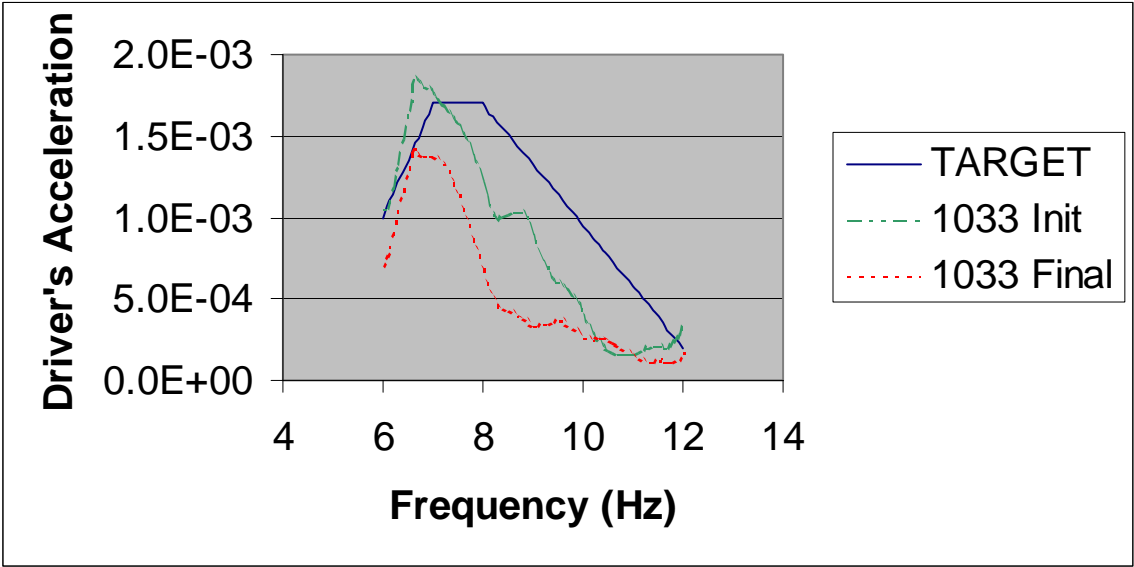


Figure 2-17 Initial and Final PSD Acceleration Plots versus a Prescribed Target.



## 2.11 Generation of a Complete Updated Bulk Data File in SOL 200 File

### Introduction

MSC.Nastran has had a longstanding capability that provides updated bulk data entries in the course of design optimization controlled by PARAMs, DESPCH, and DESPCH1. MSC.Nastran 2004 expands this capability in a major way by providing a complete new bulk data file at the end of a successful SOL 200 design task that retains the bulk data entries and the comments in the same order as in the original input file.

### Benefits

The availability of a complete, updated bulk data file significantly facilitates either continuing the SOL 200 design process after making adjustments in the design task or when analyzing the updated properties to a different analysis, such as nonlinear. Previously, the modified entries needed to be “cut and pasted” into the original bulk data file, which could be particularly troublesome if the design data were scattered throughout the original file. Also, the commented lines typically provide valuable information explaining the input, so retaining these comments captures this knowledge.

### Input

The new bulk data file is output to the .pch file based on a new option in the ECHO Case Control command. See “[Statements, Commands, Entries, and Parameters](#)” on page 621 for the revised command. Previously, the PUNCH describer served the function of providing a sorted copy of the input bulk data in the .pch file with the comment lines appearing at the front of the file. With this enhancement, the PUNCH describer has three suboptions:

1. **SORT** -- This is the default and provides the previous capability
2. **NEWBULK** -- This provides a new bulk data file in the .pch file following a SOL 200 redesign and does not provide the original sorted bulk data file.
3. **BOTH** -- Both the **SORT** and **NEWBULK** suboptions are processed.

### Outputs

The new output in the .pch file is in addition to any results produced by other PUNCH options in case control and any design data from PARAMs DESPCH and DESPCH1.

## Guidelines and Limitations

For shape optimization, it is not particularly useful to update the GRID and the DESVAR entries because the shape variation is based on the change in the design variable. Nonetheless, a simultaneous update will occur and the user needs to be aware of what this means.

The new bulk data file is produced only at the end of the design and only if the resizing has been successful. Unlike the DESPCH and DESPCH1 parameters, results from intermediate design cycles are not available.

A single bulk data file will be produced that contains all *'include'* files. Multiple bulk data files, such as those used for shape optimization, will be included.

The ECHOON/ECHOOFF features of the ECHO command are supported. If the user inadvertently places an ECHOOFF Bulk Data entry in the bulk data input file, the subsequent entries will not be stored and these entries will not be updated. On the other hand, if the user has a very large bulk data input file and only wants to update a small portion, the ECHOOFF/ECHOON features can be used to limit the size of the .pch file.

## Example - newbulk2.dat

**Listing 2-10** shows an abridged input file that demonstrates this new feature. The

***ECHO = punch(newbulk),sort(dvprel1,desvar)***

command asks that a new bulk data file be created in the .pch file. The sort request is a standard request for sorted echo of the DVPREL1 and DESVAR Bulk Data entries. The PARAM,DESPCH,-1 parameter at the end of the file requests that updated design data never be punched separately.

**Listing 2-11** contains the same fragments of the resulting .pch file. Note that the unchanged entries, including comments, appear exactly as in the original file with the minor exception that everything is now in uppercase. The listing shows the new PBEAML and DESVAR entries and these are punched using the large field format. If short field results are preferred, PARAM DESPCH would have to be set to 0 or a positive integer and DESPCH1 would have to be set to a negative integer.

### Listing 2-10 Fragments of the NEWBULK2.dat Input File

```
CONNECT BEAMEVAL BMLSERV MSCBML
$ Linear Static Analysis, Database
ID MSC, BMLOMNI1 $ ehj/grs v71 29-jan-1999
TIME 200
SOL 200
CEND
```

```

title = beam library test case
subtitle = every dimension of 19 sections is designed
ECHO = punch(newbulk),sort(dvprell,desvar)
SPC = 6
DESOBJ = 20
dessub = 1
analysis=statics
SUBCASE 1
$ Load Case Description :
  SUBTITLE=transverse tip load
  LOAD = 8
  DISPLACEMENT = ALL
  force = all
  SPCFORCE = ALL
  STRESS( CORNER, SHEAR ) = ALL
SUBCASE 2
$ Load Case Description :
  SUBTITLE=lateral tip load
  LOAD = 10
  DISPLACEMENT = ALL
  force = all
  SPCFORCE = ALL
  STRESS( CORNER, SHEAR ) = ALL
BEGIN BULK
PARAM      AUTOSPC YES
PARAM      INREL    0
PARAM      ALTRED   NO
PARAM      COUPMASS 1
PARAM      K6ROT    0.
PARAM      WTMASS   1.
$ Elements and Element Properties for region : prop3
CBEAM      3      3      3      103      0.      1.      0.
$ Elements and Element Properties for region : prop4
CBEAM      4      4      4      104      0.      1.      0.
$ Elements and Element Properties for region : prop5
CBEAM      5      5      5      105      0.      1.      0.
...
PBEAML     7      10      BMLSERV L
+          12.3    14.5    1.0    0.5
PBEAML     8      10      BMLSERV CHAN
+          5.6     15.8    0.5     0.6     0.7
DESVAR     1      rod      1.00  0.01  10.0
DESVAR     2      tubeor   1.0   0.01  10.0
DESVAR     3      tubet    1.0   0.10  10.0
DESVAR     4      bar1     1.0   0.01  10.0
DESVAR     5      bar2     1.00  0.01  10.0
DESVAR     6      box1     1.00  0.01  10.0
...
DRESP1    20      W      WEIGHT
DRESP1    23      S1     STRESS  PBEAML      4      1
          2      3      4      5      6      8
...
DCONSTR    1      23     -10000.    10000.
DCONSTR    1      24     -10000.    10000.

```

```

DCONSTR          1      25      -10000.      10000.
DCONSTR          1      26      -10000.      10000.
doptprm          p1      1      p2      15      desmax      3      delp      0.5
param            despch-1
ENDDATA

```

### Listing 2-11 Fragments of the NEWBULK2.pch File

```

$
$ *****
$ *   ENTIRE NEW BULK DATA DECK WITH UPDATED ENTRIES INSERTED *
$ *****
$
BEGIN BULK
PARAM      AUTOSPC YES
PARAM      INREL   0
PARAM      ALTRED  NO
PARAM      COUPMASS 1
PARAM      K6ROT   0.
PARAM      WTMASS  1.
$ ELEMENTS AND ELEMENT PROPERTIES FOR REGION : PROP3
CBEAM      3      3      3      103      0.      1.      0.
$ ELEMENTS AND ELEMENT PROPERTIES FOR REGION : PROP4
CBEAM      4      4      4      104      0.      1.      0.
$ ELEMENTS AND ELEMENT PROPERTIES FOR REGION : PROP5
...
PBEAML*          7      10      BMLSERV      L      *
*
*      1.78429282E+00  1.78429282E+00  3.30756336E-01  3.30756336E-01*
*      0.00000000E+00
PBEAML*          8      10      BMLSERV      CHAN      *
*
*      1.68532622E+00  1.82908583E+00  1.66459978E-01  9.20057893E-02*
*      6.99999988E-01
DESVAR *          1ROD      7.04097748E-01  9.99999978E-03+D      1V
*D      1V  1.00000000E+01  1.00000000E+00
DESVAR *          2TUBEOR      8.70542109E-01  9.99999978E-03+D      2V
*D      2V  1.00000000E+01  1.00000000E+00
DESVAR *          3TUBET      3.39091629E-01  1.00000001E-01+D      3V
*D      3V  1.00000000E+01  1.00000000E+00
DESVAR *          4BAR1      7.05203831E-01  9.99999978E-03+D      4V
*D      4V  1.00000000E+01  1.00000000E+00
DESVAR *          5BAR2      7.05203831E-01  9.99999978E-03+D      5V
*D      5V  1.00000000E+01  1.00000000E+00
DESVAR *          6BOX1      9.49697018E-01  9.99999978E-03+D      6V
*D      6V  1.00000000E+01  1.00000000E+00
...
DRESP1  20      W      WEIGHT
DRESP1  23      S1      STRESS  PBEAML      4      1
      2      3      4      5      6      8
...
DCONSTR  1      23      -10000. 10000.
DCONSTR  1      24      -10000. 10000.
DCONSTR  1      25      -10000. 10000.
DCONSTR  1      26      -10000. 10000.
DOPTPRM  P1      1      P2      15      DESMAX  3      DELP      0.5
PARAM    DESPCH  -1
ENDDATA

```

## 2.12 Responses Spanning Across Subcases or Superelements

### Introduction

A longstanding restriction requiring that DRESP1s referenced on a DRESP2 or DRESP3 entry must belong to the same subcase or the same superelement has been removed.

### Benefits

Users have frequently asked for the capability to synthesize responses from different subcases and, particularly, different analysis types. This capability has now been incorporated into MSC.Nastran 2004. At the same time, a capability to include responses from different superelements has also been provided.

### Input

The spanning of superelements requires no additional input if the condensed Case Control is used. MSC.Nastran automatically detects that the invoked responses belong to separate superelements and sets up the bookkeeping to handle this processing.

The spanning of subcases requires special Case Control commands. The DRESP2 or DRESP3 response that invokes DRESP1s that span subcases must, in turn, be invoked by a DESOBJ or DESGLB Case Control command that appears above the first subcase. Inside each subcase, a new DRSPAN Case Control command (see “**New or Modified Case Control Commands**” in Appendix A.2) is required to identify a set of responses from the current subcase that are invoked on the DRESP2. The DRSPAN=n command refers to a SET Case Control command that lists the IDs of the DRESP1s that are to be included in the DRESP2s.

### Outputs

There is no change in the outputs.

### Example - rspan.dat

This example is a variation on the familiar 3-bar truss example that illustrates the subcase spanning capability. [Listing 2-12](#) is a fragment of the input data file that shows the use of this feature. The DESGLB=56 entry points to a DCONSTR entry that in turn imposes limits on DRESP2 response ID 50. This DRESP2 averages the stresses from DRESP1s 23 and 25. DRESP1 23 is referenced in SET 523 which is, in turn, called

out by a DRSPAN request in subcase 5. Similarly, DRESP1 25 is referenced in SET 625, which is called by a DRSPAN request in subcase 6. A DSAPRT(END=SENS) command causes the job to complete following the initial sensitivity analysis and the formatted sensitivity results are printed.

### Listing 2-12 Fragment of Input File rspan.dat

```

SOL 200          $  OPTIMIZATION
CEND
TITLE = DEMONSTRATION OF RESPONSES SPANNING SUBCASES
SUBTITLE = BASELINE - 2 CROSS SECTIONAL AREAS AS DESIGN VARIABLES
ECHO = SORT
SPC = 100
DISP = ALL
STRESS = ALL
ANALYSIS = STATICS
DESOBJ = 20
DSAPRT(END=SENS)=ALL
DESGLB = 56
SET 523 = 23
SET 625 = 25
DESSUB = 5
SUBCASE 5
    DRSPAN = 523
    LABEL = LOAD CONDITION 1
    LOAD = 300
SUBCASE 6
    DRSPAN = 625
    LABEL = LOAD CONDITION 2
    LOAD = 310
    ANALYSIS = STATICS
BEGIN BULK
CROD    1      11      1      4
CROD    2      12      2      4
CROD    3      13      3      4
DCONSTR 5      21      -0.20  0.20
$
DCONSTR 56     50      -15000. 20000.
$
DESVAR  1      A1      1.0    0.1    100.0
DESVAR  2      A2      2.0    0.1    100.0
DESVAR  3      A3      1.0    0.1    100.0
DLINK   1      3       0.0    1.0    1      1.00
DOPTPRM APRCOD 2      IPRINT 2      DESMAX 10      DELP    0.5
          P1    1      P2    12      P2CDDV    0
DRESP1  20     W      WEIGHT                                ALL
DRESP1  21     U4     DISP                                12      4
DRESP1  23     S1     STRESS  PROD                        2      11
DRESP1  25     S3     STRESS  PROD                        2      13
$
DRESP2  50     AVGSTRS AVG
          DRESP1 25     23
$

```

...  
ENDDATA

The sensitivity results are shown in [Listing 2-13](#). The user can see that there is a response 23 that is provided for subcase 5 and a response 25 for subcase 6. Response 50 is printed next, which is a “synthesized” response that is the average of responses 23 and 25. Similarly, note that the sensitivities of the synthesized response is the average of the individual sensitivities.

**Listing 2-13 Formatted Design Sensitivity Results from rspan.f06**

***** * *     D E S I G N     S E N S I T I V I T Y     M A T R I X     O U T P U T     * * *     R E S P O N S E     S E N S I T I V I T Y     C O E F F I C I E N T S     * * *****									
-----									
DRESP1	ID=	20	RESPONSE TYPE= WEIGHT				SEID=		
	RESP	VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	
-----									
1		4.8284E+00	1	A1	2.8289E+00	2	A2	1.0002E+00	
-----									
0						FEBRUARY	21, 2003	MSC.NASTRAN	2/21/03    PAGE    23
BASELINE - 2 CROSS SECTIONAL AREAS AS DESIGN VARIABLES									
-----									
DRESP1	ID=	21	RESPONSE TYPE= DISP			GRID ID=	4	COMP NO=	1
SUBCASE	RESP	VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	SEID= 0
-----									
	5	2.2627E-02	1	A1	-2.2631E-02	2	A2	0.0000E+00	
	6	-4.5255E-02	1	A1	4.5262E-02	2	A2	0.0000E+00	
-----									
DRESP1	ID=	21	RESPONSE TYPE= DISP			GRID ID=	4	COMP NO=	2
SUBCASE	RESP	VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	SEID= 0
-----									
	5	-4.4328E-03	1	A1	1.1595E-03	2	A2	1.6368E-03	
	6	-8.8656E-03	1	A1	2.3190E-03	2	A2	3.2736E-03	
-----									
DRESP1	ID=	23	RESPONSE TYPE= STRESS			ELEM ID=	1	COMP NO=	2
SUBCASE	RESP	VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	SEID= 0
-----									
	5	1.3530E+04	1	A1	-1.1895E+04	2	A2	-8.1543E+02	
-----									
DRESP1	ID=	25	RESPONSE TYPE= STRESS			ELEM ID=	3	COMP NO=	2
SUBCASE	RESP	VALUE	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	SEID= 0
-----									
	6	2.7060E+04	1	A1	-2.3789E+04	2	A2	-1.6309E+03	
-----									
DRESP2	ID=	50	RESPONSE TYPE= SYNTHETIC				SEID=		
	SUBCASE	RESP VALUE    FREQ/TIME	DESIGN	VARIABLE	COEFFICIENT	DESIGN	VARIABLE	COEFFICIENT	
-----									
	6	2.0295E+04    0.0000E+00	1	A1	-1.7842E+04	2	A2	-1.2232E+03	

## 2.13 Miscellaneous Optimization Enhancements

### Multiple Frequency Sets in a Frequency Response Analysis

Multiple frequency sets have been supported in past versions of MSC.Nastran frequency response analyses (SOL 108 and 111). However, in versions prior to MSC.Nastran 2004, SOL 200 supported only one frequency set (i.e., the first frequency set was selected for all frequency response subcases and the rest of the frequency sets were neglected). Starting with MSC.Nastran 2004, SOL 200 fully supports the multiple frequency set features.

In addition to the above enhancement, an active frequency concept has also been implemented. This concept calculates sensitivities only for the active constraints instead of all constraints, thereby reducing the CPU requirement.

### Correction in the Use of OFREQ

The OFREQ Case Control command allows limitation of the frequencies at which data recovery is performed. Users were previously advised not to use the OFREQ Case Control command because an error in the implementation of SOL 2000 increased the likelihood of incorrect sensitivity results. Use of OFREQ is no longer restricted in SOL 200. The selection of frequencies of the responses that are used in design is driven by the DRESP1 entries and is not affected by OFREQ. If the OFREQ command is used, it only affects the printed output.

### Rayleigh Quotient Approximation for FREQ and EIGN Response Types

The Rayleigh Quotient Approximation has been implemented to provide an improved estimate of normal modes eigenvalues and eigenfrequencies in the approximate optimization task. In a normal response analysis, the eigenvalue can be expressed using Rayleigh's quotient:

$$\lambda = \frac{\phi^T K \phi}{\phi^T M \phi} = \frac{U}{T} \quad \text{Eq. 2-31}$$

where U is referred to as the modal strain energy or generalized stiffness and T is referred to as the modal kinetic energy or generalized mass.



Taylor series approximation to the modal strain and kinetic energies can be used to construct the eigenvalue approximation. It has been found that reciprocal variables provide a conservative approximation for the strain energies while direct variables are used for the kinetic energies:

$$\tilde{U}_R = U_0 + \Phi^T \sum_{i=1}^n \frac{\partial K}{\partial x_i} (x_i - x_{0i}) \frac{x_{0i}}{x_i} \Phi \quad \text{Eq. 2-32}$$

$$\tilde{T}_D = T_0 + \Phi^T \sum_{i=1}^n \frac{\partial M}{\partial x_i} (x_i - x_{0i}) \Phi \quad \text{Eq. 2-33}$$

And, the eigenvalue approximation by the Rayleigh Quotient is

$$\tilde{\lambda}_{RQA} = \frac{\tilde{U}_R}{\tilde{T}_D} \quad \text{Eq. 2-34}$$

This Rayleigh Quotient Approximation is the standard technique used in approximating eigenvalue (EIGN) and normal mode frequency (FREQ) responses. Approximations using direct and reciprocal approximations are also available using the ATTB field on the DRESP1 entry, but they should not be needed.

## Enhanced Optimization Algorithm

For the structural optimization capability, MSC.Nastran uses the DOT program supplied by Vanderplaats Research and Development. MSC.Nastran 2004 has incorporated the latest version of DOT (Version 5.3). The areas of major changes are:

1. The continuous improvement of the robustness and efficiency of the Sequential Quadratic Programming (SQP) algorithm.
2. Error fixes related to better solving of the optimization problems with the initially infeasible designs.
3. Better handling of the case of more active/violated constraints than the number of design variables for the Modified Method of Feasible Directions algorithm.

# Elimination of Constraints on Properties that are a Linear Function of a Single Design Variable

In the past, the approximate optimization task in MSC.Nastran has imposed constraints on all the designed properties. These constraints have been removed from designed properties that satisfy two conditions:

- 1. The property is a linear function of a single design variable defined using the DVxREL1 entry:

$$p_j = C_o + C_i X_i$$

- 2. The user has not imposed any bounds (PMIN or PMAX) on the DVxREL1 entry.

Move limits on the design variable already restrict the change in this case so that the property constraints are redundant. Removing the constraints simplifies the design task and has been shown to improve the performance and quality of the designs in some of our test cases.

This change required that the default value for the DELX parameter be changed from 1.0 to 0.5. Previously, the DELP default of 0.2 typically prevented the 100% change in design variables.

These changes are likely to result in differences in results from earlier releases of MSC.Nastran. Listing 2-14 shows examples of prints that appear in the .f06 file that reflect this enhancement. The lower and upper bounds that are now listed as *n/a* (not applicable) previously appeared as default property limits, such as -1.0E35.

Listing 2-14

1

0

SYMMETRIC THREE BAR TRUSS DESIGN OPTIMIZATION - D200X1

MAY 13, 2003 MSC.NASTRAN 5/12/03 PAGE 8

BASELINE - 2 CROSS SECTIONAL AREAS AS DESIGN VARIABLES

----- COMPARISON BETWEEN INPUT PROPERTY VALUES FROM ANALYSIS AND DESIGN MODELS -----								
PROPERTY TYPE	PROPERTY ID	PROPERTY NAME	ANALYSIS VALUE	DESIGN VALUE	LOWER BOUND	UPPER BOUND	DIFFERENCE FLAG	SPAWNING FLAG
PROD	11	A	1.000000E+00	1.000000E+00	N/A	N/A	NONE	
PROD	12	A	2.000000E+00	2.000000E+00	N/A	N/A	NONE	
PROD	13	A	1.000000E+00	1.000000E+00	N/A	N/A	NONE	

1.

2.

THE DIFFERENCE FLAG IS USED TO CHARACTERIZE DIFFERENCES BETWEEN ANALYSIS AND DESIGN MODEL PROPERTIES:  
IF THE FLAG IS NONE, THEN THERE IS NO SIGNIFICANT DIFFERENCE BETWEEN THE TWO VALUES.  
IF THE FLAG IS WARNING, THEN THE USER IS ADVISED THAT DIFFERENCES EXIST AND THE DESIGN MODEL IS BEING USED TO OVERRIDE THE ANALYSIS MODEL.  
IF THE FLAG IS FATAL, THEN THE DIFFERENCES ARE GREATER THAN 1.00000E+35 AND THE RUN WILL BE TERMINATED.

THE SPAWNING FLAG (\*) INDICATES THAT THE SPAWNED PROPERTY IS DERIVED EITHER FROM THE BEAM CROSS SECTION LIBRARY OR FROM A PBEM ENTRY. THE PROPERTY ID FOR THE SPAWNED PROPERTY IS IDENTICAL TO ITS PARENT.

## New DESVAR Case Control command to Select a Subset of Design Variables

An additional case control command has been added based on a user request that has general applicability. The new, optional, DESVAR Case Control command selects a set of design variables defined by a SET command. If the command is absent, all DESVAR Bulk Data entries will be utilized in the design task. Only one DESVAR Case Control command can appear and it must appear above the subcase level.

**Format:**

DESVAR = n (all or blank)

Describers	Meaning
All (or blank)	All DESVAR Bulk Data entries will be processed.
n	Set identification of a set of DESVAR Bulk Data entry identification numbers. (Integer > 0)

**Example:**

```
...
CEND
SET 200 = 20, 30
DESVAR = 200
BEGIN BULK
...
DESVAR,10,DV1,1.0,0.1,10.
DESVAR,20,DV1,1.0,0.1,10.
DESVAR,30,DV1,1.0,0.1,10.
DESVAR,40,DV1,1.0,0.1,10.
...
```

For the example, DESVAR entries 20 and 30 will be used in a SOL 200 design study while the other two existing DESVAR entries 10 and 40 are not used. This feature will be useful when either:

- 1. The user wants to rapidly evaluate the effects of a specific set of design variables
- 2. The user has determined that some design variables are unimportant and wants to remove them from the study for clarity and performance reasons.

The existence of this new command forces a change in the rules for what occurs when a DVxRELy entry (e.g., DVPREL1) references an undefined DESVAR entry. Previously, this resulted in the issuing of User Fatal Message 7137 and job termination. With the addition of the new command, if a DVxRELy entry references DESVAR

entries and none of the DESVAR entries are selected by the DESVAR command, then the DVxRELy entry will be discarded internally and the job will continue without a fatal message.

### Old Design Sensitivity and Optimization

The original Design Sensitivity Analysis (DSA) capability has been available in several solution sequences (i.e., SOL 101, SOL 103 and SOL 105) since Version 65. Since Version 66, Solution 200 has been available to perform general design optimization as well as sensitivity analyses.

With the introduction of the eigenvector sensitivity analysis and optimization in SOL 200 in MSC.Nastran 2004, the original DSA capability is no longer necessary as all the features that existed in the original DSA are now fully supported and exceeded by SOL 200. Considering that the existing users may still have built their simulation process around the original DSA capability, beginning in MSC.Nastran 2004, the original DSA capability will be gradually phased out. In MSC.Nastran 2004, the documentation related to the original DSA capability will be removed from the *MSC.Nastran Quick Reference Guide*. In future releases, the code that supports the original DSA will not be maintained, tested, or supported.

The following table lists the affected case control commands, bulk data entries, and parameters that are no longer included in the documentation.

Items Removed from QRG	Remarks
SENSITY	Case Control command
SET2	Case Control command
DSCONS	Bulk Data entry
DVAR	Bulk Data entry
DVSET	Bulk Data entry
EIGD	Bulk Data parameter
NORM	Bulk Data parameter

## Dynamics

---

- Enhanced Methods for Specifying Delay and Phase Angle in Dynamics
- Select Modes to be Included for Response Analysis
- Linear Transient Analysis Restarts (SOLs 109 and 112)
- Enhancements to Initial Condition Usage in Linear Transient Analysis
- Residual Vector Enhancements
- Enforced Motion Enhancements
- Random Response Analysis
- Modal Kinetic and Strain Energy Output
- Enhancements to Structural Damping for Shell and Bush Elements
- Modal Contribution Fraction (MCFRACTION)
- Enhancements to Modal Dynamic Analysis Processing
- MAX/MIN/RMS Methodology in Data Recovery
- Massless Mechanism Identification and Control Enhancements
- General Enhancements for Orthogonalizing Complex Vectors
- Complex Eigenvalue Analysis Updates
- Real Eigenvalue Analysis
- Fast Direct Frequency Response Option
- Inertia Relief with Auto-Suport
- New C-Set and R-Set Reduction

### 3.1 Enhanced Methods for Specifying Delay and Phase Angle in Dynamics

Prior to MSC.Nastran 2004, the DELAY and DPHASE fields in RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE Bulk Data entries had to be specified as integer values. These values in turn pointed to corresponding DELAY and DPHASE Bulk Data entries. Starting with MSC.Nastran 2004, this restriction has been removed, thereby permitting these fields to have either integer or real values. If an integer value is specified, the program operates as it always has. However, if a real value is specified, then that value of DELAY or DPHASE will apply to ALL degrees of freedom (DOFs) that are excited by that dynamic load entry. There is no need in this case to specify any DELAY or DPHASE Bulk Data entries.

In addition to simplifying data preparation, the above feature has one other very important advantage. Previously, time delay and phase angles could not be specified for grid points in upstream superelements because ALL grid points specified on the DELAY and DPHASE Bulk Data entries must belong to the residual structure. Because the enhancements allow the specification of the DELAY and DPHASE values directly on the dynamic load entries, the grid points to which such dynamic loads apply can now reside either in the residual structure or in upstream superelements.

## 3.2 Select Modes to be Included for Response Analysis

### Introduction

Modal dynamic analysis involves the use of mode shapes in the dynamic analysis. Normally, the analysis use all of the computed mode shapes. However, the user has the option of selecting a subset of the computed mode shapes for use in the dynamic analysis. This is accomplished by using the LMODES parameter or the LFREQ/HFREQ parameters. The LMODES parameter allows selection of mode shapes for a specified number of the lowest modes. The LFREQ/HFREQ parameters allows selection of all mode shapes whose frequencies fall within the range specified by these parameters. It should be noted in both of these scenarios that the selected mode shapes represent consecutive mode numbers. (When dealing with fluid mode shapes, the parameters LMODESFL and LFREQFL/HFREQFL play a similar role.)

### Benefits

There are cases, particularly when performing modal participation studies, where there may be a need for a combination of the computed mode shapes whose mode numbers may not all be consecutive. This may occur when only specific computed mode shapes are to be included in the dynamic analysis or specific computed mode shapes are to be excluded from the dynamic analysis. Such selections cannot be accomplished via the existing LMODES and LFREQ/HFREQ parameters. A DMAP alter package called delmodea.v2001 allows users of V2001 to delete individual mode shapes from the dynamic analysis. However, its usage is not very convenient. In order to allow such selections in an elegant and user friendly manner, a new Case Control command called MODESELECT has been developed for use in MSC.Nastran 2004.

The new MODESELECT command can also be used to accomplish the same task as the LMODES and LFREQ/HFREQ parameters. However, its main purpose and usefulness is for making selections of mode shapes that cannot be accomplished via these parameters.

### Inputs

A detailed description of the MODESELECT command is given in the *MSC.Nastran Quick Reference Guide*. Its usage is illustrated below by a few examples. For the purpose of this discussion, a hypothetical problem is considered wherein the real eigenvalue analysis has computed ten modes with natural frequencies of 10.0, 20.0, 30.0, 40.0, 50.0, 60.0, 70.0, 80.0, 90.0, and 100.0 Hz.

## Examples

Purpose: Select the first five modes.  
 Old usage: PARAM,LMODES,5  
 New feature: SET 100 = 1 THRU 5  
 MODESELECT = 100

### Example 2

Purpose: Select all modes with frequencies in the range of 45.0 – 85.0 Hz.  
 Old usage: PARAM,LFREQ,45.0  
 PARAM,HFREQ,85.0  
 New feature: SET 200 = 5 THRU 8  
 MODESELECT = 200  
 This method assumes that frequencies are known.

### Example 3

Purpose: Select all modes excluding modes 6 and 7.  
 Old usage: Not possible  
 New feature: SET 100 = 1 THRU 10 EXCEPT 6,7  
 MODESELECT = 100  
 or  
 SET 200 = 6,7  
 MODESELECT = -200

### Example 4

Purpose: Select mode 5 only.  
 Old usage: PARAM,LFREQ,49.0  
 PARAM,HFREQ,51.0  
 This method assumes that frequencies are known.  
 New feature: SET 100 = 5  
 MODESELECT = 100  
 or  
 MODESELECT = 5 \$ (No SET 5 defined)



## Example 5

Purpose: Select all modes except for mode 6.

Old usage: Not possible

New feature: SET 100 = 6  
MODESELECT = -100  
or  
MODESELECT = -6 \$ (No SET 6 defined)

## 3.3 Linear Transient Analysis Restarts (SOLs 109 and 112)

### Introduction

Previously, restarts in linear transient analysis (SOLs 109 and 112) meant new analyses starting from time zero unless the `dtranra.vxx` or `mtranra.vxx` alter was used. Starting with MSC.Nastran 2004, restarts may continue time step integration from a previous run without repeating the earlier computations. This is accomplished by extending the usage of the `STIME` parameter to SOLs 109 and 112 (while retaining its current usage in SOLs 129 and 159).

### Inputs

In order to use the new continue feature in SOLs 109 and 112 restarts, ensure that the model and the constraints, as well as the subcase setup in the restart run, are the same as those in the previous run. The user may, however, specify different `TSTEP` and `DLOAD` requests in Case Control and also different `TSTEP` and dynamic loading entries in the Bulk Data compared to the previous run. `PARAM,STIME,x` is used to specify the starting time of the restart run. If `STIME` exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to `STIME`. In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any earlier output time of the previous run. MSC.Nastran informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run. The loading and the results output from the continued restart run will start from the new starting time.

### Guidelines and Limitations

It is important to note that the use of the continue feature requires that the model and the constraints as well as the subcase setup in the restart run be the same as those in the previous run. It is the user's responsibility to ensure that this condition is satisfied. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.

This feature replaces the `dtranra.vxx` and `mtranra.vxx` alters in the `ssalter` library.

## 3.4 Enhancements to Initial Condition Usage in Linear Transient Analysis

### Introduction

Two major enhancements have been made to the use of initial conditions in linear transient analysis. These are discussed below.

- Specification of Initial Conditions in Modal Transient Analysis (SOL 112)

Prior to MSC.Nastran 2004, initial conditions were permitted only in direct transient analysis (SOLs 109 and 129). This capability was not available directly in modal transient analysis (SOL 112) unless the mica.vxx alter was used. Starting with MSC.Nastran 2004, this long-standing restriction has been removed. The new capability permits specification of initial conditions in modal transient analysis (SOL 112) either in physical coordinates (as is done in direct transient analysis) or in modal coordinates. To accomplish this, enhancements have also been made to the IC Case Control command and to the TIC Bulk Data entry.

- Usage of Static Solutions as Initial Conditions in Linear Transient Analysis (SOLs 109 and 112)

MSC.Nastran 2004 now allows for the use of static solutions as initial conditions in linear transient analysis (SOLs 109 and 112). Differential stiffness effects may be included if the user so desires. This enhancement permits specification of the results of the static analysis subcase to be used as the initial condition for the transient analysis.

---

**Note:** When initial conditions are specified in modal transient analysis (SOL 112), it is important to employ residual vector processing in order to ensure accurate results. However, this is already satisfied in MSC.Nastran 2004 where residual vector processing is the default scenario in all modal solution sequences.

---

# Input

## IC

The format of the revised IC Case Control command is as follows:

**Format:**

$$\text{IC} \begin{bmatrix} \text{PHYSICAL} \\ \text{MODAL} \\ \text{STATSUB[,DIFFK]} \end{bmatrix} = \text{n}$$

**Examples:**

IC = 10  
IC(PHYSICAL) = 100  
IC(MODAL) = 200  
IC(STATSUB) = 1000  
IC(STATSUB,DIFFK) = 2000

Describer	Meaning
PHYSICAL	The TIC Bulk Data entries selected by set n define initial conditions for coordinates involving grid, scalar, and extra points. (Default).
MODAL	The TIC Bulk Data entries selected by set n define initial conditions for modal coordinates and extra points. See Remark 3.
STATSUB	Use the solution of the static analysis subcase n as the initial condition. See Remark 4.
DIFFK	Include the effects of differential stiffness in the solution. See Remarks 4. and 5.
n	For the PHYSICAL (the default) and MODAL options, n is the set identification number of TIC Bulk Data entries for structural analysis (SOL 109, 112 and 129) or TEMP and TEMPD entries for heat transfer analysis (SOL 159). For the STATSUB option, n is the ID of a static analysis subcase. (Integer > 0)

**Remarks:**

1. For structural analysis, TIC entries will not be used (therefore, no initial conditions) unless selected in the Case Control Section.
2. Only the PHYSICAL option (the default) may be specified in heat transfer analysis (SOL 159).

3. IC(MODAL) may be specified only in modal transient analysis (SOL 112).
4. IC(STATSUB) and IC(STATSUB,DIFFK) may not both be specified in the same execution.
5. The DIFFK keyword is meaningful only when used in conjunction with the STATSUB keyword.

### Examples

1. The following example specifies initial conditions in physical coordinates using SOL 109 or SOL 112.  
IC(PHYSICAL) = 100  
or  
IC = 100
2. The following example specifies initial conditions in modal coordinates using SOL 112.  
IC(MODAL) = 200
3. The following example uses the static solution from Subcase 10 as the initial condition in SOL 109 or SOL 112. The differential stiffness effect is not included.  
SUBCASE 10 \$ STATIC ANALYSIS  
LOAD = 100  
\$  
SUBCASE 20  
IC(STATSUB) = 10
4. The following example uses the static solution from Subcase 100 as the initial condition in SOL 109 or SOL 112. The differential stiffness effect is included.  
SUBCASE 100  
LOAD = 1000  
\$  
SUBCASE 200  
IC(STATSUB,DIFFK) = 100

## 3.5 Residual Vector Enhancements

### Calculation of Residual Vectors in MSC.Nastran

Modal solutions are commonly used to reduce large-size models to relatively smaller representations that include the important dynamics of the original model. In addition to reducing the model size, modal reduction has the added advantage of producing diagonal mass and stiffness matrices for solution calculation. Although modal reduction may capture most of the dynamic response of the structure, the static response may not be complete. This 'error' is due to modal truncation of higher frequency modes that may contribute 'statically' to the total response. Residual vectors are used to improve the results of modal solutions by attempting to account for the response of these higher frequency modes.

Residual vectors can be determined from any set of base vectors. Base vectors are the raw material used to generate the residual vectors in MSC.Nastran. Although any vector can be used as a residual vector, as long as it is partially independent of the modal vectors, it may produce coupling with the modal vectors when added to the modes for matrix reduction. This coupling destroys the diagonal properties of the reduced mass and stiffness matrices.

To ensure that the residual vector used in the reduction process result in diagonal matrices, and better results, the following steps are performed.

1. Ensure that loads are linearly independent with the modal inertial forces.
2. Determine base vectors from static response due to loads.
3. Ensure that base vectors are linearly independent.
4. Orthogonalize the base vectors with respect to the modal vectors to produce residual vectors. These vectors will result in diagonal mass and stiffness matrices.

The closer the base vectors approximate the actual deformation due to the load, the better the residual vectors and the better the results. For this reason, base vectors in MSC.Nastran are determined from the static response of the structure to the following loads:

- Inertial forces due to rigid-body motion
- Applied loads
- Structural, viscous, and inertial forces due to enforced motion
- Forces at user specified discrete degrees of freedom (RVDOFi entries)

- Discrete damping forces due to viscous elements (CDAMPi and CVISC entries)

A new case control command, RESVEC, allows the user to specify the loads used for residual vector calculation. The format for the RESVEC is as follows:

$$\text{RESVEC} \left[ \left( \begin{bmatrix} \text{INRLOD} \\ \text{NOINRL} \end{bmatrix}, \begin{bmatrix} \text{APPLOD} \\ \text{NOAPPL} \end{bmatrix}, \begin{bmatrix} \text{RVDOF} \\ \text{NORVDO} \end{bmatrix}, \begin{bmatrix} \text{DMPLOD} \\ \text{NODMP} \end{bmatrix} \right) \right] = \left\{ \begin{array}{c} \text{SYSTEM/NOSYSTEM} \\ \text{COMPONENT/NOCOMPONENT} \\ \text{BOTH or YES} \\ \text{NO} \end{array} \right\}$$

**Examples:**

RESVEC=SYSTEM  
RESVEC(NOINRL)=COMPONENT  
RESVEC=NO

Describer	Meaning
INRLOD/ NOINRL	Controls calculation of residual vectors based on inertia relief. (Default=INRLOD)
APPLOD/ NOAPPL	Controls calculation of residual vectors based on applied loads. (Default=APPLOD)
RVDOF/ NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default=RVDOF)
DMPLOD/ NODMP	Controls calculation of residual vectors based on viscous damping.
SYSTEM/ NOSYSTEM	Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, describers inside the parentheses are ignored. See Remark 2 for default.
COMPONENT/ NOCOMPONENT	Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, describers inside the parentheses are ignored. See Remark 2 for default.
BOTH or YES	Requests calculation of residual vectors for both system modes and component modes. See Remark 2 for default.
NO	Turns off calculation of residual vectors for both system and component modes and describers inside the parentheses are ignored. See Remark 2 for default.

The inertial forces due to the normal mode shapes are removed from the loads used for residual vector calculation using the following:

$$\{\bar{P}\} = \{P\} - [M][\phi]([\phi][M][\phi])^{-1}[\phi]\{P\}$$

The modified loads are applied to the structure to determine the deformations that serve as the base vectors for residual vector calculation.

$$[K]\{V_{load}\} = \{\bar{P}\}$$

Additionally, for transient response, initial conditions are also added to the set of base vectors. Before the initial condition vectors are added, any mode shape content is removed.

$$\{\bar{V}_{initial}\} = \{V_{initial}\} - [\phi]([\phi][M][\phi])^{-1}[\phi][M]\{V_{initial}\}$$

The initial condition vectors are appended to the load vectors and the complete set of base vectors are made linearly independent. This process removes vectors that are linear combinations of other vectors in the base vector set. The remaining base vectors are then made orthogonal with respect to the mass and stiffness matrices. The resulting orthogonal vectors are the residual vectors.

The residual vectors are appended to the modes and the reduced mass and stiffness matrices are modified to include the residual vector additions.

$$[\bar{\phi}] = [\phi_{modes} | \phi_{residual}]$$

$$[\bar{M}] = [\bar{\phi}]^T [M] [\bar{\phi}]$$

$$[\bar{K}] = [\bar{\phi}]^T [K] [\bar{\phi}]$$

where  $[\bar{M}]$  and  $[\bar{K}]$  are diagonal matrices.

Residual vectors which produce ‘zero’ mass terms in the reduced mass matrix are treated as ‘massless’ modes. The diagonal mass is set to 1.0E-36 and the diagonal stiffness is set to 1.0. The augmented set of modes/residual vectors is used to reduce the damping, applied loads, and any external matrix input.

Residual vector calculation is available for all modal solutions in MSC.Nastran. They can be calculated for the residual structure and superelements to improve the analysis results.



## Enhancements to Degrees of Freedom Specification for Residual Vector Processing

Prior to MSC.Nastran 2004, the specification of degrees of freedom where unit loads were to be applied to obtain static solutions for use in residual vector computations required the use of USETi,U6 and SEUSETi,U6 Bulk Data entries. In analyses involving multiple superelements, these entries had the following disadvantages:

- Separate entries were required for each superelement.
- The unit loads on a superelement due to these entries were not passed downstream for residual vector processing by the downstream superelements.

In order to overcome the above drawbacks, two new Bulk Data entries called RVDOF and RVDOF1 have been incorporated into MSC.Nastran 2004. These entries have the following advantages:

- Separate entries for separate superelements are no longer required. Instead, these new entries may be used to reference points that are interior to any superelement. The program will automatically partition the data for allocation to the appropriate superelements.
- The unit loads applied to a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path. This yields more accurate results.

## 3.6 Enforced Motion Enhancements

### Introduction

The enforced motion capability in dynamic analysis using the SPC/SPCD specification was first introduced in MSC.Nastran 2001. This feature has been very well received by the user community as an excellent alternative to the large mass approach and the Lagrange Multiplier Technique (LMT), which were the only methods available for enforced motion applications until then. However, certain errors and deficiencies associated with the feature have been discovered since the release of MSC.Nastran 2001. The most important of these errors can be summarized under the following three broad categories:

- The SPC/SPCD method produces erroneous results in modal dynamic analysis (SOL 111 and SOL 112) if modal damping is specified (using SDAMPING/TABDMP1 data). The larger the amount of modal damping specified, the greater the errors in the results.
- The SPC forces, the MPC forces as well as the element stresses and forces obtained in modal frequency response analysis (SOL 111) are in error at very low forcing frequencies.
- In cases wherein the enforced motion of the base is very large compared to the motion relative to the base, both SOL 109 and SOL 112 may yield incorrect results.

In order to resolve the above serious errors and deficiencies, the formulation of the enforced motion feature in V2004 has been completely revised. It is now based on the relative motion approach rather than on the absolute motion approach used in MSC.Nastran 2001. (The mathematical details of the new formulation are given in the [\*MSC.Nastran Reference Manual\*](#).)

### Benefits

With the new formulation, all major problems associated with the SPC/SPCD method in dynamic analysis in MSC.Nastran 2001 have been fully and effectively resolved. The SPC/SPCD method may now be regarded as the method of choice for all enforced motion applications in dynamic analysis. In addition, to enhanced user convenience, a new user parameter called ENFMOTN has been introduced in MSC.Nastran 2004 for use with the SPC/SPCD method in SOLs 108, 109, 111, 112, 146, and 200. The default value for this parameter is ABS, implying that the results of the analysis represent absolute motion of the model. If the value for ENFMOTN is specified as REL, then the results represent motion relative to the enforced motion of the base. In the case of

modal dynamic analysis (SOL 111 and SOL 112), this latter scenario is equivalent to employing the large mass approach and excluding the rigid body modes from the analysis.

Another benefit of the new enforced motion formulation in MSC.Nastran 2004 is that, in many cases, it is apt to give better results in modal dynamic analysis than the old formulation in MSC.Nastran 2001 even without residual vectors. What this means is that while residual vectors were critical to getting good results when using enforced motion in modal dynamic analysis in MSC.Nastran 2001, they are not that critical to getting good results in MSC.Nastran 2004. Of course, even with the new formulation in MSC.Nastran 2004, usage of residual vectors in modal dynamic analysis is recommended in order to get improved results. (This is already satisfied in MSC.Nastran 2004 where residual vector processing is the default scenario in all modal dynamic analysis solution sequences.)

## 3.7 Random Response Analysis

### Introduction

MSC.Nastran random analysis capability originated as a data post-processing procedure following a frequency analysis. The Power Spectral Density Function (PSDF), Auto-correlation Function (AUTO), Root Mean Square (RMS) and Number of Zero Crossing (N0) of response quantities are computed through X-Y plot Case Control commands for the user-specified individual component responses at the grid points or elements.

In MSC.Nastran 2004, the random analysis capability has been significantly enhanced by adding the following features.

1. Cross-power spectral density and cross-correlation functions.
2. Cumulative RMS (CRMS).
3. Log-Log approximation when numerically computing RMS, N0 and CRMS.
4. Output of PSDF, AUTO, RMS, N0, and CRMS in both print and punch files.

### Cross-Power Spectral Density and Cross-Correlation Functions

#### Theory

With Reference 8., for a pair of response quantities,  $u_a(t)$  and  $u_b(t)$ , the cross-correlation function between them is defined as

$$R_{ab}(\tau) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_0^T u_a(t) u_b(t - \tau) dt \quad \text{Eq. 3-1}$$

where  $u_a(t)$  and  $u_b(t)$  can be displacement, velocity or single-point constraint force responses at the same or different grid points; or stress, strain, and force components in the same or different elements. The cross-correlation functions have the following relation

$$R_{ab}(-\tau) = R_{ba}(\tau) \quad \text{Eq. 3-2}$$

The cross-power spectral density (cross-PSD) of  $u_a(t)$  and  $u_b(t)$  is defined as [Eq. 3-3](#).

$$\begin{aligned}
 S_{ab}(\omega) &= \lim_{T \rightarrow \infty} \frac{2}{T} \left( \int_0^T u_a(t) e^{-i\omega t} dt \right) \left( \int_0^T u_a(t) e^{i\omega t} dt \right) \\
 &= 2 \int_{-\infty}^{+\infty} R_{ab}(\tau) e^{-i\omega \tau} d\tau
 \end{aligned}
 \tag{Eq. 3-3}$$

and  $R_{ab}(\tau)$  is related to  $S_{ab}(\omega)$  by the relation

$$R_{ab}(\tau) = \frac{1}{4\pi} \int_{-\infty}^{+\infty} S_{ab}(\omega) e^{i\omega \tau} d\omega
 \tag{Eq. 3-4}$$

It should be mentioned that the cross-PSD  $S_{ab}(\omega)$  is a complex-valued function of angular frequency  $\omega$ . The following relations hold for the cross-PSDs

$$S_{ab}^*(\omega) = S_{ab}(-\omega) = S_{ba}(\omega)
 \tag{Eq. 3-5}$$

where  $S_{ab}^*(\omega)$  is the complex conjugate of  $S_{ab}(\omega)$ . If we denote  $S_{ab}(\omega)$  by its real and imaginary parts as

$$S_{ab}(\omega) = S_{ab}^{(R)}(\omega) + iS_{ab}^{(I)}(\omega)$$

then from [Eq. 3-5](#), we can see that  $S_{ab}^{(R)}(\omega)$  is an even function and  $S_{ab}^{(I)}(\omega)$  is an odd function of  $\omega$ , i.e.,

$$S_{ab}^{(R)}(-\omega) = S_{ab}^{(R)}(\omega)
 \tag{Eq. 3-6}$$

$$S_{ab}^{(I)}(-\omega) = -S_{ab}^{(I)}(\omega)
 \tag{Eq. 3-7}$$

Substituting [Eq. 3-6](#) and [Eq. 3-7](#) into [Eq. 3-4](#), we have

$$R_{ab}(\tau) = \frac{1}{2\pi} \int_0^{\infty} [S_{ab}^{(R)}(\omega) \cos(\omega \tau) - S_{ab}^{(I)}(\omega) \sin(\omega \tau)] d\omega
 \tag{Eq. 3-8}$$

If the system is subject to multiple sources of excitation, the cross-PSD of responses  $u_a(t)$  and  $u_b(t)$  is related to the PSDs of excitation sources by the frequency response functions as

$$S_{ab}(\omega) = \sum_j \sum_k H_{aj}(\omega) H_{bk}^*(\omega) S_{jk}(\omega) \quad \text{Eq. 3-9}$$

where  $H_{aj}(\omega)$  is the frequency response function of  $u_a(t)$  corresponding to the excitation source  $Q_j(t)$ ,  $H_{bk}^*(\omega)$  the complex conjugate of  $H_{bk}(\omega)$ , and  $S_{jk}(\omega)$  the cross-PSD of excitation sources when  $j \neq k$ . If the two different excitation sources  $Q_j$  and  $Q_k$  are not correlated, we have  $S_{jk}(\omega) \equiv 0$ .

The aforementioned equations of cross-PSDs and cross-correlation functions become the ones for auto-PSDs and auto-correlation functions when  $u_a(t) = u_b(t)$ .

The calculation of the cross-correlation function from [Eq. 3-8](#) is carried out by numerical integration based on the trapezoidal approximation.

## Inputs

Case Control command, RCROSS and Bulk Data entry, RCROSS, are used to request the computation and output of cross-PSD and cross-correlation functions, along with Case Control command, RANDOM. An example of how they are used is shown as follows.

```
SOL 108
TIME 100
CEND
SPC    = 1
DLOAD = 1
FREQ   = 1
RANDOM  = 10
RCROSS(IMAG,PSDF,CORF,PRINT,PUNCH) = 100
SET 1 = 3306,7404,6708,6722,6750,202,205,3402,3405,6904,6404,6423,7504,
7519,6808,6837,9906,7004
SET 2 = 201,202,3401,3402,3301,3302,6701,6702,6703,6901,6902,7001,7002,
7401,7402,6401,6402,6411,6412,7501,7502,7511,7512,6801,6811,9901,9902
SET 3 = 201,3401,3305,9905,7001,7401,6702,6735,2001,6902,6413,3303,9903
DISP(PHASE,PRINT) = 1
ELST(PHASE,PRINT) = 2
ELFO(PHASE,PRINT) = 2
SPCF(PHASE,PRINT) = 3
$
SUBCASE 1
.
.
.
SUBCASE 2
.
.
.
BEGIN BULK
RCROSS 100      FORCE 202      167      FORCE 3302      6      1330202
```

```
RCROSS 100      FORCE 3302      6              3302      6      1330201
RCROSS 100      FORCE 7402      8              7002      8      1740204
RCROSS 100      STRESS 201      16      STRESS 3301      7      2330102
RCROSS 100      STRESS 6411      7              6411      7      2641101
RCROSS 100      FORCE 9902      6      STRESS 9901      7      2990106
RCROSS 100      STRESS 9901      7      FORCE 9902      6      2990108
RCROSS 100      DISP 3306      3      DISP 3306      3      3330601
RCROSS 100      SPCF 3305      3      DISP 3306      3      4641306
RCROSS 100      DISP 3306      3      SPCF 3305      3      4641308
.
.
.
ENDDATA
```

## Outputs

The output of both cross-power spectral density and cross-correlation functions can be written to print and punch files upon the user’s request by selecting the options in Case Control command, RCROSS. A typical output is shown below.

SEQUENTIAL CURVE-ID = 33								
C O M P L E X C R O S S - P O W E R S P E C T R A L D E N S I T Y F U N C T I O N								
(REAL/IMAGINARY)								
0	RCROSS	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID
0	100	FORC	9902	6	STRE	9901	7	2990106
FREQUENCY CPSDF FREQUENCY CPSDF								
0.0		5.833640E+05 / -1.639098E-04			1.500000E+00	2.703971E+05 / 3.082056E+01		
2.500000E+00		1.322806E+05 / 4.212517E+01			3.500000E+00	7.918695E+04 / 4.983960E+01		
4.500000E+00		5.359973E+04 / 5.640685E+01			6.599999E+00	2.977418E+04 / 6.976405E+01		
1.000000E+01		1.649376E+04 / 9.695065E+01						
SEQUENTIAL CURVE-ID = 33								
C R O S S - C O R R E L A T I O N F U N C T I O N								
0	RCROSS	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID
0	100	FORC	9902	6	STRE	9901	7	2990106
TIME CCORF TIME CCORF TIME CCORF TIME CCORF								
0.0		1.179985E+06	1.000000E-01	4.990816E+05	2.000000E-01	1.974606E+05	3.000000E-01	6.946910E+04
4.000000E-01		3.160208E+04						

A typical punch output is as follows:

```

$TITLE   = QA DECK RCROSS01, RANDOM ANALYSIS, COUPLED EXCITATIONS      449
$SUBTITLE= CROSS-PSDF AND CROSS-CORRELATION FUNCTIONS                  450
$LABEL   = RECTANGULAR FORM OF COMPLEX NUMBERS                        451
$CROSS-PSDF                                          452
$REAL-IMAGINARY OUTPUT                                          453
$RANDOM ID =          10, RCROSS ID =          100                  454
$RTYPE1 = FORC, ID1 =          9902, COMP1 =          6            454
$RTYPE2 = STRE, ID2 =          9901, COMP2 =          7            455
    0.000000E+00      5.833640E+05      -1.639098E-04            456
    1.500000E+00      2.703971E+05      3.082056E+01            457
    2.500000E+00      1.322806E+05      4.212517E+01            458
    3.500000E+00      7.918695E+04      4.983960E+01            459
    4.500000E+00      5.359973E+04      5.640685E+01            460
    6.599999E+00      2.977418E+04      6.976405E+01            461
    1.000000E+01      1.649376E+04      9.695065E+01            462

$TITLE   = QA DECK RCROSS01, RANDOM ANALYSIS, COUPLED EXCITATIONS      1113
$SUBTITLE= CROSS-PSDF AND CROSS-CORRELATION FUNCTIONS                  1114
$LABEL   = RECTANGULAR FORM OF COMPLEX NUMBERS                        1115
$CROSS-CORRELATION FUNCTION                                          1116
$REAL OUTPUT                                          1117
$RANDOM ID =          10, RCROSS ID =          100                  1118
$RTYPE1 = FORC, ID1 =          9902, COMP1 =          6            1118
$RTYPE2 = STRE, ID2 =          9901, COMP2 =          7            1119
    0.000000E+00      1.179985E+06            1120
    1.000000E-01      4.990816E+05            1121
    2.000000E-01      1.974606E+05            1122
    3.000000E-01      6.946910E+04            1123
    4.000000E-01      3.160208E+04            1124

```

## Numerical Integration Using Log-Log Approximation

### Theory

The Root Mean Square (RMS) of a random response quantity  $u_j(t)$  is defined as

$$\bar{u}_j = \left[ \frac{1}{2\pi} \int_0^{\infty} S_j(\omega) d\omega \right]^{\frac{1}{2}} \quad \text{Eq. 3-10}$$

where  $S_j(\omega)$  is the PSDF of  $u_j(t)$  and  $\omega$  the angular frequency.

In numerical calculation, the integral in **Eq. 3-10** is computed for a specified frequency range

$$\int_0^{\infty} S_j(\omega) d\omega = \sum_{i=1}^{N-1} \int_{\omega_i}^{\omega_{i+1}} S_j(\omega) d\omega \quad \text{Eq. 3-11}$$

where  $\omega_i$  ( $i = 1, 2, \dots, N$ ) are a set of selected frequencies.



In a Log-Log scaled coordinate system, denote

$$Y = \log S_j(\omega) \quad \text{and} \quad X = \log(\omega)$$

where  $\log(\cdot)$  is the logarithmic function to the base 10. The linear interpolation in a sub-domain  $[\omega_i, \omega_{i+1}]$  in this logarithmic coordinate system yields the following relation

$$Y - Y_i = \frac{Y_{i+1} - Y_i}{X_{i+1} - X_i} (X - X_i) \quad \text{Eq. 3-12}$$

From [Eq. 3-12](#), the approximation of  $S_j(\omega)$  takes the form as

$$S_j(\omega) = \frac{S_j(\omega_i)}{\omega_i^\alpha} \omega^\alpha, \quad \omega_i \leq \omega \leq \omega_{i+1} \quad \text{Eq. 3-13}$$

with

$$\alpha = \frac{\log(S_j(\omega_{i+1})/S_j(\omega_i))}{\log(\omega_{i+1}/\omega_i)} \quad \text{Eq. 3-14}$$

The integral in sub-domain  $[\omega_i, \omega_{i+1}]$  is calculated as follows

$$\int_{\omega_i}^{\omega_{i+1}} S_j(\omega) d\omega = \begin{cases} \omega_i S_j(\omega_i) \ln(\omega_{i+1}/\omega_i), & \alpha = -1 \\ \frac{S_j(\omega_i)}{(1+\alpha)\omega_i^\alpha} (\omega_{i+1}^{\alpha+1} - \omega_i^{\alpha+1}), & \alpha \neq -1 \end{cases} \quad \text{Eq. 3-15}$$

$N_0$  is defined as

$$N_0 = \frac{\overline{r_j}}{\overline{\mu_j}} \quad \text{Eq. 3-16}$$

where

$$\overline{r_j} = \left[ \frac{1}{(2\pi)^3} \sum_{i=1}^{N-1} \int_{\omega_i}^{\omega_{i+1}} \omega^2 S_j(\omega) d\omega \right]^{\frac{1}{2}} \quad \text{Eq. 3-17}$$

Using the Log-Log interpolation for the approximation, we have

$$\int_{\omega_i}^{\omega_{i+1}} \omega^2 S_j(\omega) d\omega = \begin{cases} \omega_i^3 S_j(\omega_i) \ln(\omega_{i+1} / \omega_i), & \alpha = -3 \\ \frac{S_j(\omega_i)}{(3 + \alpha)\omega_i^\alpha} (\omega_{i+1}^{\alpha+3} - \omega_i^{\alpha+3}), & \alpha \neq -3 \end{cases} \quad \text{Eq. 3-18}$$

This logarithmic approach yields more accurate results if the PSDF curve is a straight line in an integration sub-domain in a Log-Log scaled coordinate system. When the PSDF curve is close to a straight line in a Log-Log scaled coordinate system, it can be expected that the Log-Log approximation would produce a better result with fewer frequency points as compared to the trapezoidal approximation of the PSDF using a linear coordinate system.

## Inputs

Parameter, RMSINT, Log-Log selects the Log-Log approach for calculating RMS, N0 and CRMS.

## Cumulative Root Mean Square (CRMS)

### Theory

The cumulative RMS function  $\bar{u}_j(\omega)$  of a random response quantity  $u_j(t)$  at a set of selected  $N$  frequency points is defined as

$$\bar{u}_j(\omega_i) = \left[ \frac{1}{2\pi} \int_{\omega_1}^{\omega_i} S_j(\omega) d\omega \right]^{\frac{1}{2}}, \quad i = 1, 2, \dots, N \quad \text{Eq. 3-19}$$

## Inputs

CRMS function can be output in both the print and punch files. The option of CRMS is selected in Case Control commands, ACCELERATION, DISPLACEMENT, FORCE, MPCFORCES, OLOAD, SPCFORCES, STRAIN, STRESS and VELOCITY.

## Output

PSDF, AUTO, RMS, N0, and CRMS output in Both Print and Punch Files.

These output requests can be achieved by selecting the RPRINT and RPUNCH options in Case Control commands, ACCELERATION, DISPLACEMENT, FORCE, MPCFORCES, OLOAD, SPCFORCES, STRAIN, STRESS and VELOCITY.

Exercise caution when using the RPRINT and RPUNCH options. For real-world problems, an enormous amount of output may be generated using the RPRINT option. To turn this off, use the NORPRINT option. The default is RPRINT.

## 3.8 Modal Kinetic and Strain Energy Output

### Introduction

Modal kinetic and strain energy calculations and output may now be requested in both modal frequency and transient response analyses; i.e., Solution Sequences 111, 112, 146, and 200.

### Benefits

It is now possible to determine the kinetic and strain energy contribution of each normal mode in a modal frequency or transient response analysis. These enhancements are intended to replace the mfrequa and mtranea SSSAlters.

### Theory

Three types of output will be computed: Actual Kinetic (or Strain) Energy, Normalized Kinetic (or Strain) Energy, and Fractional Kinetic (or Strain) Energy.

1. Modal kinetic energy is computed as follows.

In frequency response,

$$[\text{Actual Energy}] = 0.5[\text{diag}(\omega_i^2)][M_{hh}][u_h] \cdot [u_h]$$

where  $\omega$  is the excitation (or forcing) frequency.

In transient response,

$$[\text{Actual Energy}] = 0.5[M_{hh}][u_h] \cdot [u_h]$$

2. The normalized energy is computed by normalizing each column of the actual energy by the highest energy in each column.
3. The fractional energy is computed by dividing the normalized energy at a given forcing frequency (or time step) by the sum of all normalized energies across all natural frequencies.

### Inputs

Modal kinetic and strain energy output is requested with the following Case Control commands, MODALKE and MODALSE.

**Format for kinetic energy**

$$\text{MODALKE} \left( \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix} \begin{bmatrix} \text{PRINT} \\ \text{NOPRINT} \end{bmatrix} \text{PUNCH} \right) \begin{bmatrix} \text{ESORT} = \begin{Bmatrix} \text{MODE} \\ \text{ASCEND} \\ \text{DESCEND} \end{Bmatrix} \end{bmatrix},$$

$$\left[ \text{THRESH} = e \right] \left[ \begin{Bmatrix} \text{TIME} \\ \text{FREQ} \end{Bmatrix} = \begin{Bmatrix} \text{ALL} \\ r \end{Bmatrix} \right] ) = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

**Format for strain energy**

$$\text{MODALSE} \left( \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix} \begin{bmatrix} \text{PRINT} \\ \text{NOPRINT} \end{bmatrix} \text{PUNCH} \right) \begin{bmatrix} \text{ESORT} = \begin{Bmatrix} \text{MODE} \\ \text{ASCEND} \\ \text{DESCEND} \end{Bmatrix} \end{bmatrix},$$

$$\left[ \text{THRESH} = e \right] \left[ \begin{Bmatrix} \text{TIME} \\ \text{FREQ} \end{Bmatrix} = \begin{Bmatrix} \text{ALL} \\ r \end{Bmatrix} \right] ) = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

**Examples**

MODALSE= ALL  
 SET 100= 1, 3, 4, 5, 7  
 MODALKE(ESORT=ASCEND,THRESH=.0001)= 100

Describer	Meaning
SORT1	Output will be presented as a tabular listing of modes for each frequency or time step.
SORT2	Output will be presented as a tabular listing of frequencies or time steps for each mode.
PRINT	Write the results to the .f06 file. (Default)
NOPRINT	Do not write the results to the .f06 file
PUNCH	Write the results to the punch (.f07) file
ESORT	Ppresent the modal energies sorted by mode number, ascending energy value or descending energy value. (Default is MODE).
THRESH	Write out only those energies greater than e. (Default=0.001).

Describer	Meaning
TIME or FREQ	Compute energies at all time steps or frequencies or the set of frequencies defined by SET r. (Default=ALL).
ALL, n, NONE	Compute modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.

Outputs

The output data blocks are SNRGYPLT and KNRGYPLT.

For the SORT1 option:

Kinetic (or Strain) Energy				
Time (or Frequency)= value				
MODE NUMBER	Actual	Normalized	Fractional	
i	xxx.xx	xxx.xx	xxx.xx	
i+1	.	.	.	
.	.	.	.	
.	.	.	.	
i+n	.	.	.	

Above form repeated for each time or frequency.

For the SORT2 option:

Kinetic (or Strain) Energy				
Mode Number: i				
Time (or Frequency)	Actual	Normalized	Fractional	
value <sub>1</sub>	xxx.xx	xxx.xx	xxx.xx	
value <sub>2</sub>	.	.	.	
.	.	.	.	
.	.	.	.	

Above form repeated for each mode.

Guidelines and Limitations

None.

## Example

The following commands:

```
MODALSE(SORT1, THRESH=0.0)= all
MODALKE(SORT2, ESORT=DESCEND, THRESH=0.0)= all
```

produce the following output (note these output samples are excerpts):

FREQUENCY = 1.000000E+00			
M O D A L   S T R A I N   E N E R G Y			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	2.415431E-02	1.000000E+00	7.359713E-01
2	3.425556E-04	1.418196E-02	1.043752E-02
3	1.639720E-04	6.788520E-03	4.996156E-03
4	8.846080E-06	3.662319E-04	2.695362E-04
5	6.052715E-03	2.505853E-01	1.844236E-01
MODE NUMBER =	1		
M O D A L   K I N E T I C   E N E R G Y			
FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	8.147641E-04	1.000000E+00	9.924864E-01
2.000000E+00	4.066131E-03	1.000000E+00	9.936854E-01
3.000000E+00	1.411670E-02	1.000000E+00	9.955438E-01
4.000000E+00	5.744822E-02	1.000000E+00	9.977741E-01
5.000000E+00	7.744190E-01	1.000000E+00	9.996839E-01

## 3.9 Enhancements to Structural Damping for Shell and Bush Elements

### Introduction

The ability of the CBUSH element to have a single frequency dependent structural damping coefficient has been a long-standing key feature of MSC.Nastran. MSC.Nastran 2004 extends this feature by supporting separate frequency-dependent structural damping coefficients for all six degrees of freedom for the CBUSH element.

The structural damping applied to shell elements in MSC.Nastran had been limited to the structural damping coefficient related to the MID1 field of the PSHELL entry. MSC.Nastran 2004 extends this feature by supporting structural damping coefficients, each related to the MID1, MID2, MID3, and MID4 fields of the PSHELL entry.

### Benefits

The CBUSH element is a generalized spring-damper element. In frequency response, the structural damping is associated with the imaginary portion of the complex stiffness matrix while viscous damping is associated with the  $i\omega \cdot B$  portion. Thus if a user desired different structural damping coefficients in each of the six degrees of freedom, it was necessary to invent an equivalent viscous damping for each degree of freedom. With MSC.Nastran 2004, this restriction is removed and up to six different  $ge_i$  with their associated frequency dependent tables can be defined. Thus, removing the need to invent an equivalent viscous damping for each degree of freedom.

In composite or orthotropic shell analysis, many clients have their own special pre-processing programs that create MAT2 or MAT8 entries. These programs often distinguish between membrane and bending structural damping coefficients. Historically, MSC.Nastran has only allowed the structural damping coefficient associated with the MID1 field of the PSHELL entry to be used. With MSC.Nastran 2004, this restriction is removed, and at a users request, separate structural damping coefficients can be associated with each of the MID1, MID2, MID3, and MID4 fields of the PSHELL entry.

### Theory

#### CBUSH Matrices

If both stiffness  $K$  and structural damping  $ge$  are specified, each with its own frequency dependent tables, matrix terms of the following form are created:



$$K_j^f + i \cdot ge_j^f \cdot K_j^f \quad \text{Eq. 3-20}$$

If stiffness  $K$  with frequency dependent tables and non frequency dependent structural damping  $ge$  are specified, matrix terms of the following form are created:

$$K_j^f + i \cdot ge_j^0 \cdot K_j^f \quad \text{Eq. 3-21}$$

If non frequency dependent stiffness  $K$  with frequency dependent structural damping  $ge$  are specified, matrix terms of the following form are created:

$$K_j^0 + i \cdot ge_j^f \cdot K_j^0 \quad \text{Eq. 3-22}$$

In the above three expressions, the superscript  $f$  denotes frequency dependent and the superscript 0 denotes a nominal value. The subscript  $j$  implies the  $j$ -th degree of freedom of the CBUSH element. The real term goes into the element stiffness matrix and the imaginary term goes into the element  $K^4$  matrix.

## PSHELL

If PARAM,SHLDAMP,DIFF is present in the run, a structural damping matrix  $K^4$  is formed for each shell element based on a relationship of the following form:

$$\begin{aligned} K^4 = & ge_{mid1} \times membrane \cdot stiffness \\ & + ge_{mid2} \times bending \cdot stiffness \\ & + ge_{mid3} \times transverse \cdot stiffness \\ & + ge_{mid4} \times combined \cdot stiffness \end{aligned} \quad \text{Eq. 3-23}$$

## Inputs

The existing PBUSH-PBUSHT Bulk Data interface has been enhanced by allowing for five additional structural damping coefficients and their corresponding frequency dependent tables.

The parameter SHLDAMP defaults to the value SAME, which implies use only the MID1 associated value of structural damping coefficient. A character value other than SAME such as DIFF implies that each MIDi entry on the PSHELL can have its own associated value for the structural damping coefficient.

## Outputs

The resulting  $K^4$  matrices will have appropriate terms accounting for additional structural damping coefficients

## Guidelines and Limitations

The following suggestions are associated with the new capability:

1. Multiple  $ge_i$  values on the PBUSH or the use of PARAM,SHLDAMP,DIFF causes additional element stiffness matrix generations which will increase run time.
2. With the PARAM,SHLDAMP,DIFF option, relatively large values for the structural damping coefficient associated with MID4 can cause structural instability in transient dynamics.
3. For nonlinear shell elements, PARAM,SHLDAMP,DIFF is ignored and the MID1 associated value of structural damping coefficient is used.

## Example

The following partial input example demonstrates the form of the new entries:

### Partial Input File

```
$ SHLDAMP in either BULK or CASE CONTROL
$ SHLDAMP is associated with PSHELL entries only.
$ It has no relevance to PBUSH-PBUSHT entries.
$
PARAM,SHLDAMP,DIFF
$
PSHELL  50      51      1.0      52      53
          54
MAT2    51      3.4372+61.0312+6      3.4372+6      1.2022+68.9873-4
          .01
MAT2    52      9.1151+62.7345+6      9.1151+6      3.1897+68.9873-4
          .02
MAT2    53      3.7736+4      3.7736+4      8.9873-4
          .01
MAT2    54      3.4372+31.0312+3      3.4372+3      1.2022+38.9873-4
          .01
$
PBUSH   456      K      101.      202.      303.      404.      505.      606.
          GE      .02      .02      .02      .02      .02      .02
PBUSHT  456      K      101      101      101      101      101      101
          GE      107      107      104      105      107
$
TABLED1 101
          10.      700.      20.      700.      ENDT
TABLED1 104
          10.      .023      20.      .023      ENDT
```

TABLED1	105				
	10.	.01	20.	.02	ENDT
TABLED1	107				
	10.	.01	20.	.023	ENDT
\$					

## 3.10 Modal Contribution Fraction (MCFRACTION)

### Introduction

The response of large structural components to loads in the frequency or time domain can be determined using either a direct or modal solution approach. The modal approach is typically employed for large models when the response of the structure can be approximated using only a small number of its natural modes. During this process, data recovery and postprocessing of large quantities of response data usually occurs. Often, the user desires detailed information for several points of interest in the model. One item of particular interest is often referred to as the *modal contribution fraction* or the *modal participation factor*. As these names imply, the value for this item is defined as that portion of the total response at the point that can be attributed to a particular natural mode. This type of information has been available to the DMAP user for some time through the SSS alter library (modconta.vxx). However, the user interface to control these calculations using the alter was somewhat inconvenient. With the advent of the MCFRACTION command and associated DMAP module, usage of modconta.vxx is no longer necessary.

### Benefits

The MCFRACTION command provides more control over the amount of output produced during the modal contribution fraction calculation as compared to using the feature with modconta.vxx. The SSS alter is capable of computing and printing several different “views” of the modal contribution fraction data. A DMAP parameter (PARTFAC) specifies one of the several possible views. Only one view could be produced in any single execution of MSC.Nastran. Filtering or sorting of the data could not be controlled. The new MCFRACTION command gives the analyst access to such features through a convenient user interface. All possible views of the data using the SSS alter are provided at the same time. Separate runs are not necessarily to obtain each required view. This could substantially reduce the amount of time the analyst spends gathering all of the data and quite possibly will reduce computer-processing time as well. The new punch output option allows local postprocessing of the data to be tailored to the user’s needs, further enhancing data interpretation.

### Theory

Modal analysis solutions are based on the theory that the effects of a small number of natural modes can be used to represent the response of the structure. The eigenvectors of these modes are used to transform the problem from the physical domain into a scalar domain of generalized coordinates. This transformation results in a much smaller problem size that can be solved rapidly. Once the solution to the

generalized problem is obtained, the solution in the physical domain is obtained by transformation, again using the eigenvectors of the natural modes. The response at a physical degree of freedom (a grid point's translation or rotation component) for a particular forcing frequency or time step can be found by summing the individual responses of each mode shape. This is accomplished by simple matrix multiplication of the eigenvectors times the generalized solution vectors. The calculation of modal contribution fractions can be performed as part of the total response calculation.

In MSC.Nastran, several different views of the modal contribution fractions are computed. All start with some basic ingredients. These are the total modal solution response at the required degrees of freedom and the transformation matrix from modal to physical coordinates. The solution responses are complex (a real term and an imaginary term) for frequency response and complex eigenvalue solutions. The responses involve only real terms for transient analysis. The following discussion pertains to complex response results, but is applicable to transient analysis as well. The only difference is that there are no imaginary terms and phase angles are all zero. The discussion is limited to operations on a single degree of freedom for a single solution vector. Extension to multiple degrees of freedom and solution vectors is a simple matter of substituting matrix operations for vector operations.

The total modal response,  $U$ , at a degree of freedom can be found by summing the individual modal responses.

$$U = [rphi]\{uh\} \quad \text{Eq. 3-24}$$

In [Eq. 3-24](#),  $[rphi]$  is a real one row by  $h$  column physical response data recovery matrix and  $\{uh\}$  is a complex (real for transient)  $h$  row by one column modal solution vector. The “ $h$ ” size represents the number of natural modes that are used in the solution process. The total response,  $U$ , is a complex number with real and imaginary terms. It can be represented in polar format (magnitude and phase angle) with some simple arithmetic manipulation. Now each of the individual modal contributions ( $rphi \cdot uh$ ) in the sum can be represented in complex and polar formats also. The natural mode with the largest individual contribution to the magnitude can be determined by scanning the magnitudes of each of the individual modal responses. The magnitude of the individual modal response can be *projected* onto the *magnitude* of total response. The *relative phase* angle of the individual modal response with respect to the phase angle of the total response can be calculated. Each individual modal response projected magnitude can be divided by the total response magnitude to obtain the *fraction* of the total response that is contributed by each mode. A *scaled modal response magnitude* can be calculated by dividing each modal response projected magnitude by the largest modal response magnitude. All of these different calculated quantities are present in the printed output as shown in [Figure 3-1](#).

## Inputs

The MCFRACTION Case Control command is designed to give the user new controls over the amount and ordering of the produced modal contribution fraction data. The general format of the command is:

$$\text{MCFRACTION}([STRUCTURE], [\text{PRINT, PUNCH}], [\text{REAL or IMAG}], [(SORT = \text{sorttype})],$$

$$[\text{KEY} = \frac{FRACTION}{\text{sortitem}}], [\text{ITEMS} = \frac{FRACTION}{\text{ALL (itemlist)}}] [\text{SOLUTION} = \frac{ALL}{m \text{ NONE}}] [\text{FILTER} = \frac{0.001}{\text{fratio}}]$$

$$[\text{NULL} = \frac{12}{\text{ipowr}}] = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

The complete description of the MCFRACTION command can be found in “[Statements, Commands, Entries, and Parameters](#)” on page 621. The SORT and KEY keywords are used to control sorting operations. The printed output can be sorted in one of several ways using any of the fraction views as a key. The default sorting order is in the order of the natural frequency (or mode id). The data can also be filtered to remove insignificant terms before it is printed. The SOLUTION keyword selects a subset of the available solutions for output. Any or all of the different fraction views can be sent to the punch file.

The SET Case Control command has been enhanced to recognize and process a new format for use with the MCFRACTION Case Control command. This new format allows the analyst to specify a set of grid point and component code combinations. An example of the new format is “SET 1000 = 917/T3, 85/R2, 127016/T1” which demonstrates the general input format of grid ID and component code separated by a slash, much like the manner in which XY-PLOT command grid point entities are defined. No THRU ranges are permitted.

## Outputs

The MCFRACTION Case Control command can cause generation of both printed and punched output. The output data block is OMCFRAC. Items computed and available for output are shown in [Table 3-1](#).

**Table 3-1 MCFRACTION Item List Descriptions**

Item Identifier	Description
RESPONSE	each mode's response at each degree of freedom selected
PROJECTION	projection of modal response on solution
FRACTION	fraction of total displacement per mode (PROJECTION divided by total)
SCALED	scaled magnitudes (PROJECTION divided by largest term in FRACTION)
MODEDISP	modal displacements (complex solution at each dof by mode number)
MODERESP	modal response for each mode (polar format wrt total displacement)

**Figure 3-1** is an example of the modal contribution fraction output produced in a modal frequency response solution. The heading lines identify the grid point and component, total response in both real/imaginary and magnitude/phase formats, loading condition information, maximum response contribution for a mode and the mode id, sorting and filtering information. Then, for each natural mode, a tabular listing of the various views of modal contribution data is presented. The table contains the response of the mode, in real/imaginary and magnitude phase representations, the projection magnitude, the relative phase angle, the modal fraction value, and the scaled response magnitude.

MODAL CONTRIBUTION FRACTIONS								
GRID POINT = 101/T3, TOTAL RESPONSE (R/I) = -1.69227E+00 / 3.65119E-03, (M/P) = 1.69227E+00 / 179.88								
LOAD FREQUENCY = 1.00000E+00, (SUBCASE 1, DLOAD = 15)								
MAXIMUM MODAL RESP = 1.69002E+00 FOR MODE ID = 2, SORTKEY = FRACTION, SORT = ABS VALUE ASCENDING, FILTER = 1.00000E-03								
MODE	NATURAL	MODAL RESPONSE		MODAL RESPONSE		PROJECTION	REL.	MODAL
SCALED RESPONSE								
ID	FREQ (HZ)	REAL	IMAGINARY	MAGNITUDE	PHASE	MAGNITUDE	PHASE	FRACTION
MAGNITUDE								
03	3 3.17429E+01	1.42610E-02	-8.99426E-06	1.42610E-02	359.96	-1.42610E-02	180.09	-8.42712E-
	-8.43836E-03							
	7 7.63429E+01	-1.65163E-02	4.32763E-06	1.65163E-02	179.98	1.65163E-02	0.11	9.75984E-
03	9.77286E-03							
	2 9.35245E+00	-1.69001E+00	3.65586E-03	1.69002E+00	179.88	1.69002E+00	0.00	9.98667E-
01	1.00000E+00							

**Figure 3-1 Sample Modal Frequency Response Solution Output**

The headings in **Figure 3-1** reflect the output generated in modal frequency response. For modal transient response, only real (as opposed to complex) numbers are generated and the table is reduced to that shown in **Figure 3-2**.

MODAL CONTRIBUTION FRACTIONS				
GRID POINT = 101/T3, TOTAL RESPONSE = -3.73471E-08, MAGNITUDE = 3.73471E-08				
TIME STEP = 1.00000E-02, (SUBCASE 1, DLOAD = 15)				
MAXIMUM MODAL RESP = 3.30842E-08 FOR MODE ID = 2, SORTKEY = SCALED , SORT = ABS VALUE ASCENDING, FILTER = 1.00000E-03				
MODE ID	NATURAL FREQ. (HZ)	MODAL RESPONSE	MODAL FRACTION	SCALED MAGNITUDE
1	5.44479E+00	-2.16034E-18	5.78449E-11	-6.52984E-11
10	4.19839E+02	-3.03061E-11	8.11470E-04	-9.16029E-04
3	3.17429E+01	2.69084E-09	-7.20495E-02	8.13333E-02
7	7.63429E+01	-6.92351E-09	1.85383E-01	-2.09270E-01
2	9.35245E+00	-3.30842E-08	8.85855E-01	-1.00000E+00

Figure 3-2 Sample Modal Transient Response Solution Output

Guidelines and Limitations

1. The MCFRACTION command is available in modal frequency response (SOL 111), modal transient response (SOL 112) and modal complex eigenvalue analysis (SOL 110) solution sequences only. If superelements are used, its use is restricted to residual structure data recovery operations only.
2. The modal contribution fraction computation is intended for a small sub-set of the degrees of freedom present in the model. The calculations are limited by the amount of memory available to store the physical response recovery matrix. Its columns represent a response for each mode and its rows each represent one of the grid point components for which a physical solution is requested. This matrix must fit in the available memory.
3. Only the STRUCTURE option is currently supported.
4. Both PRINT and PUNCH may be requested.
5. Printed output includes results for all of the data items described in [Table 3-1](#).
6. Punched output includes results for only the data items selected by the ITEMS keyword.
7. Modal Contribution Fractions are sorted by increasing order of mode number unless the SORT keyword specifies a particular sorting order. If a sorting order is specified, the KEY keyword selects the particular data item in the printed results tabular output listing that is sorted.
8. The SOLUTION keyword can be used to select a subset of the available solutions. If SET m is specified, the items in the SET list are forcing frequency values, time step values, or complex eigenvalue mode numbers depending upon the solution sequence used.



9. The FILTER keyword specifies a filter ratio value that is used to limit the amount of printed output produced. It applies to the data item selected by the KEY keyword if it is specified. If no KEY keyword is present, the default value of KEY=FRACTION will be used. The maximum value for the selected data item across all natural modes is determined. If the ratio of the data item value to the maximum data item value is less than fratio for any natural mode, no output for that natural mode is produced.
10. The NULL keyword can be used to establish the null response threshold value. If the magnitude of the total response at a selected grid point component is less than  $1.0 \times 10^{-\text{ipowr}}$ , no modal contribution fraction output is generated for that degree of freedom.
11. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data have no meaning. Furthermore, projections of responses onto the total response are simply the individual modal contribution to the total response at a degree of freedom. Thus, the only items available for output are the individual modal response magnitude (PROJECTION), the modal fraction (FRACTION) and the scaled response magnitude (SCALED). Selection of any of the other items from the item list of [Table 3-1](#) causes selection of the modal response magnitude (PROJECTION) item.

## Example

A simple model demonstrates the usage of the MCFRACTION Case Control command. The model input data used to demonstrate this new feature is the same as that used to demonstrate usage of the modconta DMAP alter in the SSS alter library. The basic difference is the removal of the include statement for the SSS alter in the executive section and insertion of the MCFRACTION command in the Case Control Section. Three example problems are available, one for each of the three supported solution sequences. Example mcf111.dat demonstrates usage in a modal frequency response solution; mcf110.dat demonstrates usage in a modal complex eigenvalue response solution; and mcf112.dat demonstrates usage in a modal transient response solution. Note that mcf111.dat parallels the existing modcont1.dat example in the SSS alter library. Likewise, mcf110.dat parallels modcont2.dat. The mcf112.dat file is new for this version. However, it is very similar to the other two examples.

In all of these examples, the MCFRACTION Case Control command is introduced to request calculation and output of the modal contribution fraction data. Each example problem uses different options to sort and filter the output.

## Model Description

The model used for this example is very simple as the intent is to focus on the Case Control command input. It is the double-headed flyswatter model consisting of 67 CQUAD4 shell elements and 104 grid points cantilevered at the base. The input file for only the modal frequency response run only is presented below. The input files for the modal complex eigenvalue analysis and modal transient analysis runs are very similar.

## Input File

```
$
$*****
$
$Minor Enhancement A01183 - new MCFRACTION case control capability
$VERSION: 2004
$TEST DECK NAME: mcfr111.dat
$ this deck is a modification of SSS alter library deck modcont1.dat
$
$PURPOSE:
$ Case control setup to demonstrate usage of the new MCFRACTION Case
$ Control Command in Modal Frequency Response Solution
$
$DESCRIPTION:
$ An MCFRACTION case control command is used to request output of
$ modal contribution fraction data. The command demonstrates
$ 1) request for printed output.
$ 2) request for punched output of ITEMS keyword data
$ a) modal fraction results (FRACTION item)
$ b) scaled response magnitude (SCALED item)
$ 3) results for loading frequencies 1.0, 4.0 and 20.0 are requested
$ by defining and selecting SET 88765 with the SOLUTION keyword
$ 4) results for grid point 101 components T1 and T3 are requested
$ by defining and selecting SET 98765 with the MCFRACTION command
$ 5) an ABSolute valueDescending order sort of the printed data is
$ requested by the SORT keyword selecting the ABSD (absolute value
$ descending order) option
$ 6) the modal fraction item is selected as the item to be used as the
$ sorting key by specifying the KEY keyword with the FRACTION item
$ 7) the FILTER keyword sets the filter ratio to 0.01 to eliminate
$ insignificant responses from the printed output
$ 8) the NULL keyword sets the null response threshold to 1.0e-12 to
$ eliminate output for any solution vector where the total response
$ at any of the requested degrees of freedom is less than the value
$ of the threshold
$
$EXPECTED RESULTS:
$ 1) modal contribution fraction printed output should be produced for
$ grid point 101 component T3 at forcing frequencies 1.0,4.0 and 20.0
$ 2) modal fraction and scaled response punched output should be
$ produced for grid point 101 component T3 at forcing frequencies
```

```
$      1.0,4.0 and 20.0
$  3) null response summary table should contain 3 entries: grid point
$    101 component T1 for forcing frequencies 1.0,4.0 and 20.0 because
$    the total response magnitude is less than the NULL threshold
$
init master(s)
$  modecont11.dat
$
ID SE, SAMPLE PROBLEM MODAL FREQUENCY RESPONSE
SOL 111
$
$ STATIC LOADS APPLIED TO UPSTREAM SUPERELEMENTS 1 & 2
$   USED LOADSET,LSEQ, AND DLOAD TO APPLY THEM DYNAMICALLY
$
$ ADDED DAREA CARDS ON RES STR FOR BOTH LOADS
$
TIME 25
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ include '/.../sssalter/modconta.v2001'
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
CEND
TITLE = S.E. SAMPLE PROBLEM 1 - MODAL FREQ RESP
SUBTITLE = S.E. DYNAMICS - FREQ RESP - LOADS APPLIED TO S.E
LABEL = TEST OF MODAL CONTRIBUTION FRACTION OUTPUT REQUEST
PARAM,GRDPNT,1
echo = none
$
SET 888 = ALL
$
LOADSET = 10
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
SET 98765 = 101/t1,101/t3
SET 88765 = 1.0,4.0,20.0
mcfraction(structure,print,punch,solution=88765,
  sort=absd,key=fraction,filter=0.01,null=12,
  items=(fraction,scaled)) = 98765
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
set 999 = 101
disp(phase)=999
LABEL = OPPOSING LOADS
METHOD = 2
DLOAD = 15
FREQ = 100
SDAMP = 101
$
BEGIN BULK
uset1,u3,3,101
```

```

param,partfac,7
param,ddrmm,0
$
$ ADDED DAREA CARDS
$
$ DYNAMIC LOADING CARDS
$
FREQ1      100      1.      1.      19
FREQ4      100      1.      20.     .1      5
$
TABDMP1,101,CRIT,,,,,,,,+DAMP
+DAMP,0.,.01,200.,.01,ENDT
TABLED1,15
,0.,1.,200.,1.,ENDT
TABLED1,25
,0.,1.,200.,1.,ENDT
$
$          SID    DAREA      M      N      TC      TD
RLOAD1      15      100
$
$          SID    DAREA      LID
LSEQ        10      100      12
$
FORCE,12,93,,2.,0.,0.,1.
FORCE,12,104,,2.,0.,0.,-1.
$
$
$          SID    DAREA      M      N      TC      TD
RLOAD1      25      110
$
$          SID    DAREA      LID
LSEQ        10      110      22
$
FORCE,22,93,,2.,0.,0.,1.
FORCE,22,104,,2.,0.,0.,1.
EIGRL,2,,200.
$eigr1,2,,3
$
$
$ *****
$ BASIC MODEL DEFINITION - SAME FOR ALL RUNS
$ *****
$
$ FILE NAME IS MODEL.DAT
$
GRDSET,,,,,,,,6
GRID,1,,-.4,0.,0.,,123456
GRID,3,,-.4,0.9,0.
=,*2,=,*,.9,==
=1
GRID,2,.,.4,0.,0.,,123456
GRID,4,.,.4,0.9,0.
=,*2,=,*,.9,==
=1
GRID,9,,-3.6,3.6,0.

```

```
.  
.   
.   
$   
$ ELEMENTS   
$   
CQUAD4,1,1,1,2,4,3   
=,*1,=,*2,*2,*2,*2   
=1   
CQUAD4,4,1,7,8,14,13   
CQUAD4,6,1,9,10,20,19   
=,*1,=,*1,*1,*1,*1   
=2   
MAT1,1,30.+6,,.3,.283   
PARAM,WTMASS,.00259   
PARAM,AUTOSPC,YES   
PSHELL,1,1,.05,1,,1   
$   
$ *****   
$ END OF BASIC MODEL DEFINITION   
$ *****   
ENDDATA
```

## 3.11 Enhancements to Modal Dynamic Analysis Processing

Several enhancements have been made to modal dynamic analysis processing in MSC.Nastran 2004. These are briefly described below.

### Increased Accuracy from GKAM Module

GKAM is the module that generates the modal matrices required for the solution of modal dynamic analysis equations. Prior to MSC.Nastran 2004, calculations in GKAM involving modal matrix generation used only single precision arithmetic. Starting with MSC.Nastran 2004, these calculations are performed in machine precision, thereby increasing the accuracy on double precision machines. In order to facilitate these calculations, a machine precision eigenvalue matrix (generated by the READ module) is provided as an additional input (10<sup>th</sup> input data block) to GKAM.

### Increased Accuracy from FRRD1 Module

FRRD1 is the module that performs frequency response calculations. Normally, the calculations in FRRD1 involve the solution of coupled equations of motion. However, for a modal approach with no direct input matrices and any damping being only modal damping, the calculations involve the solution of uncoupled equations of motion. Prior to MSC.Nastran 2004, the calculations for the uncoupled path were performed using only single precision arithmetic in contrast to the coupled path wherein the calculations were performed in machine precision. Starting with MSC.Nastran 2004, the uncoupled path calculations are also performed in machine precision. This will enhance the accuracy of uncoupled modal solutions on double precision machines, thereby matching the accuracy available until now only from the coupled path.

### Eigenvector Matrix Output from GKAM Module

Prior to MSC.Nastran 2004, GKAM always generated an output eigenvector matrix (4<sup>th</sup> output data block) from the input eigenvector matrix. This output matrix will be different from the input matrix only when one or the other (or both) of the following two circumstances exists:

- The user has selected a subset of the computed modes for use in the modal dynamic analysis (via the use of parameters LMODES/LMODESFL, LFREQ/LFREQFL, HFREQ/HFREQFL or the MODESELECT Case Control command).
- There are extra points in the model.

Most problems, however, do not involve either of the above circumstances. In such cases, the output eigenvector matrix is identical to the input eigenvector matrix and there is no reason for generating an output matrix that is distinct and separate from the input matrix. Accordingly, starting with MSC.Nastran 2004, GKAM will generate the output matrix *only* if either (or both) of the above circumstances exists. Otherwise, GKAM will *not* generate the output matrix and the program will, instead, regard the output eigenvector matrix as being equivalent to the input eigenvector matrix.

The above enhancement is more than of just academic interest, but is of practical significance since it can result in significantly reduced disc space requirements in very large problems with a large number of modes.

## New Eigenvalue Table Output from GKAM Module

Normally, all of the computed modes are used in a modal dynamic analysis (SOL 110, 111, 112, 145, 146 and 200). However, if the MODESELECT Case Control command has been employed or one or more of the parameters LMODES/LMODESFL, LFREQ/LFREQFL or HFREQ/HFREQFL has been specified, then the number of modes actually used in the modal dynamic analysis may be a subset of the computed modes. In such cases, starting with MSC.Nastran V2004, a new eigenvalue table indicating the actual modes used in the analysis is generated (the 7<sup>th</sup> output data block) by GKAM and is also printed in the output. (This output is also preceded by User Information Message 7588 or 7589, indicating that a subset of the computed modes is being used in the analysis.) Users should find this feature very helpful.

Note that this new eigenvalue table is generated *only* if a subset of the computed modes is selected for use in the modal dynamic analysis

## New Partitioning Vector Output from GKAM Module

Starting with MSC.Nastran 2004, a new partitioning vector is output (the 6<sup>th</sup> output data block) by GKAM if the number of modes selected for the modal dynamic analysis is a subset of the total modes computed. This partitioning vector can be used to obtain the selected eigenvectors from the computed eigenvectors and will be helpful to DMAP programmers who may want to use it in their DMAP applications. Note, however, that this partitioning vector, when it is generated, does *not* take into account the presence of any extra points in the model that may affect the size of the output eigenvector matrix mentioned earlier.

## 3.12 MAX/MIN/RMS Methodology in Data Recovery

### Introduction

The Max/Min/RMS Methodology Project has introduced and modernized data recovery in MSC.Nastran. Data Recovery has traditionally been performed in what is termed “sort1”, or all grid or elements for a specific environment, such as time. This project has introduced the concept of “sort2” processing, or, all times for a specific grid or element when performing data recovery. Along with this augmentation of processing path, the concept of monitoring component(s) for an entry without the requirement for module output has been introduced to data recovery in various solution sequences, such as the Modal Transient solution sequence, which can also be performed using the “sort2” processing methodology. Other projects impacts concern the unloading of the input/output data block locations for the SDR2 module, which allow for the module to recovery element stresses, forces and strains during a single execution.

For the “sort2” data recovery path, the DMAP support is limited to the Modal Transient solution (SOL 112). The SDR2 module has been changed to also allow for complex data recovery in “sort2” to be later used when the scope of this processing path is more fully understood.

### Benefits

The primary benefit of the MAXMIN operation is to allow for rapid identification of the grids and elements with high response values. The flexible syntax of the MAXMIN(DEF) Case Control entry allows grouping of operations to further reduce the number of components monitored. Because these operations are performed during the standard data recovery function, second computational pass is unnecessary, thus saving computer operations and storage requirements. The commands have been developed to avoid the standard data recovery operations unless downstream operations require the information for intermediate durations. The FULL operation on the MAXMIN(DEF) entry, recovers the complete grid or element entry which remains in the final response table.

A major change in data recovery strategy is the introduction of a “sort2” processing sequence. Prior to MSC.Nastran 2004, data recovery was performed using only “sort1” recovery techniques. The output tables produced were then transposed to form the “sort2” order, that is, for time or frequency domain recovery of grids or element responses. This transpose operation, although efficient, consumed computer resources, most notably, was the disk storage requirements. During the new currently released “sort2” technique in Modal Transient, the matrices are transposed in modal



space before data recovery is performed. This eliminates the “sort1” step. However, when BAR elements with intermediate station data recovery is needed, the transpose to “sort1” is still required. Be aware that the “op2” tapes used in passing data to the post processor may also require transpose operations due to their current interface limitations. The param,post,0 operation, which is the default option for MSC.Patran, is sort independent and does not require the intermediate step.

## Theory

This project introduces new Case Control entries and a new output table for the standard print and punch files. The new (default) punch must be used because the Nastran Data Definition Language (NDDL) is needed to interpret the table entries. Because the new output data block, generically named OMM, is mostly definable by the NDDL, it can be operated upon by the standard OUTPUT2 formatting function. Note that usage of the FULL action keyword on the **MAXMIN(DEF)** directive will create a data block which is currently not heterogeneously translatable due to deficiencies in the NDDL language.

The new output is sequenced within the print file before the standard data recovery of grid displacements and elements. The standard output heading is:

```
M A X / M I N      S U M M A R Y      (   xxxxxx   )
```

where xxxxx describes the class being monitored, such as displacements for grids or element names for stresses, forces or strains.

Using the FULL action keyword causes additional grid or element information to be accumulated in the OMM data block for the retained entries meeting the monitoring criteria.

## Inputs

All directives that select MAXMIN operations are located within the Case Control Section.

## Case Control

The entry **MAXMIN(DEF)** must appear before encountering any **SUBCASE** entry. Within a **SUBCASE** definition, additional entries of the form **MAXMIN(GRID)**, **MAXMIN(ELEM)**, or **MAXMIN(BOTH)** activate the operation for the SUBCASE during data recovery. Both entries are necessary for any operation to occur.

The **MAXMIN(DEF)** entry can be sub-divided into four definitional areas. Multiple **MAXMIN(DEF)** entries can be present in the Case Control section. It is recommended that only one be supplied for a class, although more than one class can be supplied on one entry. Multiple class action selection could result in formulation of an accumulative output domain.

```
MAXMIN(DEF)    class(es)  <element type(s)>  component(s)  action(s)
```

The grid classes are: **DISPLACEMENT**, **VELOCITY**, **ACCELERATION**, **OLOAD**, **SPCF** (Single Point Constraint Forces), **MPCF** (Multi-Point Constraint Forces), and **GPKE** (Grid Point Kinetic Energy). The element classes are: **STRESS**, **FORCE** and **STRAIN**. When an element class is selected, the “element types” area is required. Element types that share the same recovery components can be entered on a single **MAXMIN(DEF)** entry.

The component area is always required and can either match NDDL item descriptions or the item codes presented in “[Item Codes](#)” on page 711 of the *MSC.Nastran Quick Reference Guide*. Note that components can utilize grouping features, where the grouped components are compared to each other before they are compared to the other action requested. When an element type has multiple recovery positions, an example would be stresses for the HEXA element, where data recovery occurs at the center and vertices, these locations can be limited or grouped by keyword utilization. These operators are **/CENTER** (*default*), **/ALL**, or **/GROUP** for the repeating component locations; and **/ENDS** for the BEAM element. An example for the selection of the first principal stress component at every data location on the HEXA would be **P1/ALL**. This would cause monitoring and performance of data recovery for each of the monitored nine locations per element.

Action keywords are optional with default values. These keywords consist of **MAXALG**, **MINALG**, **ABSOLUTE**, and **ALL** that control the number of items retained during the component evaluations. When none are specified, the default is **ALL** with a retained list of five per component. When **MAXALG**, **MINALG** or **ABSOLUTE** actions are selected, the ones not referenced will default to a retained set of zero. To override the number of retained values, an integer value of 1 to 25 can be associated with the action by either an equal delimiter or by encapsulating the value in parenthesis, for example, **MAXA=5** or **MINA(2)**.

The action keyword **RMS**, causes the root mean squared value to be calculated over the domain, but may not be meaningful depending upon the interaction with other component actions such as grouping. The action keywords of **BRIEF** (*default*) and **FULL** controls the amount of presented recovery information. The **BRIEF** action only presents the components selected over the retained depth associated with the selected

items in the element or grid set. The FULL action causes the grid or element entry to be retained. Those in the retained list will be shown in their standard data recovery format. See output section for illustrations.

The last action keyword concerns the coordinate system in which the comparisons are performed. Currently this action is only available for grid recovery classes. The action keyword CID={GLOBAL, BASIC, cid} is used to override the default. The default of the grid comparison is different depending upon the SDR2 module sort processing. When the matrices are input in SORT1 order, then the comparison is performed in the BASIC reference system with the “cid” used to select an alternative. GLOBAL is not supported. When the matrices are input in SORT2 order, the transpose of SORT1, then the comparisons are performed in GLOBAL, with either BASIC or “cid” as alternative options. Note, when FULL is requested along with a CID= action, then the entries displayed are in the CID selected system. Standard data recovery is unaffected and still occurs in the global reference frame.

The MAX/MIN definitions are acted upon only if selected by the additional **SUBCASE** entries to select the domain in classical Case Control syntax.

```
MAXMIN( GRID, [ { PRINT/NOPRINT } , PUNCH ] ) = { ALL, SET, NONE )
MAXMIN( ELEM, [ { PRINT/NOPRINT } , PUNCH ] , [ VONMISES/SHEAR ] , [ STRCUR/FIBER ] , [ C
ENTER/BILIN|etc ] ) = { ALL, SET, NONE )
MAXMIN( BOTH, [ { PRINT/NOPRINT } , PUNCH ] , [ VONMISES/SHEAR ] , [ STRCUR/FIBER ] , [ C
ENTER/BILIN|etc ] ) = { ALL, SET, NONE )
```

Since the **MAXMIN(ELEM..)** directives act upon all element classes defined within the **MAXMIN(DEF)** section, the keywords modifying the data recovery operations for STRESS and STRAIN are also included since the primary Case Control entries may not be present. The invariant selection is independent between the MAXMIN and primary data recovery entry. When element strains recovery is selected, the STRAIN entry when present in the SUBCASE supercedes the MAXMIN selection concerning fiber or curvature locations for the plate elements. This is also true for the QUAD4 corner selection concerning the method for extrapolation to the vertices.

## Outputs

The OMM data block conforms to standard OFP (Output File Processor) format. That is it contains pairs of records. The first of the pair denoted as the IDENT or HEADER is 146 words in length and describes the following data record. The data record contains entries that are eleven words in length and are (perceivably) always in sort1 real format. The OMM data block can be identified as OFP output class 38. Care must be exercised interpreting the data record because item values in the entry change format classes depending upon settings within the IDENT record. The NDDL contains a full description of the control values. Another significant difference to the

OMM data block when compared to other OFP tables is that with the FULL action on the MAXMIN(DEF) entry, all other table classes can be contained as record pairs within the OMM data block.

The output examples presented are intended to assist with column interpretation within analysis approach and output classes. The two examples were created from a Modal Transient solution sequence. The OMM table identification line contains the class name of the values presented in the table. The first table indicates grid class response acceleration, and the second represents an element response, in this case a BAR element and the component name indicating a stress response. For these two tables, the label of column one changes from POINT to ELEMENT depending upon the information contained in the table. The CID or Coordinate System Identification is always present, although currently meaningful for grid recovery. The value “-1” indicates element 0 basic and positive values, the coordinate system used to measure response. The OPERATION column contains two pieces of information from the data entry, first the measure, MAX (maximum algebraic), MIN (minimum algebraic) and ABS (absolute value). The second value is the retention depth varying from 1 to 25. The next column is labeled COMPONENT and represents the keyword for the item selected from the NDDL description in the response table. Additional keywords such as VONMISES that override the NDDL description for this release. The next column is variable in column label and format. For our current example, the column header is TIME and the entry item format is real. When Linear Statics is selected as the analysis approach, the column label would be SUBCASE and the entry item format integer. The next column, VALUE, represents the real value of the monitored response quantity. These columns form the minimum set of information presented as basic output.

M A X / M I N   S U M M A R Y   ( A C C E L E R A T I O N )

POINT	CID	OPERATION	COMPONENT	TIME	VALUE
1011	0	MIN	1 R2	9.999999E-02	-8.637509E-01
1011	0		2	9.000000E-02	-5.546898E-01
1011	0		3	8.000000E-02	-2.548527E-01
1011	0		4	7.000000E-02	2.930702E-02
1011	0		5	6.000000E-02	2.917157E-01
1011	0	ABS	1 R2	0.000000E+00	1.140825E+00
1011	0		2	1.000000E-02	1.103539E+00
1011	0		3	2.000000E-02	1.021089E+00
1011	0		4	3.000000E-02	8.954600E-01
1011	0		5	9.999999E-02	8.637509E-01

M A X / M I N   S U M M A R Y   ( B A R )							
ELEMENT	CID	OPERATION	COMPONENT		TIME	VALUE	RMS
1001	-1	MAX	1	SX1B	9.999999E-02	2.695622E-02	1.362836E-02
1001	-1		2	SX1B	9.000000E-02	2.279591E-02	
1001	-1		3	SX1B	8.000000E-02	1.871261E-02	
1001	-1		4	SX1B	7.000000E-02	1.481509E-02	
1001	-1		5	SX1B	6.000000E-02	1.120520E-02	
1001	-1	MIN	1	SX1A	9.999999E-02	-6.621534E-02	1.362836E-02
1001	-1		2	SX1A	9.000000E-02	-5.536992E-02	
1001	-1		3	SX1A	8.000000E-02	-4.501026E-02	
1001	-1		4	SX1A	7.000000E-02	-3.533644E-02	
1001	-1		5	SX1A	6.000000E-02	-2.653448E-02	

The above table illustrates that two optional columns can appear after the VALUE column. The one with RMS as the label will always appear as the last column. The other additional column, currently has a label of either GRID or SD (Station Distance), and appears between VALUE and RMS when the element has multiple recovery positions, such as the HEXA element. When the GRID columns appears, it contains two different formats depending upon the component actions selected on the MAXMIN(DEF) entry. When the OPERATION column has a depth measure of “1”, the identification will appear in the GRID column. The center of the element has a grid designation of “0”. For higher depth values, a blank will appear in the GRID column when the /ALL option was selected. When the column item is non-blank, then the /GROUP option was selected and the position within the element is being identified. The SD column label indicates that the data entry contains real values for the station distance measure used for the BEAM element.

## Guidelines and Limitations

The MAXMIN capability was fundamentally designed for the Modal Transient solution sequence employing a sort2 processing methodology. This placed a certain perspective upon the global/local orientation of grids and elements during the monitoring operations. Because element or grid orientation does not change within the time domain, global directions requiring no re-orientation were perceived for the standard processing methodology. The grids were enhanced to support other directions and are necessary when the sort1 processing requirement was established. The current element orientation technology is limited in sort1, especially considering the plate elements. Considering the Transient Solution again, the placement and number of the MAXMIN(DEF) are geared to time domain processing. When processing Direct Transient, the sort1 path is used. The primary consideration here is the computational costs for performing the transpose of the “displacement” matrix at the solution set.

Since the MAXMIN operations are performed internally to the SDR2 module, effects of BAR distributed loads are currently ignored and lump end recovery is currently available. This restriction is also true for the BEAM and BEND elements. The MAXMIN operations are performed for SDR2 data recovery which currently does not include Layered Composites, Element Energies, or Grid Point Forces.

The sort2 Linear Statics solution sequence is currently not supported.

Currently there is no support for complex data recovery for either grids or element for MAXMIN operations.

## Examples

### Example 1: Modal Transient Model of a Solid (mmhex04.dat)

The Case Control Section contains the MAXMIN(DEF) and MAXMIN(ELEM) entries. The specific MAXMIN(DEF) entry has been selected to monitor stresses for the HEXA element and examine the Principal Stresses. Recovery locations are to be monitored but a single value retained with its associated vertex grid identification. All possible monitoring actions are to occur to a retained depth value of 10. The parenthesis usage surrounding the components collapses all of the principals to a single value per element. The MAXMIN(ELEM) references a set of one element.

```
MAXMIN(DEF) STRESS HEXA ( P1/group p2/group p3/group ) all=10
TITLE = modal transient with solid elements
SUBCASE 1
  method 1
  TSTEP = 1
  SPC = 1
  DLOAD = 2
  set 200 = 1
  maxmin(elem) = 200
```

The response recovered information indicated that for the algebraic maximums that the P1 direction over time was the dominating value although the grid location was not the same for all retained values. The algebraic minimums switched to the P2 direction, but had a similar pattern concerning vertex location. The absolute value monitor, however, switched direction and vertex location over the time range. The coordinate system of element comparison was the basic reference frame because that selection can be accomplished by setting on the PSOLID entry.

SUBCASE 1

## M A X / M I N S U M M A R Y ( H E X A )

ELEMENT	CID	OPERATION	COMPONENT	TIME	VALUE	GRID
1	0	MAX	1 P1	6.000000E-04	3.211954E+02	11
1	0		2 P1	9.999999E-04	2.747574E+02	23
1	0		3 P1	7.000000E-04	2.726083E+02	11
1	0		4 P1	5.000000E-04	2.719700E+02	11
1	0		5 P1	4.000000E-04	1.485749E+02	11
1	0		6 P1	8.000000E-04	1.052457E+02	11
1	0		7 P1	9.000000E-04	9.128755E+01	13
1	0		8 P1	3.000000E-04	2.646395E+01	11
1	0		9 P1	2.000000E-04	1.693706E+01	13
1	0		10 P1	1.000000E-04	4.520476E+00	21
1	0	MIN	1 P2	6.000000E-04	-3.211504E+02	23
1	0		2 P2	9.999999E-04	-2.750641E+02	11
1	0		3 P2	7.000000E-04	-2.724407E+02	23
1	0		4 P2	5.000000E-04	-2.722952E+02	23
1	0		5 P2	4.000000E-04	-1.487349E+02	23
1	0		6 P2	8.000000E-04	-1.050558E+02	23
1	0		7 P2	9.000000E-04	-9.140302E+01	21
1	0		8 P2	3.000000E-04	-2.612224E+01	23
1	0		9 P2	2.000000E-04	-1.695657E+01	21
1	0		10 P2	1.000000E-04	-4.504962E+00	23
1	0	ABS	1 P1	6.000000E-04	3.211954E+02	11
1	0		2 P2	9.999999E-04	2.750641E+02	11
1	0		3 P1	7.000000E-04	2.726083E+02	11
1	0		4 P2	5.000000E-04	2.722952E+02	23
1	0		5 P2	4.000000E-04	1.487349E+02	23
1	0		6 P1	8.000000E-04	1.052457E+02	11
1	0		7 P2	9.000000E-04	9.140302E+01	21
1	0		8 P1	3.000000E-04	2.646395E+01	11
1	0		9 P2	2.000000E-04	1.695657E+01	21
1	0		10 P1	1.000000E-04	4.520476E+00	21

**Example 2: Linear Static Model of a Quarter Plate (mmq401.dat)**

This example of a Quarter Plate in Linear Statics demonstrates the usage of three MAXMIN(DEF) entries and different options selected in the MAXMIN(ELEM) entries within the Case Control Section. The usage of FULL produces limited data recovery in standard formats encapsulated within the MAXMIN output. Only limited response information is displayed for illustration purposes.

```

MAXMIN(DEF) STRAIN QUAD4 EMAX1 EMAX2 MAXA MINA FULL
MAXMIN(DEF) STRESS QUAD4 SMAX1 SMAX2 MAXA MINA FULL
maxmin(def) disp t1 t2 maxa mina full cid=basic
MAXLINES=1000000
echo=none
SPC=1
set 100 = 10 thru 14
SUBCASE 1001
TITLE=Only MAXMIN Shear Invariant, strcur
LOAD=101
maxmin(elem,shear) = all

```

```

DISP=100
maxmin(grid) = 100
SUBCASE 1002
TITLE=Only MAXMIN Von Mises Invariant, fiber
LOAD=101
maxmin(elem,fiber) = all
SUBCASE 1003
TITLE=MAXMIN Shear Stress Von Mises, fiber
LOAD=101
maxmin(elem,shear) = all
STRESS=ALL
strain(fiber)=all
BEGIN BULK

```

The recovery for the displacements within the MAXMIN output section was within the basic reference system. One should observe here that the POINT identification is in retained sequence and that collation order is by operational depth. Also the identification column has become SUBCASE due to the solution approach.

#### MAX / MIN SUMMARY ( DISPLACEMENTS )

POINT	CID	OPERATION	COMPONENT	SUBCASE	VALUE
14	0	MAX	1 T1	1001	1.993413E-03
13	0		2	1001	1.532535E-03
12	0		3	1001	1.186146E-03
11	0		4	1001	1.104745E-03
10	0		5	1001	1.092352E-03
10	0	MIN	1 T1	1001	1.092352E-03
11	0		2	1001	1.104745E-03
12	0		3	1001	1.186146E-03
13	0		4	1001	1.532535E-03
14	0		5	1001	1.993413E-03
14	0	MAX	1 T2	1001	-1.716866E-05
11	0		2	1001	-1.911883E-05
12	0		3	1001	-2.190853E-05
10	0		4	1001	-3.123875E-05
13	0		5	1001	-4.483166E-05
13	0	MIN	1 T2	1001	-4.483166E-05
10	0		2	1001	-3.123875E-05
12	0		3	1001	-2.190853E-05
11	0		4	1001	-1.911883E-05
14	0		5	1001	-1.716866E-05

The standard output recovery for sort1 real is used for the FULL formatting. This output is recovered in the basic reference system due the request upon the MAXMIN(DEF) entry. The normal POINT collation order is maintained and the entries correspond to the retained list from the MAXMIN operation section.



SUBCASE 1001							
D I S P L A C E M E N T   V E C T O R							
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
10	G	1.092352E-03	-3.123875E-05	0.0	0.0	0.0	0.0
11	G	1.104745E-03	-1.911883E-05	0.0	0.0	0.0	0.0
12	G	1.186146E-03	-2.190853E-05	0.0	0.0	0.0	0.0
13	G	1.532535E-03	-4.483166E-05	0.0	0.0	0.0	0.0
14	G	1.993413E-03	-1.716866E-05	0.0	0.0	0.0	0.0

Once the Grid Point responses have been reported, the element requestd are then presented. Because in Linear Statics, the element reference system may not be consistent, it is suggested that the invariant be used to determine the ordering for two dimensional elements. The results presented are for the first SUBCASE which the SHEAR invariant was selected. Again note the element identification ordering is by component and depth.

M A X / M I N   S U M M A R Y   ( Q U A D 4 )						
ELEMENT	CID	OPERATION	COMPONENT	SUBCASE	VALUE	
37	-1	MAX	1	SMAX1	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	SMAX1	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03
37	-1	MAX	1	SMAX2	1001	1.404862E+04
31	-1		2		1001	1.297107E+04
25	-1		3		1001	1.003949E+04
32	-1		4		1001	9.677195E+03
26	-1		5		1001	8.383499E+03
13	-1	MIN	1	SMAX2	1001	1.699686E+03
301	-1		2		1001	2.126532E+03
401	-1		3		1001	2.584472E+03
201	-1		4		1001	2.834442E+03
14	-1		5		1001	4.062775E+03

The FULL action on the MAXMIN(DEF) then interweaves the standard element recovery for the retained list.

SUBCASE 1001

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)									
ELEMENT ID.	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			MAX SHEAR	
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
13	-5.000000E-02	-3.104902E+02	-1.519392E+03	-1.588575E+03	-34.5841	7.847449E+02	-2.614627E+03	1.699686E+03	
	5.000000E-02	-3.104902E+02	-1.519392E+03	-1.588575E+03	-34.5841	7.847449E+02	-2.614627E+03	1.699686E+03	
14	-5.000000E-02	2.803682E+02	4.815094E+02	-4.061530E+03	-45.7092	4.443714E+03	-3.681836E+03	4.062775E+03	
	5.000000E-02	2.803682E+02	4.815094E+02	-4.061530E+03	-45.7092	4.443714E+03	-3.681836E+03	4.062775E+03	
25	-5.000000E-02	1.260785E+03	2.108877E+04	-1.582418E+03	-85.4656	2.121426E+04	1.135291E+03	1.003949E+04	
	5.000000E-02	1.260785E+03	2.108877E+04	-1.582418E+03	-85.4656	2.121426E+04	1.135291E+03	1.003949E+04	
26	-5.000000E-02	2.543391E+03	1.722717E+04	-4.047183E+03	-75.5672	1.826878E+04	1.501783E+03	8.383499E+03	
	5.000000E-02	2.543391E+03	1.722717E+04	-4.047183E+03	-75.5672	1.826878E+04	1.501783E+03	8.383499E+03	
31	-5.000000E-02	1.224244E+03	2.709546E+04	-9.584824E+02	-87.8812	2.713093E+04	1.188782E+03	1.297107E+04	
	5.000000E-02	1.224244E+03	2.709546E+04	-9.584824E+02	-87.8812	2.713093E+04	1.188782E+03	1.297107E+04	
32	-5.000000E-02	2.949327E+03	2.166685E+04	-2.462046E+03	-82.6305	2.198529E+04	2.630894E+03	9.677195E+03	
	5.000000E-02	2.949327E+03	2.166685E+04	-2.462046E+03	-82.6305	2.198529E+04	2.630894E+03	9.677195E+03	
37	-5.000000E-02	1.340506E+03	2.942988E+04	-3.323971E+02	-89.3221	2.943381E+04	1.336573E+03	1.404862E+04	
	5.000000E-02	1.340506E+03	2.942988E+04	-3.323971E+02	-89.3221	2.943381E+04	1.336573E+03	1.404862E+04	
201	-5.000000E-02	-3.406158E+02	-5.742985E+03	-8.588726E+02	-8.8193	-2.073585E+02	-5.876242E+03	2.834442E+03	
	5.000000E-02	-3.406158E+02	-5.742985E+03	-8.588726E+02	-8.8193	-2.073585E+02	-5.876242E+03	2.834442E+03	
301	-5.000000E-02	9.549748E+02	-2.552619E+03	-1.202636E+03	-17.2199	1.327710E+03	-2.925354E+03	2.126532E+03	
	5.000000E-02	9.549748E+02	-2.552619E+03	-1.202636E+03	-17.2199	1.327710E+03	-2.925354E+03	2.126532E+03	
401	-5.000000E-02	3.927623E+03	-5.305154E+02	-1.307955E+03	-15.2016	4.283025E+03	-8.859182E+02	2.584472E+03	
	5.000000E-02	3.927623E+03	-5.305154E+02	-1.307955E+03	-15.2016	4.283025E+03	-8.859182E+02	2.584472E+03	

## 3.13 Massless Mechanism Identification and Control Enhancements

This capability was introduced in MSC.Nastran 2001, and is enhanced for MSC.Nastran 2004.

A massless mechanism (MM) mode is defined as a shape that causes very small strain energy when the structure is deflected in this shape. This shape also causes very little kinetic energy when the structure moves in this shape. The ratio of strain energy to kinetic energy is proportional to the eigenvalue of this mode. This result is an indeterminate quantity produced by two terms approaching zero. If applied loads excite this shape, small loads can cause very large deflections. If a real mode analysis can be forced to produce a solution in the presence of this condition, the mode produced with this shape can have any eigenvalue from positive to negative infinity. These frequencies may change drastically for small changes in the model or when run on a different computer. Eigensolution failures due to Sturm number inconsistencies are often due to the presence of massless mechanisms.

The presence of such modes reduces the reliability and repeatability of eigensolutions and dynamic analysis solutions. A method is now provided to automatically constrain massless mechanisms for eigensolutions, controlled by the parameter `PARAM,MECHFIX,AUTO`, which is turned on by default in MSC.Nastran 2001. When this method is used, it is unlikely that the eigensolution will fail because of three failed shifts, and the solution produced will be repeatable for small changes in the model, loading condition, or computer type used in the solution. The goal of this new capability is to allow the real eigensolvers to produce an eigensolution every time a proper non-zero stiffness and mass matrix are provided. The new capability will also make other eigensolution methods more reliable.

There are occasions in Version 70.7 of MSC.Nastran where the Lanczos method performs three successive shifts of the "shifted stiffness matrix" in preparation for an inverse power solution, and all shifts fail due to poor numerical conditioning of the shifted matrix. Inverse iteration is the first step in a Lanczos solution. The Lanczos method then exits with a fatal error message because it is unlikely that any other shift will produce a well-conditioned shifted matrix for this system. Similar problems can cause the xHOU methods to fail, the most likely cause of which is a massless mechanism.

Field usage of MM control found that certain models might have one or more rigid body modes improperly constrained as massless mechanisms. The method of identifying MM modes has been modified to make this unlikely in MSC.Nastran 2004. New user parameters are described at the end of this section that provide more control in this area.

This work was prototyped in an SSSALTER for MSC.Nastran 2001 (mmechfixa.v2001).

## Input

The massless mechanism feature is controlled by several parameters. The following is a list of the primary parameters along with their functions:

MECHFIX	Default = AUTO
	Controls for fixing the massless mechanism problem. The new capability is provided automatically for the default of this parameter, listed above. The new capability is executed only when the eigensolution does not provide answers because of symptoms consistent with the presence of massless mechanisms. The MMs are removed, and a second eigensolution is made. If MECHFIX is set to YES, the constraint modes are removed before attempting an eigensolution. When set to NO, the new capability is blocked, and the eigensolution uses the pre MSC.Nastran 2001 rules, i.e., three failed shifts and a fatal exit.
MECHPRT	Default = NO
	For SOL 103 only, if massless mechanisms are found, the constraint modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINT MODES, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model and a goal is to remove all massless mechanisms before starting production analysis. The number of each "mode" matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes, the output file will be large. There is no method to plot these shapes at present.

The following secondary parameters have default values that work well in most circumstances. They are provided for tuning unusual circumstances.

MAXRATIO	Default = 1.E7  This is another traditional parameter whose other functions are described in the <a href="#">MSC.Nastran Quick Reference Guide</a> . It is also the criterion for identifying massless mechanisms. A lower value is more conservative in that it is likely to identify and control more mechanisms. If blocking of this capability is desired, use PARAM,MECHFIX,NO instead of modifying this parameter. The rationale is that if the user wants to set it high enough that it never identifies a mechanism, it is better not to request massless mechanism control.
MECHFIL	Default = 1.E-6  Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constraint modes being retained.

The SHIFT1 parameter introduced in MSC.Nastran 2001 has been removed. Its function has been replaced by the following parameters:

## New Parameters for MSC.Nastran 2004

MMFIL	Default = 1.e-10  Filter value used to distinguish between MM modes and rigid body modes. A smaller value may discard rigid body modes. The default value has been effective on all problems solved to date.
NLMAX	Default = 60  The number of suspected MMs is determined from the number of high ratio messages. If this number exceeds NLMAX, the number of trial MMs is reduced to NLMAX. This avoids an expensive debug run when there may be thousands of MMs due to systematic modeling error, such as having CONM2 elements on many grid points for which structural elements have been left out through oversight. The value of this parameter may be increased on initial debug runs where many high ratio DOFs may be present, and the user prefers to see them all at once, rather than on successive runs where only a part are removed at one time.

NLMIN

Default = 10

In the case of only one or a few high ratio DOFs more MMs may be present. More trial MM vectors are used, and those that do not indicate true problems are discarded. A smaller value could be considered on a stable model undergoing small modeling changes.

## 3.14 General Enhancements for Orthogonalizing Complex Vectors

### Summary

A method of orthogonalizing vectors of any type, real or complex, is provided. An efficient path is provided for real symmetric matrices, and a general path for all types of matrices. The matrix used to weigh the vectors may be real or complex and symmetric or unsymmetric.

Vectors for complex matrices commonly come in pairs. For example, the complex eigenproblem

$$[p^2 \cdot M + p \cdot B + K] \cdot \phi = 0$$

has one set of eigenvectors  $R$ , and a second set of eigenvectors  $L$ , the so-called "left-hand eigenvectors", when the problem is transposed. If all of the matrices are symmetric  $L = R$ . Unlike the case for the real eigensolution, the eigenvectors do not necessarily diagonalize the "generalized" quantities. That is,  $L^T \cdot M \cdot R$  does not produce a diagonal matrix. However, there are applications where it would be useful to find linear combinations of these vectors such that the triple product produces an identity matrix. This is known as orthonormalizing these vectors with respect to  $M$ , or any other square matrix you select.

There are other applications such as residual vectors where a large set of orthogonalized vectors  $Ru$  and  $Lu$ , are uncoupled by definition. A few vectors  $Rc$  and  $Lc$  that are not necessarily orthogonal may be added to them. It is possible to take linear combinations of these vectors and the already-orthogonal vectors to make a new expanded set of orthogonal vectors. The cost of orthogonalizing the original set does not need to be repaid. This is known as selective orthogonalization. This technique is used in the Residual Flexibility capability, for real symmetric matrices. This Subdmap extends the capability to complex vectors and matrices, where the matrices may be unsymmetric.

The SubDMAP allows input of both the uncoupled and coupled sets of vectors. The right side vectors are appended into a matrix  $RN$ , and the left side vectors into  $LN$ . An efficient normalization scheme is used on the uncoupled vectors to normalize them, to make the coupled vectors orthogonal to the uncoupled vectors, and then to each other, so that  $RN$  and  $LN$ , the outputs of the SubDMAP, satisfy the equation.

$$LN^T \cdot M \cdot RN = I$$

If it is found that some of the coupled vectors are not linearly independent of the uncoupled vectors or the prior-orthogonalized coupled vectors the dependent vectors are discarded. The number of columns in  $RN$  is therefore the sum of the number of columns of  $RC$  and  $RU$  minus the number of discarded vectors. The number of discarded vectors is printed by the SubDMAP when requested. The uncoupled vectors, the coupled vectors, or both may be input.

At present this SubDMAP is not called from any of the solution sequences. It is provided in the Delivery Database as a convenience for the client who requested this capability. Its usage requires a DMAP alter. Demonstration problem q17538903.dat has this alter included.

## SUBDMAP CALL

```
CALL ONORM A,RU,RC,LU,LC/RN,LN/msgs/s,epso $
```

### Inputs

$A$	Square matrix of dimension $Nu$ . May be real or complex, symmetric or unsymmetric. If purged (not present) it is, in effect, replaced with an identity matrix.
$RU$	Vectors orthogonal w.r.t $A$ and $LU$ . All vectors described here have rows $Nu$ in length. May be purged when $RC$ exists.
$RC$	Vectors not orthogonal w.r.t. $A$ . May be purged when $RU$ exists.
$LU$	Vectors for which $LU^T \cdot A \cdot RU = I$ , after normalization. If purged $LU = RU$ is assumed.
$LC$	Vectors commensurate with $RC$ . If purged $LC = RC$ is assumed.

### Outputs

$LN,RN$	Vectors that meet the orthogonality conditions described above. $LN$ may be purged.
---------	---

### Parameters

Msgs	CHAR8, input, default is 'no'. No special diagnostics provided.
------	---



If 'yes', diagnostics are provided on matrix sizes, number of discarded vectors, etc.

Epso RS, output. Largest term in expression  $[LN^T \cdot A \cdot RN - I]$ , which should be computational zeros.

## Method

Find the components of the generalized matrices

$$LMRUU = LU^T \cdot A^T \cdot RU, \text{ diagonal terms only}$$

$$LMRCC = LC^T \cdot A \cdot RC, \text{ full matrices for the remaining terms}$$

$$LMRUC = LU^T \cdot A \cdot LC, \text{ cross terms}$$

$$LMRCU = LC^T \cdot A \cdot LU, \text{ cross terms}$$

Store these partitions in the system matrix

$$B = \begin{bmatrix} LMRUU & LMRUC \\ LMURC & LMRCC \end{bmatrix}$$

When  $B$  is unsymmetric the symmetric matrix  $B22 = B \cdot B^T$  is found. When symmetric,  $B22 = B \cdot BB$  is decomposed to identify nearly singular rows and columns. These are discarded from  $B$  to generate  $B11$ .

The most effective tool for orthogonalizing vectors is a Cholesky decomposition followed by a forward-only solution, available only for symmetric matrices.

Unfortunately, a Cholesky factorization is not yet available for complex symmetric matrices, so two paths are provided through the alter. For the case of  $A$ , real and symmetric and no left hand vector input, a real symmetric path is used, where matrix function modules may be used for all operations. All real arithmetic is used, and the output is real. This is called the real option.

For all other cases the complex option is used. A SubDMAP that simulates matrix operations with an element-by-element conversion of one matrix to another is used. All internal operations, and the output, are provided in complex form. While the SubDMAP is a less efficient single module equivalent, it fills in a hole in the matrix module capability. Details are given in the SubDMAP listing.

$B11$  is used to obtain a symmetric matrix  $S$ ,

$$S = 0.5 \cdot [B11 + B11^T]$$

Another set of components of  $B11$  are sought that are factors instead of parts of a sum. Let  $S$  be one of the factors, and the other an unknown factor  $X$ ,

$$B11 = X \cdot S$$

This product form allows breaking  $B22$  into two factors that are the same matrix for the real symmetric path, and almost identical matrices for the complex path. Solving for the inverse of  $X$ ,

$$XINV = S \cdot B11INV$$

$S$  is decomposed into its Cholesky factors  $S = Lc \cdot Lc^T$

Then  $X \cdot Lc \cdot Lc^T = LN1^T \cdot M \cdot RN1$

Our goal is to find  $XL$  and  $XR$  where

$$XL^T \cdot B11 \cdot XR = I$$

and  $XL = XR$  for the real symmetric path.

One possible set of choices is

$$\begin{aligned} XR &= LCinv^T \\ XL &= Xinv^T \cdot LCinv^T = Xinv^T \cdot XR \end{aligned}$$

For the real symmetric path,  $X = I$ . For a slightly unsymmetric problem,  $X$  differs only slightly from  $I$ . Plugging these values into the prior equation

$$[LCinv^T \cdot Xinv] \cdot [X \cdot Lc \cdot Lc^T] \cdot LCinv^T = I$$

by inspection, therefore these choices for  $XR$  and  $XL$  are valid.

An interesting sidelight arises when the real symmetric option is used. The standard factor of the form  $S = L \cdot D \cdot L^T$  is used instead of the Cholesky option. For all options, the Cholesky factor is built from the standard factor terms,  $Lc = L \cdot \text{sqrt}(D)$ . If  $A$  is indefinite,  $D$  has negative terms. In the real option it would be necessary to use complex numbers for the square root, and lose the real output feature. To avoid this, for the real option only,  $\text{abs}(D)$  replaces  $D$ . For the complex option, negative terms cause no problems for square root operations, and the complex form of  $D$  is used. This is equivalent to the most general form of Gram-Schmidt orthogonalization, which also can produce generalized matrices with negative 1.0 on the diagonal.

As a consequence, the generalized mass produced by real option may have negative unit terms on some diagonal terms, and positive unit terms on others. This allows all output to remain in the real domain. However, if this same system is operated on, but forced to go the complex path, the generalized mass matrix is the identity matrix, complex in form as is all other output. The orthonormalized vectors with positive real generalized mass in the real form are identical in both the real and complex output, except for the null imaginary parts in the complex form. The vectors with negative real generalized mass are all real in the real form, and equal except that all non-zero terms are in the imaginary plane in the complex form. An example problem described in the readme file demonstrates this behavior given an indefinite A matrix with two different calls to the SubDMAP, for both the real and the complex option.

One may consider using negative generalized mass of this form for the READ module. It happens that the "auto" tridiagonal methods, such as AHOU, can usually successfully generate vectors when there is an indefinite mass matrix. However, a final orthogonalization step fails on the negative square root issue. If the technique used here were added to the READ module, there would be one method that could deal with indefinite mass matrices. The LAN method will still fail, but presumably if similar techniques were used in its operation (allow negative generalized mass during iterations), it too could handle indefinite systems. That would be a larger development project.

## Example 1

Given a square matrix A, a set of arbitrary vectors RR, orthonormalize them to produce RNORM. Minimize the number of diagnostic messages.

```
ONORM A,RR,, ,/RNORM,/'no'/0.0 $
```

## Example 2

Given all of the inputs, find the outputs. Get full diagnostics.

```
TYPE PARM,,I,N,epso $
ONORM A,R1,R2,L1,L2/R,L/'yes'/s,epso $
```

## Results of Testing

The SubDMAP has been tested extensively, with some of the test problems available in the zip file. The code is reasonably fast, certainly much faster than the solvers typically used to generate complex vectors. No failures of the method have been encountered. For good accuracy, unsymmetric A matrices may require two

orthogonalization in series. An output parameter gives a measure of the error in the orthogonalization, and can be used to branch over the second orthogonalization when it is not needed. This technique is shown in the example problems.

Because all of the operations can be done in complex arithmetic, problems that can not be solved with real arithmetic, such as performing a Cholesky (square root) factorization of a non-psd matrix, can be done with the alter package. The square root of negative numbers as well as complex numbers is a defined operation in complex arithmetic.

## Limitations

The main known limitation now occurs for A matrices of the form

$$\begin{bmatrix} 0 & 1 \\ 1 & 0 \end{bmatrix}$$

The corresponding S matrix will be null, and the system will be unsolvable. Fortunately, such cases are rare.

This problem could be solved with the sparse solver, which uses an  $L \cdot T \cdot L^T$  decomposition, where T is tridiagonal, as well as a permutation matrix not shown here. We decided to use the active column solver instead because to avoid the complication of dealing with the permutation matrix. Further, the matmod option that converts its packed factor into true matrices has errors for certain types of matrices. However, this method could be implemented after some code changes are made. An SVD solution is also a possibility here.

Other limitations for efficiency could be addressed by improvements in the present modules used in the alter package, and a new module to replace the DIAGLNK SubDMAP. Designs for these enhancements can be provided for funded projects. The DIAGLNK SubDMAP provides capabilities for complex matrices that are not present in the DIAGONAL matrix. The SubDMAP user interface simulates a proposed enhancement to the DIAGONAL module in a future version.

## 3.15 Complex Eigenvalue Analysis Updates

The following enhancements have been added to the QZHESS method.

- The mathematical residual calculations, automatically executed in MSC.Nastran 2001, are now executed only at your request by setting `system(108)=8`. This will improve the performance of the QZHESS method.
- Furthermore, in MSC.Nastran 2004, QZHESS will return only the positive (principal) square roots of  $\lambda^2$ . You may still request both positive and negative roots, as was printed in MSC.Nastran 2001, by setting `system(108)=4194304`.

### ND Observance

Historically, in complex eigenvalue analysis, all the eigenvalues that were found and passed the acceptance test were printed. Starting in MSC.Nastran 2004, only ND eigenvalues will be printed. ND is set on the parent EIGC entry or its continuation entries. Both CLAN and QZHESS methods will obey the new rule.

### Output Filtering Options

Requesting the output of purely real roots (`nastran system(108)=524288`) or purely imaginary roots (`nastran system(108)=1048576`) is also an option in MSC.Nastran 2004.

## 3.16 Real Eigenvalue Analysis

The following enhancements have been added to the real eigenvalue analysis.

### Reduction Method Performance

The dynamic matrix generation phase for the reduction methods such as HOU and GIV variations has been modified to take advantage of the new sparse Cholesky factor. No user intervention is required; however, substantial performance improvement should be observed for large problems.

### Fatal/Warning Message Option

For the real Lanczos method, the job continues even when Warning Message 5407 reports the serious problem of finding spurious modes. Some users want the run terminated under this circumstance. Setting `system(317)=1`, the warning message will turn into a fatal message, and the run will terminate in MSC.Nastran 2004.

### New Eigenvalue Matrix Output From the Real Eigenvalue Analysis (READ) Module

A new matrix data block (the 5th output data block) is now routinely generated by the real eigenvalue analysis (READ) module. It is a square matrix in machine precision that has terms only along its diagonal, with the values along the diagonal representing the eigenvalues. For DMAP programmers interested in retrieving and manipulating eigenvalues, this matrix offers an easier and more precise means than the use of the LAMX module which operates on the LAMA (eigenvalue) table, containing eigenvalues and the related data only in single precision.

## 3.17 Fast Direct Frequency Response Option

The Fast Direct Frequency Response option is activated with the system cell NASTRAN FASTDFR=yes. The default setting is FASTDFR=no resulting in the conventional FRRD1 methods. Otherwise, it will be transparent to the user running SOL 108. In MSC.Nastran 2004, only the FREQ and FREQ1 entries will be supported in specifying the forcing frequencies. The excitation may be described by RLOAD1, RLOAD2, DAREA and only the linear interpolation TABLED1, 2, 3 entries are supported.

### Theory

The method is based on the principle of building a Krylov subspace at some of the requested frequency response locations and calculating approximate responses at other locations without redecomposing the dynamic matrix. The building of the Krylov subspace is accomplished via the Lanczos method or the Arnoldi method. The latter one has some advantages when the matrices are unsymmetric.

### Inputs

The option selection, the subspace generation methods, as well as several parameters available to control the fast direct frequency response option are shown in [Table 3-2](#):

**Table 3-2**

Cell Number	Functions & References
387	Options related to fast direct frequency response analysis. -1: yes 0: no (default)
388	Options related to fast direct frequency response analysis. Selects subspace generation method 1: Lanczos (default) 2: Arnoldi
389	Options related to fast direct frequency response analysis. Defines exponent of relative accuracy -4: error<1.0E-4 (default) -6: error<1.0E-6

Table 3-2

Cell Number	Functions & References
390	Options related to fast direct frequency response analysis. Defines pole selection distance 0: next pole is next unconverged frequency (default) 2: 2*next frequency distance -2: 1/2*next frequency distance
391	Options related to fast direct frequency response analysis. Selects decomp/fbs trade-off parameter 1: fbs time = decomp time (default) 2: fbs time - 2*decomp time -2: fbs time - 1/2*decomp time

The FASTDFR option involves many forward-backward substitutions (FBS). The performance advantage comes from trading decompositions for FBSs. This trade-off is heavily influenced by the discretization and the modal density of the frequency range and is also affected by the available memory. At this stage, the option seems to have a performance advantage mainly on predominantly solid element models with smaller frequency steps and less modal density. The results in these cases are sometimes significantly faster. The method, however, is less efficient on direct frequency response of carbody type models.



## 3.18 Inertia Relief with Auto-Suport

### Enhancement

MSC.Nastran's inertia relief capability in linear statics SOL 101 was evaluated in MSC.Nastran 2001 with PARAM, INREL,-2, using auto-suport. This capability is fully deployed for MSC.Nastran 2004.

PARAM,INREL,-2, provides the following enhancements to inertia relief analysis:

1. Auto-suport (SUPPORT entries are not required) for the 6 rigid body motions of an unconstrained connected structure.
2. Unique displacement solution that is decoupled from rigid body motion and identical to SOL 111 elastic modes static solution as  $f \rightarrow 0$  Hz.
3. Improved accuracy in internal loads and stress calculations in “difficult-to-suport” models.

### Review

Inertia relief is a feature in MSC.Nastran SOL 101, which enables applied static loading to an unconstrained structure and the calculation of deformed shape and internal loads within the accelerated structure. Inertia relief calculates the rigid body mass x acceleration loads imparted by the applied loads, and applies them in combination to the flexible body to produce a load-balanced static formulation in the linear acceleration reference frame. The “steady-state” relative structural displacements and internal loads are calculated using suport entries (PARAM,INREL,-1) or the new auto-suport capability (PARAM,INREL,-2).

Inertia relief is commonly used to measure inertial compliance or dynamic stiffness of an unconstrained component under attachment loads or under center of gravity unit loads. Inertia relief has the valuable property within SOL 101 of providing the static-dynamic convergence shape and elastic modes solution of a dynamically loaded free-free structure as  $f \rightarrow 0$  hz in SOL 111 (using PARAM,LFREQ and residual load vectors).

### General Formulation

Consider common 3-D unconstrained structure with six 0 Hz rigid body modes.

Rigid body mechanics loads balance (small motion):

$$\sum \bar{F}_i - \sum m_i \bar{a}_i = \bar{0} \text{ and } \sum (\bar{M}_i + \dot{\bar{r}}_{i/o} \cdot \bar{F}_i) - \sum (I_i \bar{\alpha}_i + \dot{\bar{r}}_{i/o} \cdot m_i \bar{a}_i) = \bar{0} \text{ Eq. 3-25}$$

In finite element matrix notation:

$$[R]^T \{P\} - [R]^T [M] [R] \{\ddot{u}_o\} = \{0\} \quad \text{Eq. 3-26}$$

$R$  is a (a-dof x 6) geometric rigid body matrix resulting from unit displacements in each basic direction with respect to GRDPNT or (0,0,0).  $R$  provides summation and cross-product utilities for loads and motion at each dof  $i$ . Rigid body accelerations  $\ddot{a}_i$  and  $\ddot{\alpha}_i$  are represented by 6 x 1  $\ddot{u}_o$  at PARAM,GRDPNT. All  $\vec{F}_i$  and  $\vec{M}_i$  are entered into load vector  $P$ ;  $m_i$  and  $I_i$  are entered into a-set mass matrix  $M$ . Solve for the rigid body accelerations:

$$\{\ddot{u}_o\} = ([R]^T [M] [R])^{-1} [R]^T \{P\} \quad \text{Eq. 3-27}$$

$R^T M R$  is the total 6 x 6 a-set mass, nonsingular for normal 3-D models with appropriate mass properties.

Apply the balanced loads to the finite element structure in linear statics formulation. This form is employed by the PARAM,INREL,-2 method:

$$[K] \{U\} = \{P\} - [M] [R] \{\ddot{u}_o\} \quad (\text{Inrel} = -2) \quad \text{Eq. 3-28}$$

In contrast, older method INREL = -1 and SOL 111 employ the following:

$$[K] \{U\} = \{P\} - [M] [D] \{\ddot{U}_r\} \quad (\text{Inrel} = -1) \quad \text{Eq. 3-29}$$

$$[K] \{U\} = \{P\} - [M] [\phi_r] \{\ddot{q}_r\} \quad (\text{SOL 111 free-free RESVEC's}) \quad \text{Eq. 3-30}$$

Each method uses a different representation for the rigid body matrix and accelerations. Stiffness matrix  $K$  is singular (i.e., rank  $l = a - 6$ ), and each method likewise employs different techniques to solve for displacement shape  $U$ .

## Inrel = -2

Inrel = -2 with auto-suport is designed for 3-D models with exactly 6 rigid body modes. The geometric  $R$  matrix is generated about reference GRDPNT, used instead of suport entries.  $R$  is a linear combination of normal rigid body modes  $\phi_r$  and can be used to decouple rigid motion and provide a unique inertia relief solution, convergent to the SOL 111  $f \rightarrow 0$  result.

## SOL 111 Basis

In MSC.Nastran modal frequency response SOL 111, the elastic solution on a free-free structure converges statically as follows (using elastic modes notation  $e$ ):

$$\{U_e\} = \lim_{f \rightarrow 0} \{U_e(f)\} = [\phi_e][k_{ee}]^{-1}[\phi_e]^T\{P\} \quad \text{Eq. 3-31}$$

$U_e$  also satisfies [Eq. 3-30](#). It is obtained in MSC.Nastran SOL 111 doing the following:

- Set PARAM,LFREQ or EIGRL f1 to low frequency value (i.e., 0.1 Hz) to exclude use of rigid body modes.
- Remove stiffness damping or load phasing to get real-valued, true static result.
- Include residual load vectors (2004 default) to ensure convergence on the static result.

An important observation is that  $U_e$  is orthogonal or “decoupled” from eigensolution rigid body modes  $\phi_r$ . For “clean” low-strain rigid modes  $r \geq 6$ , geometric rigid body vectors  $R$  can be expressed as a linear combination of  $\phi_r$ . SOL 111 static  $U_e$  is likewise decoupled from rigid motion  $R$ :

$$[\phi_r]^T[M]\{U_e\} = \{0\} \quad \rightarrow \quad [R]^T[M]\{U_e\} = \{0\} \quad \text{Eq. 3-32}$$

## SOL 101 Implementation

SOL 101 can use geometric  $R$  at low computational cost without having to compute normal rigid modes  $\phi_r$ . For INREL = -2, impose the rigid body decoupling constraint observed in [Eq. 3-32](#). Re-arrange [Eq. 3-28](#), [Eq. 3-27](#), and [Eq. 3-32](#):

$$\begin{aligned} [K]\{U\} + [M][R]\{\ddot{u}_o\} &= \{P\} \\ [R]^T[M][R]\{\ddot{u}_o\} &= [R]^T\{P\} \\ [R]^T[M]\{U\} &= \{0\} \end{aligned}$$

Add together to obtain:

$$\begin{bmatrix} K & MR \\ R^T M & R^T MR \end{bmatrix} \begin{Bmatrix} U \\ \ddot{u}_o \end{Bmatrix} = \begin{Bmatrix} P \\ R^T P \end{Bmatrix} \quad \text{Eq. 3-33}$$

**Eq. 3-33** is the resultant INREL = -2 formulation. Augmented **Eq. 3-33** matrix is nonsingular for models with 6 rigid mode singularities in matrix  $K$ , because of the addition of 6 acceleration + 6 decoupling equations. For this scenario the solution is unique and identical to the SOL 111 elastic convergence solution. If  $K$  has more than 6 singularities - local, massless, or true mechanisms - **Eq. 3-33** will become singular. Auto-support removes extra singularities in the matrix reduction above, but the quality of the inertia relief solution could reduce. Check for high factor diagonal ratios in the .f06 output. It is recommended that  $K$  have only the 6 rigid mode singularities for inertia relief with auto-support to work most effectively.

---

**Note:** There is no actual Support of grids or scalar points in the Inrel = -2 implementation. Unlike INREL = -1, there are no enforced zeroes or “center of displacement.” The INREL = -2 solution typically balances around the CG of the structure, with the CG displacing slightly in the direction of the applied loads. This “balance” differs for each applied load  $P$ . INREL = -2 uses PARAM,GRDPNT and rigid body acceleration as a geometric reference, not required to be part of structure or independent a-set.

---

## Inrel = -1

Support DOF (r-set) are commonly used in MSC.Nastran to define reference DOF and partition singular stiffness  $K$  to calculate a relative displacement shape  $U$ . They are part of structure and solution ( $U_r$ ), and must be independent DOF in the a-set. In inertia relief, Support DOF also define the rigid body accelerations of the structure with non-zero  $\ddot{U}_r$ . INREL = -1 uses a stiffness/support-generated rigid body matrix  $D$  applying unit displacements at each r-DOF in the flexible body:

$$[D] = \begin{bmatrix} -K_{ll}^{-1} K_{lr} \\ I_{rr} \end{bmatrix} = \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix} \quad \text{Eq. 3-34}$$

INREL= -1, **Eq. 3-29** becomes:

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{Bmatrix} U_l \\ U_r \end{Bmatrix} = \begin{Bmatrix} P_l \\ P_r \end{Bmatrix} - \begin{bmatrix} M_{ll} & M_{lr} \\ M_{rl} & M_{rr} \end{bmatrix} \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix} \{\ddot{U}_r\} \quad \text{Eq. 3-35}$$

Premultiply by  $D^T$  and solve for acceleration  $\ddot{U}_r$  similar to **Eq. 3-27**:

$$\{\ddot{U}_r\} = ([D]^T [M] [D])^{-1} [D]^T \{P\} \quad \text{Eq. 3-36}$$

**Note:**  $[D]^T[K] = [0]$  is assumed, and valid when the structure is not grounded, and support DOF are a well-chosen statically determinate set, i.e., at 6 stiff DOF on the structure.  $D$  is comprised of strain-free rigid body vectors.

Solve top portion of Eq. 3-29 for  $U_i$  in terms of  $U_r$ :

$$\{U_i\} = [D_{lr}]\{U_r\} + [K_{ll}]^{-1}(\{P_l\} - [\bar{M}_{lr}]\{\ddot{U}_r\}) \quad \text{Eq. 3-37}$$

Setting  $U_r = 0$  yields following relative displacement solution  $U$ :

$$\{U\} = \begin{bmatrix} U_i \\ U_r \end{bmatrix} = \begin{bmatrix} [K_{ll}]^{-1}\{\bar{P}_l\} \\ 0 \end{bmatrix}, \quad \{\bar{P}_l\} = \{P_l\} - [(M_{ll}D_{lr} + M_{lr})]\{\ddot{U}_r\} \quad \text{Eq. 3-38}$$

A significant property of INREL = -1 is that displacement  $U$  solution is not unique and differs with selection of support DOF. But deformed shape of  $U$  and corresponding internal loads and stresses should not vary with support sets. For well-chosen, statically determinate, stiff support sets 1a and 1b:

$$\{U^{1a}\} \neq \{U^{1b}\} \quad \text{shape}\{U^{1a}\} = \text{shape}\{U^{1b}\} \quad \sigma\{U^{1a}\} = \sigma\{U^{1b}\} \quad \text{Eq. 3-39}$$

On standard unconstrained 6 rigid-body-mode structure, ideal support selections are:

- All 6 DOF at stiff independent grid at or near CG.
- 6 DOF spread out to 3 distant stiff locations on the structure, in 3-2-1 statically determinate balance (i.e., support 123 on one corner, 23 and 3 on other corners).

## Internal Loads and Accuracy

Inertia relief theory is predicated upon strain-free rigid body vectors available in the finite element model to formulate the free-free deformed solution:

$$\sigma[R] \approx \sigma[D] \approx \sigma[\phi_r] \approx [0] \quad \text{Eq. 3-40}$$

When this assumption is valid, the following convergence and accuracy can be obtained:

$$\sigma\{U^2\} = \sigma\{U^1\} = \sigma\left\{U_e^{111}\right\} \quad \text{Eq. 3-41}$$

INREL = -1 and matrix D tend to be more susceptible to inaccuracies above, because of the vagaries of support selection, and the potential for “soft” supports. INREL = -2 tends to be more numerically robust for internal loads as well as displacement, as long as 6 rigid body modes exist which remain distinct from the 7th mode. Ideally rigid body mode frequencies should be on the order E-3 or E-4 less than first flexible mode. If difficult to achieve, both methods may be required to increase confidence in the analyses.

## Guidelines for Inertia Relief

1. Inertia relief analysis requires that the model have inertia, that is, mass. This is easily accomplished for most models by use of the RHO field (mass density) on the MATi Bulk Data entries. Line structures, such as a shaft made from BAR elements, may not have moments of inertia along the torsion axis. Use either BEAM elements, which can have torsion mass moments of inertia, or add inertia terms with CONM2 entries.
2. INREL = -2 with auto-support is designed for the common 6 rigid-body-mode unconstrained connected model. If the model is intended to have exactly 6 rigid body modes, then INREL = -2 is the most convenient and numerically stable option. SUPORTi entries must NOT be used.
3. If the model is semi-constrained with < 6 rigid body modes, including 1-D or 2-D models, INREL = -1 is recommended, using < 6 appropriate support DOF.
4. Inertia relief is not recommended on models with extra mechanisms or unconnected parts with mass, with > 6 zero Hz modes. INREL = -1 will error off with supports > 6, and INREL = -2 with auto-support will error off or give unreliable results.
5. For models with local or massless singularities beyond the 6 rigid body modes, INREL = -2 with auto-support can produce a reasonable result. But care should be taken, and singular DOF with high factor diagonals in the .f06 should be checked.
6. PARAM,GRDPNT is the reference point for INREL = -2. If default (0,0,0) is far from the model CG, then GRDPNT should be defined at or near the CG.
7. When SUPORTi entries are used in INREL = -1, a good rule of thumb is that they be on structure strong and balanced enough to sustain the weight of the structure in a laboratory test (they must have significant stiffness). If the structure does not have such DOFs, a new grid can be defined as a reference node, and connected to many DOFs with an RBE3 element. The reference node must be in the n-set which implies that the UM feature of the RBE3 element must be used.

8.  $\text{Inrel} = -1$  SUPORTi entries must constitute a statically determinate set  $\leq 6$ . Good selections are either: a) all DOF at CG, or b) DOF distanced out in 3-2-1 fashion to create a good balance of the structure. See discussion in  $\text{INREL} = -1$  section.
9.  $\text{INREL} = -2$  should be used if the inertia relief solution is intended to match the SOL 111  $f \rightarrow 0$  Hz elastic solution exactly.
10.  $\text{INREL} = -1$  should be used if a zeroed “center of displacement” is desired.
11. Both inertia relief methods are most accurate on ungrounded “clean” models with rigid body modes on the order  $E-4$  less than first flexible mode frequency. Case Control command, GROUNDCHECK, can be helpful in detecting grounding problems. For  $\text{INREL} = -1$ , Support Strain Energy in .f06 should also be checked. The  $\text{INREL} = -2$  method remains more robust and accurate, as long as 6 rigid modes are intended and remain fairly distinct from the 7th mode. For models where separation between 6th and 7th mode is less than  $E-1$ , then  $\text{INREL} = -1$  with good support selection should be considered as well.

### 3.19 New C-Set and R-Set Reduction

In previous version of MSC.Nastran, the o-set (interior dofs) motion associated with the b-set, c-set, and r-set (boundary dofs) was generated from the static reduction of the stiffness matrix.

$$[G_{ot}] = [G_{ob} | G_{ow}] = [-K_{oo}]^{-1} [K_{ob} | K_{ow}] \quad \text{Eq. 3-42}$$

where the w-set = c-set + r-set.

The boundary motion determines the response of the o-set.

$$\{u_0\} = [G_{ob} | G_{ow}] \begin{Bmatrix} u_b \\ u_w \end{Bmatrix} \quad \text{Eq. 3-43}$$

For dynamic analyses, the o-set motion can be augmented by the use of component modes to improve the response estimate. The motion of the component modes is represented by the q-set.

$$\{u_0\} = [G_{ob} | G_{ow} | \bar{\Phi}_{oq}] \begin{Bmatrix} u_b \\ u_w \\ \xi_q \end{Bmatrix} \quad \text{Eq. 3-44}$$

where the component modes,  $\bar{\Phi}_{oq}$ , are determined from modes of the v-set (v-set = o-set + c-set + r-set).

The modes of the v-set are calculated in the standard fashion.

$$(-\lambda[M_{vv}] + [K_{vv}])\{\phi_v\} = \{0\} \quad \text{Eq. 3-45}$$

The motion of the boundary grids are then removed from the calculated modes

$$\{\bar{\Phi}_o\} = \{\phi_o\} - [G_{ow}]\{\phi_w\} \quad \text{Eq. 3-46}$$

where

$$\{\phi_v\} = \begin{Bmatrix} \phi_o \\ \phi_w \end{Bmatrix}$$

$\{\phi_v\}$  are modes of the v-set dofs

$\{\bar{\Phi}_o\}$  are the v-set modes with the w-set boundary motion removed



The previous operation decouples the w-set boundary motion  $[G_{ow}]$  from the v-set modes  $[\phi_v]$ . This decoupling allows the boundary and component mode dofs to move independently and the total motion of the structure to be written in terms of the a-set (b-set + w-set + q-set).

$$\begin{Bmatrix} u_b \\ u_w \\ u_o \end{Bmatrix} = \begin{bmatrix} I & 0 & 0 \\ 0 & I & 0 \\ G_{ob} & G_{ow} & \bar{\Phi}_{oq} \end{bmatrix} \begin{Bmatrix} u_b \\ u_w \\ \xi_q \end{Bmatrix} \quad \text{Eq. 3-47}$$

If residual vectors are requested, they are calculated using the o-set matrices and appended to the component modes.

The new w-set processing follows a slightly different path. The procedure is based on a method suggested by Dr. Arya Majed and Ed Henkel. This method is the new default procedure for c-set reduction. The previous method for c-set reduction can be selected by setting PARAM,MHRED to 'NO'.

The new c-set procedure uses static reduction to determine the motion of the v-set due to boundary b-set motion.

$$\{u_v\} = [G_{vb}]\{u_b\} \quad \text{Eq. 3-48}$$

where

$$[G_{vb}] = [-K_{vv}]^{-1}[K_{vb}] \quad \text{Eq. 3-49}$$

As previously was done, modes of the v-set are also calculated.

$$(-\lambda[M_{vv}] + [K_{vv}])\{\phi_v\} = \{0\} \quad \text{Eq. 3-50}$$

If residual vectors are requested, they are calculated using the v-set size matrices and appended to the v-set modes.

At this point, the b-set boundary vectors and the v-set modes can describe the motion of the structure.

$$\begin{Bmatrix} u_b \\ u_v \end{Bmatrix} = \begin{bmatrix} I & 0 \\ G_{vb} & \phi_{vi} \end{bmatrix} \begin{Bmatrix} u_b \\ \xi_q \end{Bmatrix} \quad \text{Eq. 3-51}$$

or splitting the v-set into its o-set and w-set partitions,

$$\begin{Bmatrix} u_b \\ u_w \\ u_o \end{Bmatrix} = \begin{bmatrix} I & 0 \\ G_{wb} & \phi_{wq} \\ G_{ob} & \phi_{oq} \end{bmatrix} \begin{Bmatrix} u_b \\ \xi_q \end{Bmatrix} \quad \text{Eq. 3-52}$$

In the above representation, the w-set dofs are not independent dofs, but are dependent on b-set and q-set motion. In order to make the w-set independent, vectors that associate o-set motion with individual w-set dofs must be found. Once these vectors are found, the components are removed from the b-set and q-set motion resulting in w-set vectors that are independent.

The w-set vectors are determined using the flexibility of the v-set.

$$\begin{bmatrix} Y_{oo} & Y_{ow} \\ Y_{wo} & Y_{ww} \end{bmatrix} \begin{Bmatrix} P_o \\ P_w \end{Bmatrix} = \begin{Bmatrix} u_o \\ u_w \end{Bmatrix} \quad \text{Eq. 3-53}$$

If loads are applied only to the w-set, the displacements are dependent only on  $P_w$ .

$$\begin{bmatrix} Y_{ow} \\ Y_{ww} \end{bmatrix} \{P_w\} = \begin{Bmatrix} u_o \\ u_w \end{Bmatrix} \quad \text{Eq. 3-54}$$

Vectors that associate o-set motion with individual w-set dofs can be found by setting  $P_w = Y_{ww}^{-1}$ .

$$\begin{bmatrix} Y_{ow} \\ Y_{ww} \end{bmatrix} [Y_{ww}^{-1}] = \begin{bmatrix} Y_{ow} Y_{ww}^{-1} \\ I_{ww} \end{bmatrix} \quad \text{Eq. 3-55}$$

The above vectors are used to make the w-set an independent set.

$$\begin{Bmatrix} u_b \\ u_w \\ u_o \end{Bmatrix} = \begin{bmatrix} I & 0 & 0 \\ 0 & I & 0 \\ G_{ob} - Y_{ow} Y_{ww}^{-1} G_{wb} & Y_{ow} Y_{ww}^{-1} \phi_{oq} - Y_{ow} Y_{ww}^{-1} \phi_{wq} \end{bmatrix} \begin{Bmatrix} u_b \\ u_w \\ \xi_q \end{Bmatrix} \quad \text{Eq. 3-56}$$

The flexibility matrices  $Y$  are determined from displacements due to unit forces on the w-set.

$$[K_{vv}]^{-1} \begin{bmatrix} 0 \\ I_{ww} \end{bmatrix} = [\bar{Y}_{vv}] \quad \text{Eq. 3-57}$$

The modal flexibility is removed from the displacement response to determine the flexibility matrices used in [Eq. 3-56](#).

$$[Y_{vv}] = [\bar{Y}_{vv}] - [\phi_{vq}]([\phi_{vq}]^T[M_{vv}][\phi_{vq}])^{-1}[\phi_{vq}]^T[M_{vv}][\bar{Y}_{vv}] \quad \text{Eq. 3-58}$$

$$[Y_{vv}] = \begin{bmatrix} Y_{ow} \\ Y_{ww} \end{bmatrix} \quad \text{Eq. 3-59}$$



CHAPTER

# 4

## Rotordynamics

- Introduction
- Examples
- Equations Used in Analyses

## 4.1 Introduction

The new rotordynamics capability provides users with a relatively simple method of performing the design and analysis of structures with rotating components. The procedures were developed with input from aerojet engine manufacturers to ensure meeting their basic requirements. The new rotordynamics capability can be used in frequency response (direct and modal), complex modes (direct and modal), static, nonlinear transient, and linear transient (direct only) analyses.

Frequency response analyses determine the rotor/support response to arbitrary excitation with the rotors spinning at user specified rates (ASync) or excitation that is synchronous with the reference rotor (Sync). The ASync calculation determines the response of the system to an external input that is independent of the rotor speed. The Sync calculation determines the system response to a rotor imbalance or other excitation that is dependent on the rotor spin rate.

Complex modes analyses determine the whirl frequencies and critical speeds. Whirl modes are modes of a rotor/support system with the rotors spinning at specified rates. Critical speeds are whirl frequencies that coincide with a rotor spin rate. Calculation of whirl frequencies is performed by the asynchronous option (ASync). Calculation of critical speeds is performed by the synchronous option (Sync).

Static analysis determines the loads on the rotor due to pitch or yaw of an aircraft. Severe transverse rotations may be part of the design environment and limiting rotor motion to prevent rotor blades from rubbing against the interior of the casing may be design criteria.

Direct linear and nonlinear transient analyses allow simulation of engine blade-out and subsequent windmilling. The analysis is used to ensure structural integrity during flight and to guarantee that excessive vibration levels are not reached.

All the above analyses are performed in the stationary (inertial) coordinate system with the assumption that the rotors have rotational symmetry of three or greater and are reduced (by static condensation) to a line model. The rotor is assumed to be flexible along its axis and planes perpendicular to the axis remain perpendicular to the axis as the axis flexes. Motion off-axis can be recovered from the motion of the point on the line model that is in the same perpendicular plane.

### Benefits

Traditionally, companies have used proprietary, in-house codes and customized DMAP solutions to perform analyses of structures with rotating components. The rotordynamics capability in MSC.Nastran provides an easy-to-use standardized

procedure for analyzing structures with rotating components. It is now an integral part of MSC.Nastran dynamic solution sequences, and is invoked using case control commands and bulk data entries. This standardization allows easier exchange of model data between in-house groups or external vendors.

The design of the rotordynamics capability allows multiple rotors to be included in the analyses. The rotors may spin at different rates and be aligned at arbitrary angles.

## Basic Theory and Methods

All analyses are performed in the stationary coordinate system. This requires that the rotors are symmetric about the rotation axis. Analysis in the stationary system also requires that the rotors be modeled as line models or 3-D models statically reduced to a line model (component-mode models are not allowed). Data recovery is correct for the engine casing and support structure. Data recovery for the rotors is limited to the overall response along the axis of rotation. Detailed displacements and stress recovery off the axis of rotation are not correct.

## Basic Equations

The MSC.Nastran time-domain equation solved by the new dynamic solution sequences is:

$$M\ddot{u}(t) + \left( B_s + \left( \frac{g}{WR3} \right) K_s + \left( \frac{1}{WR4} \right) K4_s + B_r + \left( \frac{g_r}{WR3} \right) K_r + \left( \frac{1}{WR4} \right) K4_r + \Omega B^G \right) \dot{u}(t) + \left( K_s + K_r + \Omega \left( K^{Cv} + \left( \frac{g_r}{WR3} \right) K^{Cgr} + \left( \frac{1}{WR4} \right) K^{Cge} \right) \right) u(t) = F(t) \quad \text{Eq. 4-1}$$

$M$  = total mass matrix (CONMi, CMASSi, MATi)

$B_s$  = support viscous damping matrix (CVISC, CDAMPi)

$\left( \frac{g}{WR3} \right) K_s$  = support viscous damping equivalent to structural damping (PARAM,G)

$\left( \frac{1}{WR4} \right) K4_s$  = support viscous damping equivalent to material structural damping (GE on MATi)

$B_r$  = rotor viscous damping matrix (CVISC, CDAMPi)

$\left( \frac{g_r}{WR3} \right) K_r$  = rotor viscous damping equivalent to structural damping (GR on RSPINT)

$\left( \frac{1}{WR4} \right) K4_r$  = rotor viscous damping equivalent to material structural damping (GE on MATi)

$B^G$  = gyroscopic force matrix (dependent on polar moment of inertia)

$K_s$  = support stiffness matrix

$K_r$  = rotor stiffness matrix

$K4_s$  = support material damping matrix (GE on MATi)

$K4_r$  = rotor material damping matrix (GE on MATi)

$\Omega$  = rotor spin rate

$K^{Cv}$  = “circulation” matrix due to  $B_r$

$g_r K_r^{Cgr}$  = “circulation” matrix due to  $g_r K_r$

$K_r^{Cge}$  = “circulation” matrix due to  $K4_r$

$G$ ,  $WR3$ , and  $WR4$  are user parameters

The MSC.Nastran frequency-domain equation is:

$$(-\omega^2 M + i\omega(B_s + B_r + \Omega B^G) + \left( (1 + ig)K_s + iK4_s + (1 + ig_r)K_r + iK4_r + \Omega \left( K^{Cv} + \left( \frac{g_r}{\omega} \right) K^{Cgr} + \left( \frac{1}{\omega} \right) K^{Cge} \right) \right)) u(\omega) = F(\omega) \quad \text{Eq. 4-2}$$

The rotor spin rate,  $\Omega$ , is specified by the user. In the synchronous option where  $\omega = \Omega$ , the above equation becomes

$$(-\Omega^2(M - iB^G) + i\Omega(B_s + B_r - iK^{Cv}) + ((1 + ig)K_s + iK4_s + (1 + ig_r)K_r + iK4_r + g_r K_r^{Cgr} + K^{Cge})) u(\Omega) = F(\Omega) \quad \text{Eq. 4-3}$$

**Eq. 4-1** is used in transient analysis; **Eq. 4-2**, and **Eq. 4-3** are used in frequency response and complex modes analyses. See “**Equations Used in Analyses**” on page 192 for more details.

## Whirl Modes and Critical Speeds

A characteristic of the motion associated with whirl frequencies and critical speeds is that the rotor appears to “whirl” in a circular or elliptical motion relative to the support structure. The whirl motion may be either in the direction of rotor rotation (forward whirl) or against it (backward whirl). Rotor “whirl” can be at a frequency that is different from the rotor spin rate. Critical speeds can be viewed as a special case of whirl frequencies: the rotor “whirl” is at the same frequency as the rotor spin rate.



Complex eigenvalue analysis can be used to determine whirl frequencies and critical speeds, and their associated motion. For whirl frequencies, [Eq. 4-2](#) is solved for  $\omega$  (whirl frequency) and  $u$  (mode shape) with  $F = 0$  and the rotor spin rate  $\Omega$  specified by the user. For critical speeds, [Eq. 4-3](#) is solved for  $\Omega$  (critical speed) and  $u$  (mode shape) with  $F = 0$ .

Critical speeds can also be determined using whirl frequency calculations. This is done using Campbell diagrams. Campbell diagrams plot eigenvalues versus rotor speed or other parameters that affect the natural frequencies of the system. [Eq. 4-2](#) is solved for a series of rotor speeds and the results plotted on the Campbell diagram. Eigenvalues with similar modes shapes are connected to form a series of lines that represent the changing natural frequencies. Critical speeds are determined by assessing which eigenvalues are identical to the rotor speed. An additional line for  $\omega = \Omega$  (natural frequency = rotor spin rate) is drawn. The intersection of the  $\omega = \Omega$  line with the eigenvalue lines are the critical speeds.

Whirl frequencies are important in determining rotor stability. Although the whirling motion may not be at the rotational speed, whirl modes may be unstable due to internal rotor damping or external forces due to bearing and seals. Estimating the stability of the whirl modes is important for ensuring a system operates smoothly.

Critical speeds are also important for ensuring that a system does not become unstable. Although usually finely balanced, rotors are never perfectly balanced. Any imbalance in the rotor will produce oscillating forces in the rotor/support structure. If rotor speed is at or near a critical speed, the system will experience high vibratory motion. If the imbalance is large enough, the structure may fail or cause significant damage. Knowing the critical speeds allows the user to specify safe operating ranges.

## Multiple Rotors and the Reference Rotor

In case of multiple rotors, [Eq. 4-1](#) through [Eq. 4-3](#) are modified to include the gyroscopic terms and spin rates of the individual rotors (see [Section 4.3](#).) For time-domain analyses, the user specifies the rotor spin rates directly. For frequency domain and static analyses, a reference rotor must be specified. The analyses are performed with the reference rotor spinning at the specified speed. The spin rates of the other rotors are determined by means of user specified relationships between the rotor spin rates (see the RSPINR entry).

For frequency-domain analyses (complex modes and frequency response) with synchronous gyroscopic forces, remember that the analyses are performed with respect to the reference rotor. The reference rotor is spinning at the analysis frequency, or for complex modes, at the eigenfrequency. The results should be interpreted in terms of the reference rotor.

For example, using complex eigenvalue analysis, it is possible to determine the natural frequencies of a multi-rotor system consisting of rotors spinning at different rates. Consider a structure with two rotors. The second rotor spins at twice the speed of the first rotor. The first rotor is defined as the reference rotor and complex eigenvalue analysis is performed using the synchronous option  $\omega = \Omega$ . Upon inspection of the results, a mode appears to be a whirl mode of the second rotor. One would expect that the second rotor has a critical speed at twice the reported eigenfrequency (because with the synchronous option, the eigenfrequency corresponds to the rotation rate of the reference or first rotor, and the second rotor is spinning at twice the rate of the first). This assumption would be incorrect. When the second rotor is spinning at twice the eigenfrequency (and the reference rotor is spinning at the eigenfrequency), there is a whirl mode (not a critical speed mode) of the second rotor. When inspecting the mode shapes, whirl modes of the reference rotor should be noted: their eigenfrequencies are critical speeds of the reference rotor. The effects of the other rotors are included in the analysis, but their modes are not critical speed modes.

## Input

### Case Control

RGYRO Case Control Command invokes all rotordynamics capabilities.

- RGYRO = YES/NO,  $n$  (Default = No)
  - For complex modes, frequency response, and static analysis, the RGYRO Case Control command selects the RGYRO Bulk Data entry for use in the analysis.
  - For direct linear and nonlinear transient response, the RGYRO Case Control command selects the UNBALNC Bulk Data entry that specifies the unbalance loading. If no unbalance loading is present, but gyroscopic terms are desired in the analysis, the RGYRO Case Control command should be set to YES.

### Bulk Data Additions for All Rotordynamic Analyses

#### Bulk Data

The rotordynamics capability requires the user to specify the rotor grids. This is done using the ROTORG entry. It is required for all rotordynamics options.

For frequency response, complex modes and static analysis, the RGYRO entry specifies the SYNC or ASYNC option, the reference rotor and rotor speed information. It is selected by the RGYRO Case Control command. Additionally, RSPINR entries are required to specify relative rotor speeds for multiple rotors. (This entry is also required for single models.)

For linear and nonlinear transient response, the RSPINT entries specify rotor speeds vs. time. Additionally, UNBALNC entries can be used to apply unbalance loads. UNBALNC entries are selected by the RGYRO Case Control command.

**ROTORG** - Specifies grids that compose the rotor line model

**Format:**

1	2	3	4	5	6	7	8	9	10
ROTORG	ROTORID	GRID1	GRID2	...	GRIDn				

or

ROTORG	ROTORID	GRID1	THRU	GRID2	BY	INC			
--------	---------	-------	------	-------	----	-----	--	--	--

Field	Contents
ROTORID	Identification number of rotor group. (Integer > 0; Required)
GRIDi	Grids comprising the rotor. (Integer > 0; Required, no Default)
THRU	Specifies a range of identification numbers. (Optional)
BY	Specifies an increment for a THRU specification. (Optional)
INC	Increment for THRU range. (Integer > 0; Default = 1)

**Remarks:**

1. Grid entries must be unique, duplicate entries will produce a fatal error.
2. Multiple ROTORG entries with the same ROTORID are supported.
3. All grids specified on ROTORG entries for a specific ROTORID must be collinear. Collinearity will be checked.

**Bulk Data Additions for Frequency Response, Complex Modes, and Static Analysis**

**RGYRO** - Specifies synchronous or asynchronous motion of the reference rotor, rotation speed of the reference rotor.

**Format:**

1	2	3	4	5	6	7	8	9	10
RGYRO	RID	SYNCLFG	REFROTR	SPDUNIT	SPDLOW	SPDHIGH	SPEED		

Field	Contents
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required, no Default)
SYNCLFG	Specifies whether the analysis is synchronous or asynchronous analysis. Required input for frequency response and complex modes analyses. Not required for static analyses. (Character: 'SYNC', 'ASYN', or blank)
REFROTR	Specifies the reference rotor ID for the analysis. (Integer > 0; Required, no Default)
SPDUNIT	Specifies whether the entries SPDLOW, SPDHIGH, and SPEED are given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/unit time). (Character: 'RPM' or 'FREQ'; no Default)
SPDLOW	Specifies the low speed for synchronous analysis. See Remark 2. (Real, Default = 0.0)
SPDHIGH	Specifies the high speed for synchronous analysis. See Remark 2. (Real, Default = 99999.0)
SPEED	Specifies reference rotor speed for asynchronous analysis. Also required for static analyses. See Remark 2. (Default = 0)

**Remarks:**

- 1. Multiple RGYRO entries with the same RID value are not allowed.
- 2. The required information on the RGYRO entries varies for different analyses. Values for the RID and SPDUNIT fields are always required. Values for SPDLOW, SPDHIGH and SPEED are analysis dependent as shown in the table below:

Solution Sequence	Type of Analysis	PARAM, GYROAVG	Required Entry	COMMENT
Frequency Response	SYNC	0	None	--
	SYNC	-1	SPDLOW, SPDHIGH	a, b
	ASync	0	SPEED	--
	ASync	-1	SPEED	b
Complex Modes	SYNC	--	SPDLOW, SPDHIGH	a, b
	ASync	--	SPEED	b
Static Analysis	--	--	SPEED	--

- a. The relative rotor speeds will be treated as linearly dependent on the reference rotor speed ( $\Omega = A0 + A1\Omega_{\text{reference}}$ ). The scale factors A0 and A1 will be determined by a least-mean-square fit of the relative rotor speeds input on the RSPINR entries between SPDLOW and SPDHIGH of the reference rotor. If SPDLOW or SPDHIGH are outside the range specified on the RSPINR entry, the values will be extrapolated from the RSPINR entry values.
- b. PARAM, WR3 and PARAM, WR4 are required for rotor damping.

**RSPINR** - Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
RSPINR	ROTORID	GRIDA	GRIDB	GR	SPDUNT	SPEED1	...	SPEEDn	

Field	Contents
ROTORID	Identification number of rotor. (Integer > 0; Required)
GRIDA/GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB (Integer > 0; Required, See Remark 2)
GR	Rotor structural damping factor, see Remark 3. (Real, Default = 0.0)

Field	Contents
SPDUNIT	Specifies whether the listing of relative spin rates is given in terms of RPM (revolutions/minute) or frequency (revolutions/unit time). (Character; 'RPM' or 'FREQ'; required)
SPEEDi	List of relative spin rates, entries for reference rotor must be in ascending or descending order. (Real; at least one entry required, no Default)

**Remarks:**

1. A RSPINR entry must be present for each rotor defined by a ROTORG entry.
2. The rotor spin axis is determined from the ROTORG entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG entry.
3. Rotor structural damping specified by the GR entry will be added as either viscous damping or structural damping.
4. The default is to add the damping as structural damping:

$$[\bar{K}_{rotor}] = (1 + iGR)[K_{rotor}]$$

Setting PARAM,GYROAVG,-1 will add the damping as viscous damping:

$$[B_{rotor}] = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

WR3 is specified by PARAM,WR3 (default = 0.0, no damping added). See “[Equations Used in Analyses](#)” in Chapter 4 of the *MSC.Nastran 2004 Release Guide* for complete details.

5. Number of entries for each rotor must be the same. Relative spin rates are determined by correlation of table entries. The ith entry for each rotor specifies the relative spin rates between rotors at RPMi/FREQi.

## Bulk Data Additions for Transient Response Analysis

**RSPINT** - Specifies rotor spin rates for nonlinear transient analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
RSPINT	ROTORID	GRIDA	GRIDB	GR	SPDUNIT	TID			

Field	Contents
ROTORID	Identification number of rotor. (Integer > 0; Required, no Default)
GRIDA/GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB (Integer > 0; Required, see Remark 1)
GR	Rotor structural damping factor, see Remark 2. (Real, Default = 0.0)
SPDUNIT	Specifies whether the spin rates is given in terms of RPM (revolutions/minute) or frequency (revolutions(cycles)/unit time). (Character; 'RPM' or 'FREQ'; Required, no Default)
TID	Identification number of TABLEDi entry specifying spin rate versus time. (Integer > 0; Required, no Default)

**Remarks:**

1. The rotor spin axis is determined from the ROTORG entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG entry.
2. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}] = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

WR3 is specified by PARAM,WR3 (default = 0.0, no damping added).

**UNBALNC** - Specifies an unbalance load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

**Format:**

1	2	3	4	5	6	7	8	9	10
UNBALNC	RID	MASS	GRID	X1	X2	X3			
	ROFFSET	THETA	ZOFFSET	T <sub>on</sub>	T <sub>OFF</sub>	CFLAG			

Field	Contents
RID	Identification number of UNBALNC entry. Selected by Case Control command, RGYRO. (Integer; Required; no Default)
MASS	Mass of imbalance (real or integer; if integer must be > 0; see Remark 3; Required, no Default)
GRID	Grid identification number of applying imbalance. The grid must appear on a ROTORG entry. (Integer .0; Required, no Default)
X1, X2, X3	Components of the vector, from GRID, in the displacement coordinate system of GRID, which is used to define a cylindrical coordinate system centered at GRID; see Remark 4. (Real, Required, no Default)
ROFFSET	Offset of mass in the radial direction of the unbalance coordinate system (real or integer; if integer, must be > 0; see Remark 3; Default = 1.0)
THETA	Angular position of the mass in the unbalance coordinate system. (Real; Default = 0.0)
ZOFFSET	Offset of mass in the z-direction of the unbalance coordinate system. (Real or Integer; if integer, must be > 0; see Remark 3; Default = 0.0)
T <sub>ON</sub>	Start time for applying imbalance load. (Real ≥ 0.0; Default = 0.0)
T <sub>OFF</sub>	Time for terminating imbalance load. (Real > 0.0; Default = 999999.0)
CFLAG	Correct flag to specify whether 1) the mass will be used to modify the total mass in the transient response calculations, 2) the effect of the rotor spin rate change will be included in the transient response calculation, or 3) both; see Remark 5. (Character: NONE, MASS, SPEED, or BOTH, Default = NONE)

**Remarks:**

1. Multiple UNBALNC entries with the same RID value may be used.



2. The imbalance load will be generated based on the mass value, offset values, and the rotor spin speed.
3. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a table entry that specifies the value as a function of time.
4. The cylindrical coordinate system used for the initial position of the mass unbalance has the positive z-axis direction from GRIDA to GRIDB specified on the RSPINT entry. Theta is measured from the plane defined by the z-axis and the user specified vector (X1, X2, X3). Theta = 0.0 is in the direction of the user-specified vector.
5. If the mass loss (increase) is relatively small, the correction may safely be ignored. (CFLAG = NONE)

## Parameters

The rotor dynamic parameters can be added in the Case Control or Bulk Data Sections.

GYROAVG	Used to specify one of two formulations for frequency response analysis using the rotordynamic capability. The default is to determine any frequency-dependent terms for each frequency. Setting the value < 0 uses an 'average' frequency formulation. This option avoids using the frequency-dependent looping and results in a shorter execution time. For this option, PARAM,WR3 and PARAM,WR4 must be specified to include rotor damping. See <a href="#">"Equations Used in Analyses"</a> on page 192 for detailed information. (Default=0)
WR3, WR4	Specifies "average" excitation frequency for calculation of rotor damping and circulation terms. See <a href="#">"Equations Used in Analyses"</a> on page 192 for equations. Default is 0.0, no rotor damping or circulation terms.

## New Case Control Options

As part of the rotordynamic implementation, the following Case Control commands and parameters can be changed for each subcase:

1. K2PP, M2PP, and B2PP (external matrices)
2. TFL (transfer functions)
3. RGYRO (rotordynamic specification)
4. SDAMP (modal damping)

5. PARAM,G (structural damping)
6. PARAM, ALPHA1 and PARAM, ALPHA2 (Rayleigh damping)
7. DLOAD (dynamic loads)
8. FREQ (frequency range)
9. CMETHOD (complex eigenvalue extraction method)

All the above items can be different in each subcase, but for efficiency reasons, subcases with the same values for items 1 through 6 should be grouped together.

## Additional Rotor Output

The new rotordynamics option produces additional rotor summary information in the output file. This information is output in three tables:

1. Summary of rotor grids and rotation axis for each rotor.
2. Summary of rotor grid mass properties and total rotor mass properties.
3. Rotor kinetic and strain energies for complex modes analysis.

Examples of the output for the rotor in [Figure 4-1](#) are shown.

The rotor grid and rotation axis summary:

### ROTOR DYNAMICS SUMMARY

-----

3 GRIDS FOUND ON ROTOR 10

101 102 103

#### SPIN DIRECTION IN BASIC COORDINATE SYSTEM

( FROM GRID 101 TO GRID 102 )

X 1.00000E+00

Y 0.00000E+00

Z 0.00000E+00

ALL 3 GRIDS FALL ON THE ROTOR AXIS

-----

## The rotor mass summary:

R O T O R   D Y N A M I C S   M A S S   S U M M A R Y				
-----				
MASS PROPERTIES OF ROTOR            10				
GRID ID	AXIAL MASS	TRANSVERSE MASS	POLAR MOMENT	TRANSVERSE MOMENT
101	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
102	5.00000E+01	5.00000E+01	5.00000E+00	1.50000E+01
103	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
-----				
ALL	5.00000E+01	5.00000E+01	5.00000E+00	1.50000E+01

For easier verification, the mass properties are in the same units used on the MATi and CONMi entries.

## The kinetic and strain energy summary for complex modes:

R O T O R   E N E R G I E S			
MODE NUMBER=	1		
COMPLEX EIGENVALUE=	0.000000E+00	6.324555E+01	
ROTOR	PERCENTAGE	PERCENTAGE	
NUMBER	STRAIN ENERGY	KINETIC ENERGY	
10	0.000000E+00	1.000000E+02	
MODE NUMBER=	2		
COMPLEX EIGENVALUE=	0.000000E+00	6.324555E+01	
ROTOR	PERCENTAGE	PERCENTAGE	
NUMBER	STRAIN ENERGY	KINETIC ENERGY	
10	0.000000E+00	1.000000E+02	
MODE NUMBER=	3		
COMPLEX EIGENVALUE=	0.000000E+00	1.000000E+02	
ROTOR	PERCENTAGE	PERCENTAGE	
NUMBER	STRAIN ENERGY	KINETIC ENERGY	
10	0.000000E+00	1.000000E+02	
MODE NUMBER=	4		
COMPLEX EIGENVALUE=	0.000000E+00	1.414214E+02	
ROTOR	PERCENTAGE	PERCENTAGE	
NUMBER	STRAIN ENERGY	KINETIC ENERGY	
10	0.000000E+00	1.000000E+02	

## 4.2 Examples

The rotors must be modeled as a detached line model in the G-set of the residual structure (superelement 0). If a 3-D model of a rotor is used, it must be reduced to a line model in the residual structure using superelement static reduction. Note that only static reduction may be used; dynamic reduction using component mode synthesis is not supported.

The rotor line model must be attached to the support using RBEi elements. If a mass, damping, or stiffness connection between the rotor and bearing or the support structure are detected, a fatal error will be issued. It is important to have the rotor detached from the support structure because the inertial and stiffness properties for the calculation of the gyroscopic terms are determined using the G-size matrices. Connections to ground or support elements may cause incorrect determination of the gyroscopic terms.

The following rotor consists of four rigid elements with a concentrated mass at the center grid. The end points are connected to ground with springs and dampers through RBE2 elements.

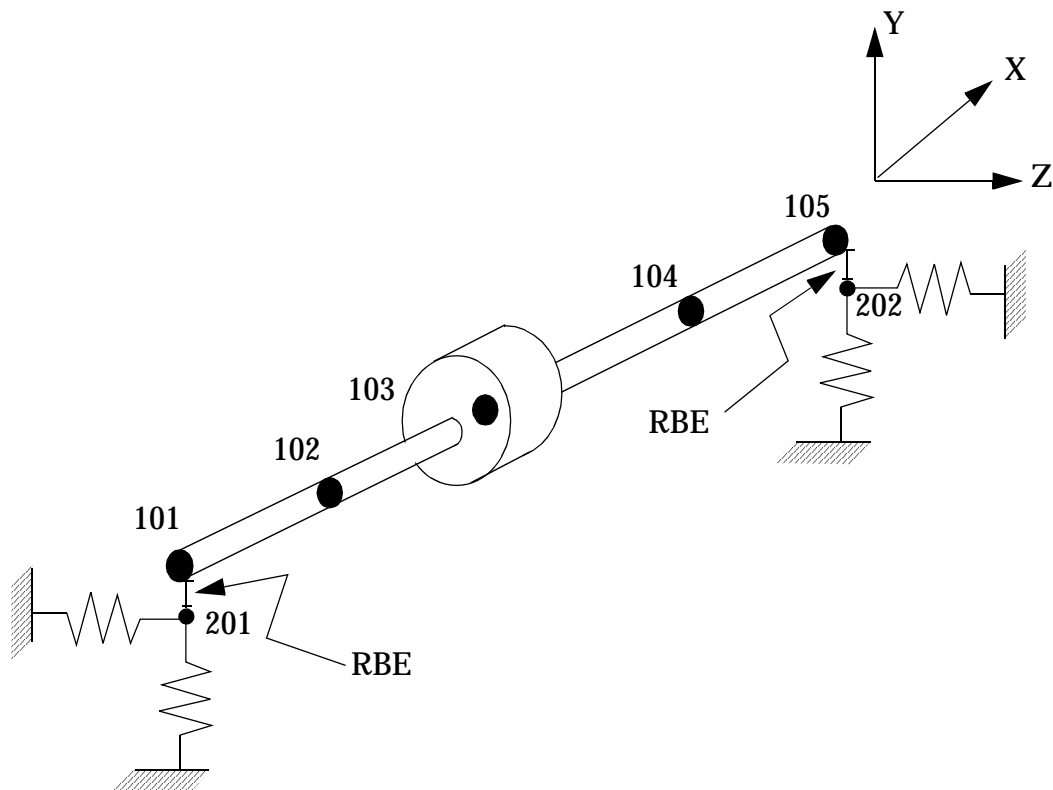


Figure 4-1 Example Rotor Model

**Listing 4-1 Rotor Model Bulk Data**

```

$
$ ROTOR
$
GRID, 101, , 0.0, 0.0, 0.0, , 14
GRID, 102, , 1.0, 0.0, 0.0, , 14
GRID, 103, , 2.0, 0.0, 0.0, , 14
GRID, 104, , 3.0, 0.0, 0.0, , 14
GRID, 105, , 4.0, 0.0, 0.0, , 14
$
GRID, 201, , 0.0, 0.0, 0.0, , 1456
GRID, 205, , 4.0, 0.0, 0.0, , 1456
$
RBE2, 1001, 103, 2356, 101, 102, 104, 105
$
CONM2, 2001, 103, , 3.0E-2,
, 2.0E-2, , 5.0E-2, , , 5.0E-2
$
$ SUPPORT
$
CELAS1, 3001, 3000, 201, 2
CELAS1, 3002, 3000, 201, 3
CELAS1, 3003, 3000, 205, 2
CELAS1, 3004, 3000, 205, 3
PELAS, 3000, 1.0E+2, 0.0
$
CDAMP1, 6001, 6000, 201, 2
CDAMP1, 6002, 6000, 201, 3
CDAMP1, 6003, 6000, 205, 2
CDAMP1, 6004, 6000, 205, 3
PDAMP, 6000, 5.0E-3
$
RBE2, 4001, 101, 23, 201
RBE2, 4002, 105, 23, 205
$

```

The following examples demonstrate the new capability to solve three common rotordynamic problems:

1. Calculation of rotor whirl frequencies and critical speeds.
2. Calculation of steady-state rotor response due to unbalance forces.
3. Calculation of rotor transient response due to spin-up of a rotor with an unbalance.

All examples will use the rotor model shown in [Figure 4-1](#).

## Example 1a: Rotor Whirl Frequencies

Rotor whirl frequencies are determined using complex modes analysis with the asynchronous option. The calculated eigenfrequencies are modes of the rotor/support structure with the rotor spinning at a specified rate. The model rotor is spinning at 10 revolutions/second. The RGYRO Bulk Data entry required for this example is:

```
$
$RGYRO RID      SYNCFLG REFROTR SPDUNIT SPDLOW  SPDHIGH SPEED
RGYRO  100      ASYNC   10      FREQ          10.0
$
$
ROTORG 10      101      THRU    105
RSPINR 10      101      102     0.0      FREQ      1.
```

The whirl frequencies for this rotor can also be determined from a theoretical solution for comparison. The results are shown below:

	Theoretical Results	MSC.Nastran Results
Modes 1 and 2	13.0 Hz	13.0 Hz
Mode 3	18.2 Hz	18.2 Hz
Mode 4	22.2 Hz	22.2Hz

## Example 1b: Rotor Critical Speeds

Rotor critical speeds can be determined using complex modes analysis by using the synchronous option. The calculated eigenfrequencies are critical speeds for the rotor. The RGYRO Bulk Data entry required for this example is:

```
$
$RGYRO RID      SYNCFLG REFROTR SPDUNIT SPDLOW  SPDHIGH SPEED
RGYRO  100      SYNC    10      FREQ      0.0      100.0
$
$
ROTORG 10      101      THRU    105
RSPINR 10      101      102     0.0      FREQ      1.
```

The critical speeds for this rotor can also be determined from a theoretical solution for comparison. The results are shown below:

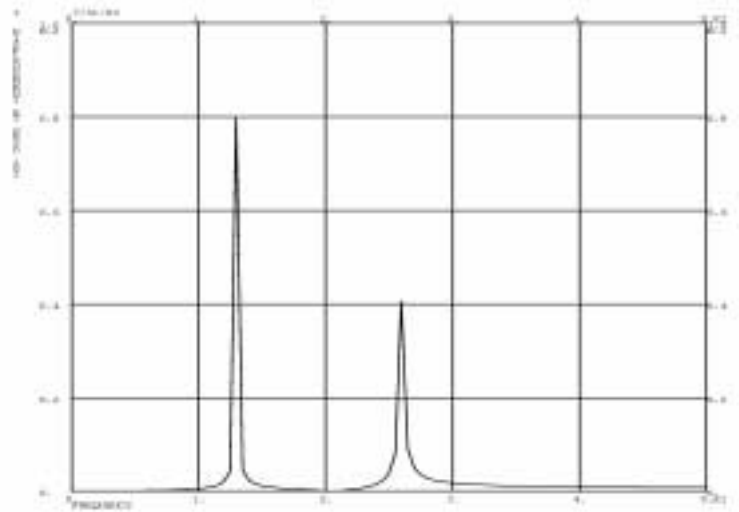
	Theoretical Results	MSC.Nastran Results
Modes 1 and 2	13.0 Hz	13.0 Hz
Mode 3	17.0 Hz	17.0 Hz
Mode 4	26.0 Hz	26.0 Hz

## Example 2: Steady-State Rotor Response due to Unbalance Forces

The steady-state response of the rotor/support structure due to unbalance forces can be determined using frequency response analysis with the synchronous option. For this example, the unbalance of  $1.0E-6\omega^2$  will be placed midway between the rotor and a support. The loading will excite the ‘bounce’ mode (rigid-body lateral translation) and the forward whirl mode of the rotor. The ‘bounce’ mode is at 13 Hz and the forward whirl mode is at 26.0 Hz (from Example 1b). The RGYRO and RLOAD1 entries used in this example are:

```
$
$RGYRO  RID      SYNCFLG REFROTR SPDUNIT SPDLOW  SPDHIGH SPEED
RGYRO    100      SYNC     10      FREQ     0.0    100.0
$
$
$
$
$
$ UNBALANCE LOAD
$
DLOAD, 1000, 1.0, 1.0, 1001, 1.0, 1002
RLOAD2, 1001, 1001, , , 1000
RLOAD2, 1002, 1002, , 1002, 1000
DAREA, 1001, 102, 2, 1.0E-6
DAREA, 1002, 102, 3, 1.0E-6
DPHASE, 1002, 102, 3, -90.0
TABLED4, 1000, 0.0, 1.0, 0.0, 1000.0
, 0.0, 0.0, 39.46, endt
$
```

The response for grid 102 is shown in the following plot:



**Figure 4-2 Steady-State Displacement Response due to Imbalance**

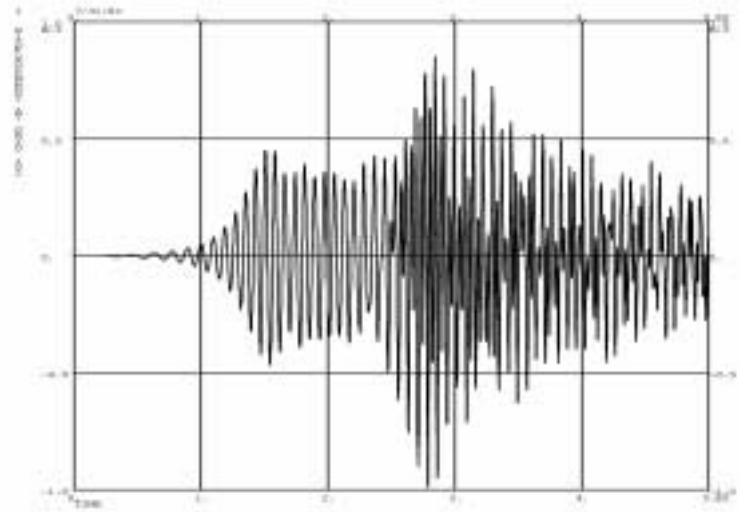
### Example 3: Response to Rotor Imbalance During Start-up

Rotor vibration due to rotor imbalance during start-up, spin-down or other transient event can be determined using either linear or nonlinear transient response analysis. For this example, the imbalance is the same used in Example 2 ( $1.0\text{E-}6\Omega^2$  at grid 102). The rotor spin rate will start at zero at  $t=0.0$  and increase linearly to 50 cycle/second at  $t=5.0$  sec. The entries used to apply the unbalance load are:

```
$
$UNBALNCRID      MASS      GRID      X1      X2      X3
UNBALNC 1000      1.0E-6    102      0.0      1.0      0.0
$
$RSPINR ROTORID  GRIDA      GRIDB      GR      SPDUNIT  TID
RSPINT  10      101      102      0.0      FREQ      1000
$
TABLED1, 1000,
,0.0, 0.0, 5.0, 50.0, ENDT
$
ROTORG  10      101      THRU      105
RSPINR  10      101      102      0.0      FREQ      1.
$
```



The response for grid 102 is shown in the following plot:



**Figure 4-3 Transient Displacement Response due to Imbalance**

Note the increase in response at approximately 1.3 sec (13 Hz) and again at 2.6 sec (26 Hz), which correspond to the 'bounce' and forward whirl frequencies respectively.

## 4.3 Equations Used in Analyses

### Frequency Response

For frequency response with asynchronous excitation,  $\Omega$  for each rotor is constant and can be determined from the rotation speed of the reference rotor,  $\Omega_{ref}$ , and relative rotation rates specified by the user. The equation of motion to be solved is:

$$\begin{aligned} & \left( -\omega^2 M + \left( i\omega B_s + B_r + \sum_{j=1}^n \Omega_j(\Omega_{ref}) B_j^G \right) \right. \\ & \left. + \left( (1 + ig)K_s + iK4_s + \sum_{j=1}^n \left( (1 + ig_{rj})K_{rj} + iK4_{rj} + \Omega_j(\Omega_{ref}) \left( K_f^{Cv} + \left( \frac{g_{rj}}{\omega} \right) K_j^{Cgr} + \left( \frac{1}{\omega} \right) K_j^{Cge} \right) \right) \right) \right) u(\omega) = F(\omega) \end{aligned} \quad \text{Eq. 4-4}$$

where the subscript  $j$  references the individual rotors.

$1/\omega$  will be determined for each excitation frequency, similar to frequency-dependent elements.

For the option to bypass the frequency-dependent lookup of rotor speeds (PARAM, GYROAVG, -1), the equation of motion to be solved is:

$$\begin{aligned} & - \left( \omega^2 M + i\omega \left( B_s + B_r + \sum_{j=1}^n \left( \Omega_j(\Omega_{ref}) B_j^G + \left( \frac{g_{rj}}{WR3} \right) K_{rj} + \left( \frac{1}{WR4} \right) K_{rj} \right) \right) \right. \\ & \left. + \left( (1 + ig)K_s + iK4_s + \sum_{j=1}^n \left( K_{rj} + \Omega_j(\Omega_{ref}) \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) \right) u(\omega) = F(\omega) \end{aligned} \quad \text{Eq. 4-5}$$

where the subscript  $j$  references the individual rotors.

WR3 and WR4 are specified using PARAM, WR3 and PARAM, WR4 respectively.

For frequency response with synchronous excitation, the excitation frequency is equal to the spin rate of the reference rotor,  $\omega = \Omega_{ref}$ . The spin rates of the additional rotors can be determined from the relative spin rates specified by the user. The equation of motion to be solved is:

$$\begin{aligned}
 & \left( -\Omega_{ref}^2 M + i\Omega_{ref} \left( B_s + B_r + \sum_{j=1}^n \Omega_j(\Omega_{ref}) B_j^G \right) \right. \\
 & \left. + \left( (1 + ig)K_s + iK4_s + \sum_{j=1}^n \left( (1 + ig_{rj})K_{rj} + iK4_{rj} + \Omega_j(\Omega_{ref}) \left( K_j^{Cv} + \left( \frac{g_{rj}}{\Omega_{ref}} \right) K_j^{Cgr} + \left( \frac{1}{\Omega_{ref}} \right) K_j^{Cge} \right) \right) \right) u(\Omega_{ref}) = F(\Omega_{ref}) \right)
 \end{aligned} \tag{Eq. 4-6}$$

where the subject  $j$  references the individual rotors.

$\Omega_j(\Omega_{ref})$  will be determined for each excitation frequency, similar to frequency-dependent elements.

For the option to bypass the frequency-dependent lookup of rotor speeds (PARAM, GYROAVG, -1),  $\Omega$  for each rotor is written as a linear function dependent on the reference rotor spin rate ( $\Omega_j(\Omega_{ref}) = \alpha_j + \beta_j \Omega_{ref}$ ). The scaling factors,  $\alpha_j$  and  $\beta_j$ , are determined from the relative spin rates specified by the user on the RSPINR entries. The  $\Omega_{ref}$  in the  $1/\Omega_{ref}$  terms are replaced by the values of user parameters WR3 and WR4. The equation of motion to be solved is:

Eq. 4-7

$$\begin{aligned}
 & \left( -\Omega_{ref}^2 \left( M - i \sum_{j=1}^n \beta_j \beta_j^G \right) \right. \\
 & \left. + i\Omega_{ref} \left( B_s + B_r + \sum_{j=1}^n \left( \alpha_j B_j^G + \left( \frac{g_{rj}}{WR3} \right) K_{rj} + \left( \frac{1}{WR4} \right) K4_{rj} - i\beta_j \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) \right. \\
 & \left. + \left( (1 + ig)K_s + iK4_s + \sum_{j=1}^n K_{rj} + \alpha_j \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) u(\Omega_{ref}) = F(\Omega_{ref})
 \end{aligned}$$

where the subscript  $j$  references the individual rotors. WR3 and WR4 are specified using PARAM, WR3 and PARAM, WR4 respectively.

## Complex Modes

For complex modes with asynchronous excitation, the spin rate  $\Omega$  for each rotor is constant and can be determined from user input and the reference rotor spin rate,  $\Omega_{ref}$ . The equation of motion to be solved is:

$$\begin{aligned} & \left( -\omega^2 M + i\omega \left( B_s + B_r + \sum_{j=1}^n \left( \Omega_j(\Omega_{ref}) B_j^G + \left( \frac{g_{rj}}{WR3} \right) K_{rj} + \left( \frac{1}{WR4} \right) K4_{rj} \right) \right) \right. \\ & \left. + \left( (1 + ig)K_s + iK4_s + \sum \left( K_{rj} + \Omega_j(\Omega_{ref}) \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) \right) u(\omega) = 0 \end{aligned} \quad \text{Eq. 4-8}$$

where the subscript  $j$  references the individual rotors.

For complex modes analysis with synchronous excitation, the excitation frequency is equal to the spin rate of the reference rotor,  $\omega = \Omega_{ref}$ .  $\Omega$  for each rotor is written as a linear function dependent on the reference rotor spin rate ( $\Omega_j(\Omega_{ref}) = \alpha_j + \beta_j \Omega_{ref}$ ). The scaling factors,  $\alpha_j$  and  $\beta_j$ , are determined from the relative spin rates specified by the user. The equation of motion to be solved is:

Eq. 4-9

$$\begin{aligned} & \left( -\Omega_{ref}^2 \left( M - i \sum_{j=1}^n \beta_j B_j^G \right) \right. \\ & + i\Omega_{ref} \left( B_s + B_r + \sum_{j=1}^n \left( \alpha_j B_j^G + \left( \frac{g_{rj}}{WR3} \right) K_{rj} + \left( \frac{1}{WR4} \right) K4_{rj} - i\beta_j \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) \\ & \left. + \left( (1 + ig)K_s + iK4_s + \sum_{j=1}^n K_{rj} + \alpha_j \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cge} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) \right) \right) u(\Omega_{ref}) = 0 \end{aligned}$$

and the subscript  $j$  references the individual rotors.

## Nonlinear Transient Response

For nonlinear transient response,  $\Omega$  is time dependent, the equation of motion to be solved is:

$$\begin{aligned} & M\ddot{u}(t) + \left( B_s + \left( \frac{g}{W3} \right) K_s + \left( \frac{1}{W4} \right) K4_s + \sum_{j=1}^n B_r + \left( \frac{g_{rj}}{WR3} \right) K_r + \left( \frac{1}{WR4} \right) K4_{rj} + \Omega_j(t) B_j^G \right) \dot{u}(t) \\ & + \left( K_s + \sum K_{rj} + \Omega_j(t) \left( K_j^{Cv} + \left( \frac{g_{rj}}{WR3} \right) K_j^{Cgr} + \left( \frac{1}{WR4} \right) K_j^{Cge} \right) + \dot{\Omega}_j(t) K_j^T \right) u(t) = F(t) \end{aligned} \quad \text{Eq. 4-10}$$

where the subscript  $j$  references the individual rotors.

The MSC.Nastran Newmark-Beta Method discretizes the following standard equation,

$$M\ddot{u}(t) + B\dot{u}(t) + F(u(t), \dot{u}(t), t) = P(u(t), \dot{u}(t), t) \quad \text{Eq. 4-11}$$

Comparing terms between and Eq. 4-11, the stiffness and gyroscopic terms are incorporated into  $F$ .

$$F(u(t), \dot{u}(t), t) = \sum_{j=1}^n (\Omega_j(t) B_j^G) \dot{u}(t) + \left( K + \sum_{j=1}^n (\Omega_j(t) K_j^{Cv} + g_{rj} K_r^{Cgr} + K_j^{Cge} + \dot{\Omega}_j(t) K_j^T) \right) u(t) \quad \text{Eq. 4-12}$$

The gyroscopic matrix,  $B^G$  in the above equations, is a skew-symmetric matrix generated with the following matrix defined in the rotor coordinate system:

$$B_r^G = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & I_{11} \\ 0 & 0 & 0 & 0 & -I_{11} & 0 \end{bmatrix} \quad \text{Eq. 4-13}$$

$I$  is determined for each grid along the axis of rotation by taking the appropriate diagonal mass term from the 6X6 mass matrix for the grid. The 6X6 gyroscopic matrix is repeated for each grid along the axis of rotation (individual  $B^G$  matrixes are placed along the diagonal of global matrix resulting in a global skew-symmetric matrix).

The viscous damping ‘circulation’ matrix,  $K^{Cv}$ , in the above equations, is based on the rotor-damping matrix. It is formed using the following equation.

$$K_j^{Cv} = (T_I B_j + B_j T_I) \quad \text{Eq. 4-14}$$

The structural damping ‘circulation’ matrices,  $K^{Cgr}$  and  $K^{Cge}$ , in the above equations, is based on the rotor-stiffness matrix. It is formed using the following equation.

$$K_j^{Cgr} = (T_I K_{rj} + K_{rj} T_I) \quad \text{Eq. 4-15}$$

$$K_j^{Cge} = (T_I K_{rj} + K_{rj} T_I) \quad \text{Eq. 4-16}$$

$T_I$  is a skew-symmetric matrix with the following form in the rotor coordinate system:

$$[T_I]_r = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0.5 & 0 & 0 & 0 \\ 0 & -0.5 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5 \\ 0 & 0 & 0 & 0 & -0.5 & 0 \end{bmatrix} \quad \text{Eq. 4-17}$$

The 6X6  $T_I$  matrix is repeated for each grid along the axis of rotation (individual  $T_I$  matrixes are placed along the diagonal of global matrix resulting in a global skew-symmetric matrix).

The tangential matrix,  $K^T$  in the above equations, is a skew-symmetric matrix generated with the following matrix defined in the rotor coordinate system:

$$K_r^T = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & I_{11} \\ 0 & 0 & 0 & 0 & -I_{11} & 0 \end{bmatrix} \quad \text{Eq. 4-18}$$

$I_{11}$  is determined for each grid along the axis of rotation by taking the appropriate diagonal mass term from the 6X6 mass matrix for the grid transformed to the rotor coordinate system. The 6X6 Tangent matrix is repeated for each grid along the axis of rotation (individual  $K_r^T$  matrixes are placed along the diagonal of global matrix resulting in a global skew-symmetric matrix).

In addition to the applied time-varying forces  $P(t)$  in [Eq. 4-11](#). The time-varying forces and moments due to a mass imbalance at grid  $k$ , in the rotor-force coordinate system, are:

$$\begin{Bmatrix} \bar{F}_1(t) \\ \bar{F}_2(t) \end{Bmatrix}_k = WTMASS \cdot \begin{Bmatrix} m_k(t)r_k(t)(\dot{\Omega}_j^2(t)\cos\theta_{jk}(t) + \dot{\Omega}_j(t)\sin\theta_{jk}(t)) \\ m_k(t)r_k(t)(\dot{\Omega}_j^2(t)\sin\theta_{jk}(t) - \dot{\Omega}_j(t)\cos\theta_{jk}(t)) \end{Bmatrix} \quad \text{Eq. 4-19}$$

$$\begin{Bmatrix} \bar{M}_1(t) \\ \bar{M}_2(t) \end{Bmatrix}_k = \text{offset}_3(t) \begin{bmatrix} 0 & -1 \\ 1 & 0 \end{bmatrix} \begin{Bmatrix} \bar{F}_1(t) \\ \bar{F}_2(t) \end{Bmatrix}_k \quad \text{Eq. 4-20}$$

where:

$m_k(t)$  = specified by the user

$r_k(t)$  = specified by the user

$\Omega_j(t)$  = specified by the user

$\dot{\Omega}_j(t)$  = is derived from  $\Omega_j(t)$

$\theta_{jk}(t) = \int_0^t \Omega_j(t) dt + \theta_k$ ,  $\theta$  measured from axis 1 to axis 2

$\theta_k$  = specified by the user

$j$  = the rotor on which the grid is located

If CFLAG on the UNBALNC entry is 'MASS' or 'BOTH', the forces calculated in [Eq. 4-19](#) are augmented to correct for the additional imbalance mass.

$$\begin{Bmatrix} F_1(t) \\ F_2(t) \\ F_3(t) \end{Bmatrix}_k = \begin{Bmatrix} \bar{F}_1(t) \\ \bar{F}_2(t) \\ \mathbf{0} \end{Bmatrix}_k + WTMASS \cdot \begin{Bmatrix} -m_k(t)\ddot{u}_1(t)_k \\ -m_k(t)\ddot{u}_2(t)_k \\ -m_k(t)\ddot{u}_3(t)_k \end{Bmatrix} \quad \text{Eq. 4-21}$$

The applied moments are also modified to correct for the additional imbalance mass.

$$\begin{Bmatrix} M_1(t) \\ M_2(t) \\ M_3(t) \end{Bmatrix}_k = \begin{Bmatrix} \bar{M}_1(t) \\ \bar{M}_2(t) \\ \mathbf{0} \end{Bmatrix}_k + WTMASS \cdot \begin{Bmatrix} -m_k(t)(r_k(t) \sin \theta_{jk}(t))^2 \ddot{\Phi}_1(t)_k \\ -m_k(t)(r_k(t) \cos \theta_{jk}(t))^2 \ddot{\Phi}_2(t)_k \\ -m_k(t)r_k^2(t) \ddot{\Phi}_3(t)_k \end{Bmatrix} \quad \text{Eq. 4-22}$$

$$+ \begin{Bmatrix} -2m_k(t)(r_k(t) \sin \theta_{jk}(t))^2 \Omega_j \dot{\Phi}_1(t)_k \\ 2m_k(t)(r_k(t) \cos \theta_{jk}(t))^2 \Omega_j \dot{\Phi}_2(t)_k \\ \mathbf{0} \end{Bmatrix} + \begin{Bmatrix} -m_k(t)(r_k(t) \sin \theta_{jk}(t))^2 \dot{\Omega}_j(t) \dot{\Phi}_1(t)_k \\ m_k(t)(r_k(t) \cos \theta_{jk}(t))^2 \dot{\Omega}_j(t) \dot{\Phi}_2(t)_k \\ \mathbf{0} \end{Bmatrix}$$

The first correction term accounts for the change in rotational inertia due to the imbalance mass, the second term corrects the gyroscopic matrix,  $B^G$ . These terms are added if CFLAG on the UNBALNC entry is 'MASS' or 'BOTH'. The third term corrects the tangent matrix,  $K^T$ . This term is added if CFLAG is 'SPEED' or 'BOTH'.

Collecting like terms in the above equation produces:

$$\begin{Bmatrix} M_1(t) \\ M_2(t) \\ M_3(t) \end{Bmatrix}_k = \begin{Bmatrix} \bar{M}_1(t) \\ \bar{M}_2(t) \\ \mathbf{0} \end{Bmatrix}_k \quad \text{Eq. 4-23}$$

$$+ WTMASS \cdot \begin{Bmatrix} -m_k(t)(r_k(t)\sin\theta_{jk}(t))^2(\ddot{\phi}_1(t)_k + 2\Omega_j(t)\dot{\phi}_1(t)_k + \dot{\Omega}_j(t)\phi_1(t)_k) \\ -m_k(t)(r_k(t)\cos\theta_{jk}(t))^2(\ddot{\phi}_1(t)_k - 2\Omega_j(t)\dot{\phi}_1(t)_k - \dot{\Omega}_j(t)\phi_2(t)_k) \\ -m_k(t)r_k^2(t)\ddot{\phi}_3(t)_k \end{Bmatrix}$$

## Static Analysis

For static analysis, in addition to the centripetal forces, the RFORCE entry will generate moments on the rotor grids due to the change of the rotor rotation vector relative to the inertial reference frame. The moments in the rotor coordinate system are calculated as follows:

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \end{Bmatrix}_r = -\Omega[I_{kk}]_r \begin{Bmatrix} \omega_1 \\ \omega_2 \\ \omega_3 \end{Bmatrix}_r \quad \text{Eq. 4-24}$$

$\Omega$  is the rotor spin rate. The rotation terms,  $(\omega_1, \omega_2, \text{ and } \omega_3)$ , are the components of the rotation vector from the RFORCE entry transformed to the rotor coordinate system.  $\omega_1, \omega_2, \omega_3$  and units are radians/unit time.

$$[I_{kk}]_r = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & I_{11} \\ 0 & -I_{11} & 0 \end{bmatrix} \quad \text{Eq. 4-25}$$

The moments are transformed back to the global coordinate system and added to the static loading vector. The moments can also be calculated in the global system using the following:



$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \end{Bmatrix}_{global} = -\Omega [I_{kk}]_{global} \begin{Bmatrix} \omega_1 \\ \omega_2 \\ \omega_3 \end{Bmatrix}_{global} \quad \text{Eq. 4-26}$$

where  $\Omega$  is the rotor spin rate,  $[I_{kk}]_g$  is the transformation of  $[I_{kk}]_r$  from rotor to global coordinates and the rotation terms,  $(\omega_1, \omega_2, \text{ and } \omega_3)_g$ , are the components of the rotation vector from the RFORCE entry in the global coordinate system.



CHAPTER

5

## Elements, Loads, and Model Checkout Tools

- CBAR/CBEAM Elements
- Nonstructural Mass Enhancements
- Enhanced Element Summary Printout (ELSUM)
- Equilibrium Checks
- New Shell Elements Corner Thickness Option
- Strength Ratio Output for Laminated Composites
- Additional Laminate Options for the PCOMP Entry
- Temperature-Dependent Composites
- Rigid Element Enhancements
- QUADR and TRIAR Elements
- CWELD Element Enhancements
- New K6ROT Default

## 5.1 CBAR/CBEAM Elements

### Introduction

Bar elements (CBAR and CBEAM Bulk Data entries) have been used for many years to model structural members that can transmit axial loads, shear forces, and bending moments. In MSC.Nastran 2004, the option to define the offsets in element coordinates has been added. The offset feature of these elements allows the user to position the beam axis relative to the line connecting the two grid points used in the definition of the element. Until now, the offset vector components had to be specified in the principal directions of the global (local) coordinate system of the two end grid points (coordinates defined on the CD fields of the grid entries).

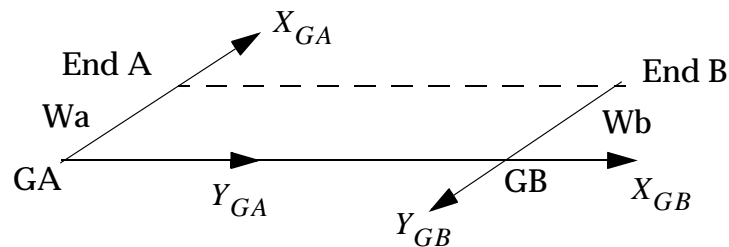
### Benefits

The specification of the offset vectors in the local coordinate systems of the grid points can be cumbersome in some modeling situations. While this method has been used successfully for many years, occasionally, it is much more convenient for MSC.Patran users and others to specify the offset vectors in the local element coordinate system. The natural definition for this local element coordinate system uses the vector connecting the two end grid points to define the x-axis and the orientation vector (sometimes referred to as the reference vector) to define the x-y plane. The y-axis lies in this plane in the direction of the orientation vector. The element z-axis is normal to the x-y plane and it is computed by taking the vector cross product of the x-axis and y-axis using the right hand rule. The definition of such a local element coordinate system imposes no restrictions on our ability to use offset vectors. The intuitive nature of the local element system and the ease with which offset vectors can be specified.

### Theory

Structural stiffeners, such as I-beams, wide flange beams, channel sections, z-sections, and hat sections are typically modeled using the bar and beam finite elements available in MSC.Nastran. When these members are used to support flat or curved panel structures, it is often desirable to include the offset effects of the member neutral/shear axis location when modeling the structure. These effects are most prevalent when the grid points are located at the mid-plane or outer surface of the panels. Under these circumstances, when a beam element is attached to these points, the line connecting the beam element end grid points does not correctly represent the element neutral axis location relative to the panel surface. To handle this situation, the CBAR and CBEAM Bulk Data entries provide for the specification of offset vectors for each end of the element that locate the positions of the element end points with respect to the locations of the element connection grid points.

Until now, the components of the offset vectors had to be specified in the principal directions of the output coordinate systems of the connection grid points. The program would then transform each of the offset vectors so that the components were measured in the basic coordinate reference system of the model. The locations of the end grid points are also converted to this basic system if necessary. Adding the offsets to the coordinates of the grid points produces the element end point locations. The vector between the two end points, together with the orientation vector can be used to generate the element coordinate system. Thus, when defining element offset vectors, the analyst must keep in mind the fact that the vectors must be specified in the output coordinate system directions of the connection grid points. For example, in [Figure 5-1](#), the local x direction at grid point GA is in the opposite direction of the local y direction at grid point GB and the local y at grid point GA and the local x at grid point GB are in the same direction. Let us assume that the element x-axis is offset 0.5 units in the local x direction at grid point GA from the line connecting grid points GA and GB. Then, in order to define the neutral axis of the element properly, the analyst must define the End A offset vector as  $W_a = \{0.5, 0.0, 0.0\}$  and at the same time specify the End B offset vector as  $W_b = \{0.0, -0.5, 0.0\}$ .



**Figure 5-1**

If we use the concept of the local element coordinate system, then no discussion of the local output coordinate systems employed at the connection grid points is necessary. For this case, the element x-axis is from GA to GB and the orientation vector is in the direction of  $X_{GA}$ , defining the direction of the element- y-axis and the x-y plane. The offset vector components will be the same at both ends and will be specified as  $W_a = W_b = \{0.0, 0.5, 0.0\}$ . The program will take care of any necessary transformations between the element system, the basic system, and the local systems at the end grid points.

## Inputs

The specification of the offset vectors for the two ends of the bar and beam elements is still accomplished using fields four through nine of the first continuation entry for the CBAR and CBEAM Bulk Data entries, respectively. The orientation (reference)

vector is specified using fields 6, 7, and 8 of the main bulk data entries. How MSC.Nastran interprets the contents of these fields depends upon the offset and reference vector specification codes supplied in field nine of the main (parent) bulk data entry. Each of the entries is described by one character. There must be three characters in the string, if it is present. The first character is associated with the end A offset vector. The second character is associated with the end B offset vector. The third character is associated with the orientation vector. The character “G” signifies that the input is specified in the Global coordinate reference system. The character “E” signifies that the input is specified in the Element local coordinate reference system. The character “B” signifies that the input is specified in the Basic coordinate reference system. Offset vectors can be defined in Global (default) or Element systems. The orientation vector can be defined in Global (default) or Basic systems. The following table describes the various valid character combinations that can be entered in this field.

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGE	Global	Global	Element
BGE	Basic	Global	Element
GEG	Global	Element	Global
BEG	Basic	Element	Global
GEE	Global	Element	Element
BEE	Basic	Element	Element

The default entry for this field is “GGG”. This indicates that all vectors are defined in the global reference system. This is the historical specification for the vector inputs. Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of EEE indicates offset vectors for end A and end B, as well as the orientation vector, are all specified in the local Element reference system. This usage results in a fatal error because the orientation vector can be specified only in Global or Basic systems but not the Element local system.

**Outputs**

There are no special output considerations when the offset vectors are specified in the local element coordinate system.

## Guidelines and Limitations

This feature is not applicable to the CBEAM entry when it is acting as a p-element beam.

## Example

A simple model is presented to demonstrate the specification of the offset vectors in the element coordinate system. The input file contains two CBAR elements that use offsets. One of the CBARs uses the original method of specifying the offsets in the global system components at the connection grid points. The second of the CBARs uses the new technique of specifying the offsets in the element coordinate system. The displacements and element forces should be the same for the two elements if the offset vectors are “equivalent”.

## Model Description

Two CBAR elements are used to demonstrate the use of the offset vectors in the element coordinate system. The grid points that define the ends of the elements are situated along the basic y-axis. The output coordinate system at each grid point is the basic coordinate system. The global system is equivalent to the basic system. The orientation vector for both elements places the element x-y plane at a thirty degree cant to the basic y-z plane toward the negative x-direction. Element 101 defines the offsets to be -0.50 units in the global (basic) x-direction and 0.866 units in the global (basic) z-direction. This is standard legacy usage of the offset vectors. Element 111 defines the offsets to be 1.0 units in the y-direction of the local element coordinate system. One end of each bar is completely constrained resulting in a cantilever beam model. Three subcases are defined to apply unit loads to the free end of the beam in each of the basic directions. Element forces are recovered for the CBAR elements for each subcase.

## Input File

```
id test,cbar
sol 101
cend
$
title = v2004 release notes demonstration problem
subtitle = test cbar offset vector specification options
label= two equivalent cbars
disp=all
spcf=all
elfo=all
spc = 100
$
```

```

$ unit load in basic x-direction
$
subcase 100
load=100
$
$ unit load in basic y-direction
$
subcase 200
load = 200
$
$ unit load in basic z-direction
$
subcase 300
load = 300
begin bulk
$
$=====
$ offset bar example.
$ grid output cstm is basic
$ offset in element coordinate system is {0.0,1.0,0.0}(t)
$ which is equivalent to {-0.5,0.0,0.866025}(t) in basic
$ element force results should be the same for both elements
$
grid,1001,,0.0,11.0,0.0
grid,1002,,0.0,20.0,0.0
$
$ Standard legacy input.
$ Element 101 specifies offsets and orientation vector in the
$ global coordinate system.
$
cbar,101,1101,1001,1002,-0.50,0.0,.866025,,+br101
+br101,,,-0.50,0.0,0.866025,-0.50,0.0,0.866025
$
$ New feature option input.
$ Element 111 specifies offsets in the local element system and
$ orientation vector in the global coordinate system.
$
cbar,111,1101,1001,1002,-0.50,0.0,.866025,eeg,+br111
+br111,,0.0,1.0,0.0,0.0,1.0,0.0
$
force,100,1002,,1000.0,1.0,0.0,0.0
force,200,1002,,1000.0,0.0,1.0,0.0
force,300,1002,,1000.0,0.0,0.0,1.0
spc1,100,123456,1001
$
$=====
$ common data
$
pbar,1101,1,1.0,1.0,1.0,1.0
mat1,1,10.7+6,,.33
enddata

```



## 5.2 Nonstructural Mass Enhancements

### Introduction

A long-standing key feature of MSC.Nastran is the ability of the PSHELL(, PCOMP), PBAR(, PBARL), PBEAM(, PBEAML, PBCOMP), PROD(, CONROD), PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D property entries to include nonstructural mass. MSC.Nastran 2004 extends this feature by allowing five new Bulk Data entries--NSM, NSM1, NSML, NSML1, and NSMADD--to distribute nonstructural mass by element lists or specific property lists associated with the above fourteen property entries.

### Benefits

In achieving mass balance, users often wish to experiment with nonstructural mass distributions. With the NSM, NSM1, and NSMADD entries along with the NSM command, nsm values can be specified across lists of elements, or properties, or both for the residual and each superelement.

Additionally, clients may have some lumped mass value they wish to distribute as nonstructural mass. With the NSML and NSML1 entries, they can smear this lumped value across lists of elements or properties or both.

### Theory

#### NSML and NSML1 Entries

The NSML and NSML1 entries compute a nonstructural mass coefficient value for “area” elements identified by an element list or property list or both by the relationship:

$$\text{NSM\_value} = \frac{\text{Lumped\_non\_structural\_mass\_value}}{\sum_{\text{elements}} \text{AREA}}$$

The NSML and NSML1 entries compute a nonstructural mass coefficient value for “line” elements identified by an element list or property list or both by the relationship:

$$\text{NSM\_value} = \frac{\text{Lumped\_non\_structural\_mass\_value}}{\sum_{\text{elements}} \text{LENGTH}}$$

In the above two expressions, AREA corresponds to the area of each individual area element (CQUAD4 for example) and LENGTH corresponds to the length of each individual length element (CBAR for example).

The NSML and NSML1 entries are then converted internally to NSM and NSM1 entries.

## Inputs

### Case Control

NSM Case Control command allows for the selection of different NSM sets for residual and superelements.

### Bulk Data Entries

NSM and its alternate form NSM1 allows the user to allocate an NSM\_value to selected sets of elements.

NSML and its alternate form NSML1 allows the user to allocate and smear a lumped nonstructural mass value to selected sets of elements.

NSMADD allows the user to form combinations of NSM, NSM1, NSML, and NSML1 sets and sum their results to selected sets of elements.

## Outputs

The resulting mass matrices will have appropriate terms accounting for additional nonstructural mass coefficients.

Case Control command

ELSUM(PID,EID,BOTH,PRINT,PUNCH,PLOT,SUMMARY)=n, ALL, or NONE prints a summary of the element mass and nonstructural mass. The EID option sorts by element type. The PID option sorts by property type. SUMMARY omits the individual element data from the PID or EID sort. See “[Enhanced Element Summary Printout \(ELSUM\)](#)” on page 213 for a more thorough description of the ELSUM command.

## Guidelines and Limitations

The following requirements are associated with the new capability:

1. The NSML and NSML1 entries cannot mix “area” and “line” elements on the same entry.

- Area elements are: CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D. (The CCONEAX is a stand alone element).
- Line elements are: CBAR, CBEAM, CBEND, CTUBE, CROD, and CONROD.
- The CCONEAX is excluded from the NSML and NSML1 entries.

## Example

The following example partial input demonstrates the form of the new entries and the expected result:

### Partial Input File

```
SOL      108  $
CEND
    DISPL = ALL
    .
    .
    .
    ELSUM = ALL
    NSM = 1222
BEGIN BULK
$
    .
    .
    .
$
$      2      3      4      5      6      7      8      9      0
NSML   9      PBEAM  211    .026   311    .028   411    .030
NSML   9      PBEND  21     .4869  31     .4869
NSML  10      PRAC2D  1     .132
NSML1  10      PBCOMP .126   21     THRU   41
NSML1  11      PBEAML .138   212    312    412
NSML   11      PROD   12     .17
NSML   12      CONROD 12     .19
NSML   12      PSHELL 51     4.5
NSML1  13      PSHELL 132.   ALL
NSML1  13      PSHELL 107.8  50     THRU   52
NSML1  14      PSHELL 7.5    57
NSML1  14      ELEMENT .594   20     30     40     201    202
        301    302    401    402
NSML1  15      PSHEAR .264   25     THRU   45     BY     10
NSML   15      PBAR   22     .22
NSML1  15      PBARL  .52    ALL
NSML   16      ELEMENT 101    .077
NSML   16      ELEMENT 27     1.021  13     .67
NSML   17      ELEMENT 14     .41
NSML   17      PTUBE  14     .43
NSML1  18      ELEMENT 16.2   41     81234
NSML1  18      PSHEAR 1.476  ALL
```

NSML	18	PSHEAR	1	1.476				
NSML1	19	ELEMENT	.112	201	402			
NSML1	19	ELEMENT	4.2	3123	6134	134		
\$								
NSMADD	1222	9	11	12	14	13	15	
	16	1718	19	2010	12			
\$								
\$								
\$		FINAL				PROPERTY		
\$		DISTRIBUTED				VALUE		
\$	ELM	NSM	ID	PID	PROPERTY	NSM		
\$	----	---	--	---	-----	----		
\$	CBAR	.061	25	22	- PBAR	.013		
\$	CBAR	.039	26	26	- PBARL	.013		
\$	CBEAM	.065	20	21	- PBCOMP	.011		
\$		.065						
\$	CBEAM	.065	30	31	- PBCOMP	.011		
\$		.065						
\$	CBEAM	.065	40	41	- PBCOMP	.011		
\$		.065						
\$	CBEAM	.085	201	211	- PBEAM	.011,.012,.014		
\$		.086 is1						
\$		.088 is2 all others zero						
\$		.088 end B						
\$	CBEAM	.058	301	311	- PBEAM	.011,.012,.014		
\$		.059 is1						
\$		.061 is2 all others zero						
\$		.061 end B						
\$	CBEAM	.059	401	411	- PBEAM	.011,.012,.014		
\$		.060 is1						
\$		.062 is2 all others zero						
\$		.062 end B						
\$	CBEAM	.0671	202	212	- PBEAML	.0111,.0112,.0113,.0114,.0115		
\$		.0672 is1						
\$		.0673 is2						
\$		.0674 is3 all others zero						
\$		.0675 is4						
\$		.0675 end B						
\$	CBEAM	.0671	302	312	- PBEAML	.0111,.0112,.0113,.0114,.0115		
\$		.0672 is1						
\$		.0673 is2						
\$		.0674 is3 all others zero						
\$		.0675 is4						
\$		.0675 end B						
\$	CBEAM	.0951	402	412	- PBEAML	.0111,.0112,.0113,.0114,.0115		
\$		.0952 is1						
\$		.0953 is2						
\$		.0954 is3 all others zero						
\$		.0955 is4						
\$		.0955 end B						
\$	CBEND	.109	27	21	- PBEND	.013		
\$	CBEND	.044	37	31	- PBEND	.013		
\$	CBEND	.013	47	41	- PBEND	.013		
\$	CONROD	.032	12	1	- CONROD	.013		
\$	CQUAD4	.257	41	57	- PCOMP	.013		

\$	CQUAD4	.178	41234	50	-	PSHELL	.013
\$	CQUAD4	.178	4123456	52	-	PSHELL	.013
\$	CQUAD4	.193	4123456751		-	PSHELL	.013
\$	CQUAD8	.178	81	50	-	PSHELL	.013
\$	CQUAD8	.259	81234	50	-	PSHELL	.013
\$	CQUAD8	.178	8123456	52	-	PSHELL	.013
\$	CQUAD8	.193	8123456751		-	PSHELL	.013
\$	CQUADR	.178	1	50	-	PSHELL	.013
\$	CQUADR	.178	1234	50	-	PSHELL	.013
\$	CQUADR	.178	123456	52	-	PSHELL	.013
\$	CQUADR	.193	1234567	51	-	PSHELL	.013
\$	CROD	.097	13	12	-	PROD	.013
\$	CSHEAR	.158	25	1	-	PSHEAR	.013
\$	CSHEAR	.158	35	1	-	PSHEAR	.013
\$	CSHEAR	.158	45	1	-	PSHEAR	.013
\$	CTRIA3	.206	3123	50	-	PSHELL	.013
\$	CTRIA3	.178	3134	50	-	PSHELL	.013
\$	CTRIA6	.178	6123	50	-	PSHELL	.013
\$	CTRIA6	.206	6134	50	-	PSHELL	.013
\$	CTRIAR	.178	123	50	-	PSHELL	.013
\$	CTRIAR	.206	134	50	-	PSHELL	.013
\$	CTUBE	.097	14	14	-	PTUBE	.013
\$							
\$							
	.						
	.						
	.						
\$							
ENDDATA							

Typical Element mass summary is given as:

## ELEMENT PROPERTY SUMMARY

ELEMENT TYPE = BAR

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
25	1	1.000000E+01	2.000000E-02	2.000000E-01	0.000000E+00	6.100000E-01	6.100000E-01	6.100000E-01
26	1	1.000000E+01	3.141593E-02	3.141593E-01	0.000000E+00	3.900000E-01	3.900000E-01	3.900000E-01
SUBTOTAL MASS =					0.000000E+00	1.000000E+00	1.000000E+00	1.000000E+00

ELEMENT TYPE = BEAM

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
20	1	2.000000E+00	1.200800E-02	2.401600E-02	0.000000E+00	1.300000E-01	1.300000E-01	1.300000E-01
30	1	2.000000E+00	1.200800E-02	2.401600E-02	0.000000E+00	1.300000E-01	1.300000E-01	1.300000E-01
40	1	2.000000E+00	1.200800E-02	2.401600E-02	0.000000E+00	1.300000E-01	1.300000E-01	1.300000E-01
201	1	2.000000E+00	2.000000E-02	4.000000E-02	0.000000E+00	1.700000E-01	1.700000E-01	1.700000E-01
202	1	2.000000E+00	1.256637E+00	2.513275E+00	0.000000E+00	1.342000E-01	1.342000E-01	1.342000E-01
301	1	2.000000E+00	2.000000E-02	4.000000E-02	0.000000E+00	1.160000E-01	1.160000E-01	1.160000E-01
302	1	2.000000E+00	1.256637E+00	2.513275E+00	0.000000E+00	1.342000E-01	1.342000E-01	1.342000E-01
401	1	2.000000E+00	2.000000E-02	4.000000E-02	0.000000E+00	1.180000E-01	1.180000E-01	1.180000E-01
402	1	2.000000E+00	1.256637E+00	2.513275E+00	0.000000E+00	1.902000E-01	1.902000E-01	1.902000E-01
SUBTOTAL MASS =					0.000000E+00	1.252600E+00	1.252600E+00	1.252600E+00

ELEMENT TYPE = BEND

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
27	1	1.570796E+01	1.000000E+00	1.570796E+01	0.000000E+00	1.712104E+00	1.712104E+00	1.712104E+00
37	1	1.570796E+01	1.000000E+00	1.570796E+01	0.000000E+00	6.911035E-01	6.911035E-01	6.911035E-01
47	1	1.570796E+01	1.000000E+00	1.570796E+01	0.000000E+00	2.042035E-01	2.042035E-01	2.042035E-01
SUBTOTAL MASS =					0.000000E+00	2.607411E+00	2.607411E+00	2.607411E+00

ELEMENT TYPE = HEXA

ID	MID	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
1001	1003	8.000000E+03	6.400000E+01	0.000000E+00	6.400000E+01	6.400000E+01
SUBTOTAL MASS =			6.400000E+01	0.000000E+00	6.400000E+01	6.400000E+01

ELEMENT TYPE = CONROD

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
12	51	1.000000E+01	1.100000E+00	1.100000E+01	6.600000E-01	3.200000E-01	9.800000E-01	9.800000E-01
SUBTOTAL MASS =					6.600000E-01	3.200000E-01	9.800000E-01	9.800000E-01

ELEMENT TYPE = QUAD4

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
41	100000057	2.000000E-01	1.000000E+02	2.000000E+01	5.04089E+01	2.570000E+01	7.610089E+01	7.610089E+01
41234	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
4123456	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
41234567	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.930000E+01	2.530000E+01	2.530000E+01
SUBTOTAL MASS =					6.840089E+01	8.060000E+01	1.490009E+02	1.490009E+02

ELEMENT TYPE = QUAD8

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
81	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
81234	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	2.590000E+01	3.190000E+01	3.190000E+01
8123456	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
81234567	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.930000E+01	2.530000E+01	2.530000E+01
SUBTOTAL MASS =					2.400000E+01	8.080000E+01	1.048000E+02	1.048000E+02

ELEMENT TYPE = QUADR

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
1	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
1234	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
123456	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.780000E+01	2.380000E+01	2.380000E+01
1234567	51	1.000000E+00	1.000000E+02	1.000000E+02	6.000000E+00	1.930000E+01	2.530000E+01	2.530000E+01
SUBTOTAL MASS =					2.400000E+01	7.270000E+01	9.670000E+01	9.670000E+01

ELEMENT TYPE = ROD

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
13	51	1.000000E+01	1.100000E+00	1.100000E+01	6.600000E-01	9.700000E-01	1.630000E+00	1.630000E+00
SUBTOTAL MASS =					6.600000E-01	9.700000E-01	1.630000E+00	1.630000E+00

ELEMENT TYPE = SHEAR

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
25	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	3.860000E-01	3.860000E-01	3.860000E-01
35	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	3.860000E-01	3.860000E-01	3.860000E-01
45	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	3.860000E-01	3.860000E-01	3.860000E-01
125	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	7.900000E-01	7.900000E-01	7.900000E-01
135	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	7.900000E-01	7.900000E-01	7.900000E-01
145	1	1.000000E+00	4.000000E+00	4.000000E+00	0.000000E+00	7.900000E-01	7.900000E-01	7.900000E-01
SUBTOTAL MASS =					0.000000E+00	3.528000E+00	3.528000E+00	3.528000E+00

ELEMENT TYPE = TRIA3

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3123	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	1.030000E+01	1.330000E+01	1.330000E+01
3134	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	8.900000E+00	1.190000E+01	1.190000E+01
SUBTOTAL MASS =					6.000000E+00	1.920000E+01	2.520000E+01	2.520000E+01

ELEMENT TYPE = TRIA6

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
6123	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	8.900000E+00	1.190000E+01	1.190000E+01
6134	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	1.030000E+01	1.330000E+01	1.330000E+01
SUBTOTAL MASS =					6.000000E+00	1.920000E+01	2.520000E+01	2.520000E+01

ELEMENT TYPE = TRIAR

ID	MID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
123	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	8.900000E+00	1.190000E+01	1.190000E+01
134	51	1.000000E+00	5.000000E+01	5.000000E+01	3.000000E+00	1.030000E+01	1.330000E+01	1.330000E+01
SUBTOTAL MASS =					6.000000E+00	1.920000E+01	2.520000E+01	2.520000E+01

ELEMENT TYPE = TUBE

ID	MID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
14	51	1.000000E+01	3.141593E-01	3.141593E+00	1.884956E-01	9.700000E-01	1.158496E+00	1.158496E+00
SUBTOTAL MASS =					1.884956E-01	9.700000E-01	1.158496E+00	1.158496E+00

TOTAL MASS =					1.999094E+02	3.023480E+02	5.022575E+02	5.022575E+02
--------------	--	--	--	--	--------------	--------------	--------------	--------------

## 5.3 Enhanced Element Summary Printout (ELSUM)

### Introduction

The ELSUM Case Control command controls the generation of a printed table of informational properties for the various element types present in the input data file. The information produced includes element measures (e.g. length, thickness, area, volume) and mass property data. The information is grouped according to the finite element type. Several new options have been added to the ELSUM command giving the users more control over the amount of output produced and another view of the data grouped by element property.

### Benefits

The enhancements to the ELSUM Case Control command allow for selection of output summaries grouped by element type or by element property type. Both groupings may be selected at the same time. In addition, a new summary only option limits the output to only the mass property sub-totals. No individual element property output is generated. This reduces the amount of information that must be sifted if only the totals for an element type are of interest. With the new grouping by element property type, mass property output can now be summarized for all of the elements that reference property ids within each property type.

### Theory

For each element present in the model, physical metrics appropriate to the element type are computed. These metrics may include length, thickness, area, and volume. Mass properties are computed for supported element types. For the CONM1 element, the mass is taken to be the average of the M11, M22 and M33 values on the CONM1 Bulk Data entry. In addition, references to material properties and element properties are gathered for each element. For bar, beam and shell elements, an attempt is made to determine whether the property reference is to a basic property entry, or has been derived from one of the available alternate input entries (PBARL, PBEAML, PBCOMP, PCOMP). If it is determined that the property is derived, a character (L for PBARL or PBEAML, P for PBCOMP or PCOMP) is appended to the property identification number on output. For the case of the element type grouping, each element type is summarized in turn. For each element type, information for each element is displayed. Sub-totals are generated for mass property information. For the case of the property type grouping, each property type referenced by an element type is summarized in turn. All of the property ids referenced within the type are

summarized in ascending property id order. Within this summary, information for each element referencing the property id is displayed. Various subtotals are accumulated and displayed.

## Inputs

The ELSUM Case Control command has been enhanced to give the user new controls over the amount and ordering of the element summary information produced. The general format of the command is:

$$\text{ELSUM}([\text{EID}, \text{PID}, \text{BOTH}, \text{PIDSUM}, \text{EIDSUM}]) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

The complete description of the ELSUM command can be found in “[Statements, Commands, Entries, and Parameters](#)” on page 621. The EID and PID keywords are used to select the particular grouping desired for the element summary output. The EID keyword requests grouping by element type while the PID keyword requests grouping by element property type. The BOTH keyword requests both groupings. The PIDSUM keyword requests that only mass property totals be output for the PID grouping. The EIDSUM keyword requests that only mass property totals be output for the EID grouping.

## Outputs

The ELSUM Case Control command causes printed output to be generated. [Figure 5-2](#) is an example of the EID grouping output. [Figure 5-3](#) is an example of the PID grouping output. If PIDSUM or EIDSUM were used, then only the subtotals and totals information would be present in the figures.



E L E M E N T   P R O P E R T Y   S U M M A R Y (BY ELEMENT TYPE / ID)									
ELEMENT TYPE = BAR									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3401	3401	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
3403	3403	1	3.00000E+00	1.00000E+00	3.00000E+00	1.50000E-01	1.50000E+00	1.65000E+00	1.65000E+00
3410	3403	1	2.00000E+00	1.00000E+00	2.00000E+00	1.00000E-01	1.00000E+00	1.10000E+00	1.10000E+00
17101	17103 L	1	1.00000E+00	1.01034E+00	1.01034E+00	5.05168E-02	0.00000E+00	5.05168E-02	5.05168E-02
17103	17103 L	1	3.00000E+00	1.01034E+00	3.03101E+00	1.51550E-01	0.00000E+00	1.51550E-01	1.51550E-01
17110	17103 L	1	2.00000E+00	1.01034E+00	2.02067E+00	1.01034E-01	0.00000E+00	1.01034E-01	1.01034E-01
SUBTOTAL MASS FOR ALL BAR						6.03101E-01	3.00000E+00	3.60310E+00	3.60310E+00
.									
.									
.									
ELEMENT TYPE = BEAM									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
200	200	1	1.00000E+00	1.00000E+02	1.00000E+02	5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00
17000	17000 L	1	1.00000E+00	9.99987E+01	9.99987E+01	4.99994E+00	0.00000E+00	4.99994E+00	4.99994E+00
SUBTOTAL MASS FOR ALL BEAM						9.99994E+00	0.00000E+00	9.99994E+00	9.99994E+00
.									
.									
.									
ELEMENT TYPE = PENTA									
ELEM ID	PROP ID	MATL ID			VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
6801	6801	2			4.00000E-02	2.00000E-03	0.00000E+00	2.00000E-03	2.00000E-03
SUBTOTAL MASS FOR ALL PENTA						2.00000E-03	0.00000E+00	2.00000E-03	2.00000E-03
.									
.									
.									
ELEMENT TYPE = QUAD4									
ELEM ID	PROP ID	MATL ID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3301	3301	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
3313	3313	1	6.12009E-01	6.36864E+00	3.89766E+00	1.94883E-01	1.91715E+00	2.11203E+00	2.11203E+00
3321	3321 P	3321	1.00000E+00	1.00000E+02	1.00000E+02	1.00000E+02	0.00000E+00	1.00000E+02	1.00000E+02
17503	17503	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	0.00000E+00	5.00000E-02	5.00000E-02
SUBTOTAL MASS FOR ALL QUAD4						1.00295E+02	2.41715E+00	1.02712E+02	1.02712E+02
.									
.									
.									
ELEMENT TYPE = ROD									
ELEM ID	PROP ID	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
101	101	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
107	101	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01
SUBTOTAL MASS FOR ALL ROD						1.00000E-01	1.00000E+00	1.10000E+00	1.10000E+00
.									
.									
.									
ELEMENT TYPE = TETRA									
ELEM ID	PROP ID	MATL ID			VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS
3901	3901	1			5.77350E-01	2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02
SUBTOTAL MASS FOR ALL TETRA						2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02
.									
.									
.									
TOTAL MASS FOR ALL SUPPORTED ELEMENT TYPES						1.38647E+02	1.53344E+01	1.53981E+02	1.53981E+02

Figure 5-2 Sample Element Summary by Element Type Output

E L E M E N T   P R O P E R T Y   S U M M A R Y										(BY PROPERTY TYPE / ID)
PROPERTY TYPE = PBAR, ID = 3401 *****										
ELEM ID	ELEM TYPE	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
3401	BAR	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
SUBTOTAL MASS FOR ALL BAR			ELEMENTS	FOR PBAR, ID =	3401	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
.										
.										
.										
PROPERTY TYPE = PBEAM, ID = 200 *****										
ELEM ID	ELEM TYPE	MATL ID	LENGTH	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
200	BEAM	1	1.00000E+00	1.00000E+02	1.00000E+02	5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00	
SUBTOTAL MASS FOR ALL BEAM			ELEMENTS	FOR PBEAM, ID =	200	5.00000E+00	0.00000E+00	5.00000E+00	5.00000E+00	
.										
.										
.										
PROPERTY TYPE = PSOLID, ID = 3901 *****										
ELEM ID	ELEM TYPE	MATL ID			VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
3901	TETRA	1			5.77350E-01	2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02	
SUBTOTAL MASS FOR ALL SOLID			ELEMENTS	FOR PSOLID, ID =	3901	2.88675E-02	0.00000E+00	2.88675E-02	2.88675E-02	
.										
.										
.										
PROPERTY TYPE = (NONE) *****										
ELEM ID	ELEM TYPE	MATL ID				STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
2901	CONM1	0				1.00000E+00	0.00000E+00	1.00000E+00	1.00000E+00	
2904	CONM1	0				2.20000E+00	0.00000E+00	2.20000E+00	2.20000E+00	
SUBTOTAL MASS FOR ALL CONM1			ELEMENTS			3.20000E+00	0.00000E+00	3.20000E+00	3.20000E+00	
.										
.										
.										
PROPERTY TYPE = PSHELL, ID = 3301 *****										
ELEM ID	ELEM TYPE	MATL ID	THICKNESS	AREA	VOLUME	STRUCT.MASS	NON-STR.MASS	TOTAL MASS	TM*WTMASS	
3301	QUAD4	1	1.00000E+00	1.00000E+00	1.00000E+00	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
SUBTOTAL MASS FOR ALL SHELL			ELEMENTS	FOR PSHELL, ID =	3301	5.00000E-02	5.00000E-01	5.50000E-01	5.50000E-01	
.										
.										
.										
TOTAL MASS FOR ALL SUPPORTED ELEMENT TYPES						=====	=====	=====	=====	
						1.38647E+02	1.53344E+01	1.53981E+02	1.53981E+02	
						=====	=====	=====	=====	

Figure 5-3 Sample Element Summary by Element Property Type Output

## Guidelines and Limitations

1. The ELSUM command keywords can be used in any combination. However, if both EID and EIDSUM are present, EIDSUM takes precedence over EID and only the mass property totals summary will be generated for the grouping by element type. This is also true for the PID and PIDSUM keywords. PIDSUM takes precedence over PID and only the mass property totals will be generated for the grouping by element property type.
2. Only the following element types produce mass property information: CBAR, CBEAM, CBEND, CHEXA, CMASSi, CONM1, CONM2, CONROD, CPENTA, CQUAD4, CQUAD8, CQUADR, CRAC2D, CRAC3D, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, CTRIAX6, and CTUBE.
3. The ELSUM command is ignored in heat transfer solution sequences.

## Example

No new example is presented for this enhanced ELSUM Case Control command because any existing input data file available can be used to demonstrate the new features. One simply inserts an ELSUM command with the desired keywords into the Case Control Section of the input file. Element summary information will be generated and placed on the standard .f06 output file.

## 5.4 Equilibrium Checks

### Introduction

Many analysts employ various techniques to ensure the validity and integrity of their finite element analysis solutions. Over the years, many of these techniques have become integrated into the MSC.Nastran solution sequences. One of the most widely used methods in static analysis is the equilibrium balance check. For MSC.Nastran 2004, a simple user interface is introduced to produce a summation of the vector resultants (which should be zero if the solution is in equilibrium). Prior to MSC.Nastran 2004, analysts performed this step using hand calculations and the individual LOAD, SPCF and MPCF resultants. The new EQUILIBRIUM Case Control command automates this procedure.

### Benefits

Checking the equilibrium conditions at various steps is a simple and effective tool that can be used to detect modeling difficulties and evaluate the quality of the solution. For example, a column of the stiffness matrix of a structure is a set of forces that develop when the degree of freedom represented by that column is given a unit displacement. If the resultant of those forces about any reference point is calculated, it should be zero if no unintentional constraints are imposed. It is also useful to compute and compare the resultants of the applied loads and the constraint forces (single point as well as multi-point). If they balance to zero, confidence in the solution is increased.

The application of equilibrium checking techniques is currently available in MSC.Nastran through using the VECPLOT module options and other matrix operators requiring DMAP intervention.

### Theory

The EQUILIBRIUM Case Control command requests the new capability. The command can be used to specify a grid point to be used as the reference point for the summation. Each subcase can specify a different point, if desired. If the point specified does not exist, the input vector resultants are used without modification. The origin of the basic coordinate system may also be specified as the reference point.

The VECPLOT module performs the tasks associated with generation of the equilibrium results. The input to the module consists of previously computed force vector resultants produced using either VECPLOT module processing option number one or number eight. This option transforms the input force vectors to the basic

system and computes the resultants with respect to the grid point specified by the GRDPNT DMAP parameter. If that point does not exist, the origin of the basic coordinate system is used instead.

The presence of the EQUILIBRIUM Case Control command in the Case Control Section causes the VECPLOT module to produce an output data block suitable for input into the OFP module for print and punch file processing. The data block contains the force resultants. The VECPLOT module also produces a summary listing of the force resultants. These formats are shown in subsequent sections.

The equilibrium command generates a total summation that includes the effects of all available force vectors. These force vectors are the applied loads, single point constraint forces and multi-point/rigid element force vectors. These vectors are produced by the presence of the associated Case Control commands, LOAD (for applied loads), SPCF (for single point constraint forces) and MPCF (for multi-point/RBE constraint forces). If an EQUILIBRIUM command is present in the Case Control Section for a subcase, then the program ensures computation of the SPCF and MPCF for the subcase.

## Inputs

The new EQUILIBRIUM Case Control command is introduced to activate the feature. The Case Control command format is as follows:

$$\text{EQUILIBRIUM} \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) = \left\{ \begin{array}{c} \text{YES} \\ \text{gridid} \\ \text{NONE} \end{array} \right\}$$

The command can be used to specify a grid point that is to be used as the reference point for the summation. Each subcase can specify a different point if desired. If the point specified does not exist, the input vector resultants are used without modification. The YES request implies that the origin of the basic coordinate system be used.

## Outputs

There are two sets of output that can be generated by the EQUILIBRIUM command. The first set of output is produced by the VECPLOT module and consists of only a printed summary. The second set of output is produced by the Output File Processor module. The output produced by the VECPLOT module is limited to printed results (no punched output) even though the case control command format is cast in the “standard” form. VECPLOT output is as follows:

E Q U I L I B R I U M   S U M M A T I O N S   I N   B A S I C   C O O R D I N A T E   S Y S T E M								
SUBCASE ID	TYPE	SET ID	T1	T2	T3	R1	R2	R3
1001	LOAD	100	1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	3.000000E+03	-2.000000E+03
	SPCF	200	-1.000000E+03	-3.179752E-13	-4.128382E-12	-6.889597E-12	-3.000000E+03	2.000000E+03
TOTALS WRT POINT		101	0.000000E+00	-3.179752E-13	-4.128382E-12	-6.889597E-12	0.000000E+00	0.000000E+00
3001	LOAD	300	0.000000E+00	0.000000E+00	1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00
	SPCF	200	1.215391E-14	-3.839810E-15	-1.000000E+03	0.000000E+00	-3.646172E-14	3.198743E-14
TOTALS WRT POINT		302	1.215391E-14	-3.839810E-15	0.000000E+00	-1.151943E-14	-3.646172E-14	3.198743E-14

The VECPLOT module will produce an output data block that is suitable for input into the OFP module. The Case Control print/punch requests will be processed at this point. The OFP format is as follows:

E Q U I L I B R I U M   C H E C K								
RESULTANT LOADS IN BASIC COORDINATE SYSTEM								
SUBCASE NO.	REFERENCE POINT	LOAD TYPE	T1	T2	T3	R1	R2	R3
1001	101	APP-LOAD	1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	3.000000E+03	-2.000000E+03
		F-OF-SPC	-1.000000E+03	-3.179752E-13	-4.128382E-12	-6.889597E-12	-3.000000E+03	2.000000E+03
*TOTALS*			0.000000E+00	-3.179752E-13	-4.128382E-12	-6.889597E-12	0.000000E+00	0.000000E+00
2001	ORIGIN	APP-LOAD	0.000000E+00	1.000000E+03	0.000000E+00	-3.000000E+03	0.000000E+00	4.000000E+03
		F-OF-SPC	-2.343780E-29	-1.000000E+03	-1.013140E-11	3.000000E+03	4.052559E-11	-4.000000E+03
*TOTALS*			-2.343780E-29	0.000000E+00	-1.013140E-11	0.000000E+00	4.052559E-11	0.000000E+00

## Guidelines and Limitations

The use of the EQUILIBRIUM Case Control command is limited to static analysis.

## 5.5 New Shell Elements Corner Thickness Option

### Shell Elements Corner Thickness Specification

#### Introduction

The CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, and CTRIAR elements allow the user to specify either corner element thickness or a single thickness on the PSHELL entry. A TFLAG field (field 13) has been added to the connection entries of each of the above shell elements that allows the user to specify the corner thickness as a fraction relative to the T value of the PSHELL entry.

#### Benefits

In forming shell like structures, quite often the taper at the corners is really just a function of some average thickness. With this option, the average thickness can be specified as T on the PSHELL and the corner thickness will be computed as a fraction of the T on the PSHELL. In SOL 200, the T on the PSHELL can be specified on the DVPREL1 with average thickness specified using a DESVAR entry and the shell will be tapered accordingly.

#### Theory

If TFLAG=1 the following calculation is used for each corner:

$$T_i = T_i \times T$$

#### Inputs

If the fraction relative to the T value of the PSHELL is desired, TFLAG=1 should be specified on the element connection entry.

#### Outputs

Standard shell element output is obtained.

#### Guidelines and Limitations

This option should be useful in design optimization.

## Example

The following example input demonstrates the form and use of the new entries:

### Input File

```
diag 8,15,56
sol 200
cend
$
title = Cantilever Plate: Optimization
subtitle = Relative Thickness
$
analysis = statics
$
desobj = 1000
dessub = 2000
$
dsaprt
$
load = 10
spc = 20
$
disp = all
stress = all
$
begin bulk
$
param, grdpnt, 0
$
drespl, 1000, weight, weight
$
drespl, 2000, disp, disp,,, 2,, 5
dconstr, 2000, 2000, -0.5
$
doptprm, desmax, 15
$
$ Design Model: Relative Thickness
$
desvar, 1, t1, 1., .001
desvar, 2, t2, 1., .001
desvar, 3, t3, 1., .001
$
dvprell, 100, pshell, 1, t
, 1, 0.1
dvprell, 200, pshell, 2, t
, 2, 0.2
dvprell, 300, pshell, 3, t
, 3, 0.1
$
$ Test Plate: Sheet Metal Thickness Variation
$
$ Relative Thicknesses on CQUAD4/CTRIA3 Entries
```



```

$
$ -----
$
grid, 1,, 0., 0., 0.
= ,*(1),, *(1.), ==
=(3)
grid, 11,, 0., 1., 0.
= ,*(1),, *(1.), ==
=(2)
grid, 21,, 0., 2., 0.
= ,*(1),, *(1.), ==
=(3)
$
cquad4, 1, 1, 1, 2, 12, 11,
, , 1, 1., 1.1, 1.1, 1.
cquad4, 2, 1, 2, 3, 13, 12,
, , 1, 1.1, 1.2, 1.3, 1.1
cquad4, 3, 1, 3, 4, 14, 13,
, , 1, 1.2, 1.1, 1.6, 1.3
$
pshell, 1, 1, 0.1, 1
$
cquad4, 11, 2, 11, 12, 22, 21
, , 1, 1.0, 1.1, 1.1, 1.0
cquad4, 12, 2, 12, 13, 23, 22
, , 1, 1.1, 1.0, 1.2, 1.1
cquad4, 13, 2, 13, 14, 24, 23
, , 1, 1.0, 0.8, 1.1, 1.2
$
pshell, 2, 1, 0.2, 1
$
ctria3, 21, 3, 4, 5, 14
, , 1, 1.1, 1.0, 1.6
ctria3, 22, 3, 5, 25, 14
, , 1, 1.0, 2.0, 1.6
ctria3, 23, 3, 14, 25, 24
, , 1, 1.6, 2.0, 2.2
$
pshell, 3, 1, 0.1, 1
$
mat1, 1, 10000.,, 0.3, 1.
$
force, 10, 5,, 10., 0., -1.
force, 10, 25,, 10., 0., -1.
$
spc1, 20, 12, 1, 11, 21
spc1, 20, 3456, 1, thru, 25
$
enddata

```

Partial Output File

SUMMARY OF DESIGN CYCLE HISTORY

\*\*\*\*\*

(HARD CONVERGENCE ACHIEVED)

(SOFT CONVERGENCE ACHIEVED)

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED5

NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS4

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		1.299167E+00		-4.583377E-02
1	1.088284E+00	1.088204E+00	7.350594E-05	-1.224685E-02
2	1.067696E+00	1.067703E+00	-6.364063E-06	1.266837E-03
3	1.065613E+00	1.065610E+00	2.349259E-06	2.949357E-03
4	1.065610E+00	1.065610E+00	0.000000E+00	2.949357E-03

\*

\*\*\*\*\*

1	CANTILEVER PLATE: OPTIMIZATION	MARCH	18, 2003	MSC.NASTRAN	3/18/03	PAGE	40
	RELATIVE THICKNESS						
0							

DESIGN VARIABLE HISTORY

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL	:	1	:	2	:	3	:	4	:	5	:
1	1	T1	1.0000E+00	:	1.2538E+00	:	1.3250E+00	:	1.3398E+00	:	1.3398E+00	:		
2	2	T2	1.0000E+00	:	8.2646E-01	:	7.5902E-01	:	7.5085E-01	:	7.5085E-01	:		
3	3	T3	1.0000E+00	:	3.8964E-01	:	3.8075E-01	:	3.7388E-01	:	3.7388E-01	:		

\*\*\* USER INFORMATION MESSAGE 6464 (DOM12E)

RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER =4.

## 5.6 Strength Ratio Output for Laminated Composites

### Introduction

For laminated composites, Strength Ratio (SR) is a direct failure indicator compared to Failure Index (FI) which indicates only if failure occurs. Generally, Strength Ratio is defined as:

$$\text{Strength Ratio (SR)} = \text{Allowable Stress} / \text{Calculated Stress}$$

For example a SR = 1.2 indicates that the applied loads can be increased by 20% before failure occurs. A FI = 0.8 indicates that failure has not occurred and does not indicate 20% safety margin. Therefore the SR is a much more practical design indicator for analysis and strength-criteria based design.

This new resultant quantity is available for all solution sequences that support composites data recovery.

### Theory

For Maximum Stress/Strain and Transverse Stress theories, the Strength Ratio (SR) is simply the inverse of the FI. For the quadratic failure theories such as Hill, Hoffman, and Tsai-Wu, the SR is calculated by solving the quadratic equation with the FI set to unity and replacing the applied stress with the SR · applied stress.

The SR expression (see References [11.](#) and [12.](#)) for each Failure Criteria is defined.

### HILL Failure Criteria

The HILL failure criteria is defined as:

$$FI = \left( \frac{\sigma_1^2}{X^2} \right) + \left( \frac{\sigma_2^2}{Y^2} \right) - \left( \frac{\sigma_1 \sigma_2}{X^2} \right) + \left( \frac{\sigma_{12}^2}{S^2} \right) \quad \text{Eq. 5-1}$$

where:

$X = X_T$  if  $\sigma_1$  is Tensile

$X = X_C$  if  $\sigma_1$  is Compressive

$Y = Y_T$  if  $\sigma_2$  is Tensiles

$Y = Y_C$  if  $\sigma_2$  is Compressive

$S$  is Allowable shear stress

$\sigma_1$  is Ply longitudinal stress

$\sigma_2$  is Ply transverse stress

$\sigma_{12}$  is Ply shear stress

For the product term:

$$\frac{\sigma_1, \sigma_2}{X^2}$$

$X = X_T$  if  $\sigma_1$  and  $\sigma_2$  are of the same sign.

$X = X_C$  if  $\sigma_1$  and  $\sigma_2$  are of different signs.

The FI expression can be redefined in terms of a strength ratio. The roots of the resulting quadratic expression are the reserve factors, the first one for the stresses as applied and the second for when the sign is reversed for all stress components.

The FI expression is re-arranged with the applied stress replaced by a ratio SR times the applied stress and equated to a FI of unity, the SR can now be determined.

$$1.0 = [(\text{SR}^2 \sigma_1^2) \cdot (1/X^2)] + [(\text{SR}^2 \sigma_2^2) \cdot (1/Y^2)] - \left[ \frac{(\text{SR}^2 \sigma_1 \sigma_2)}{(S^2)} \right] + \left[ \frac{(\text{SR}^2 \sigma_{12}^2)}{(S^2)} \right] \quad \text{Eq. 5-2}$$

$$[(\sigma_1^2/X^2) + (\sigma_2^2/Y^2) - (\sigma_1 \sigma_2/X^2) + (\sigma_{12}^2/Y^2)] \text{SR}^2 - 1 = 0$$

$$a \text{SR}^2 + b \text{SR} + c = 0$$

from the general solution of a quadratic equation,

$$\text{SR}_1 = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \quad \text{and} \quad \text{SR}_2 = \frac{-b - \sqrt{b^2 - 4ac}}{2a}$$

For the HILL criteria,

$$a = (\sigma_1^2/X^2) + (\sigma_2^2/Y^2) - (\sigma_1 \sigma_2)/X^2 + (\sigma_{12}^2/S^2)$$

$$b = 0.0$$

$$c = -1$$

$$SR_1 = \frac{-b + \sqrt{b^2 - 4ac}}{2a}$$

$$SR_1 = \frac{\sqrt{-4 \cdot a \cdot 1.0}}{2 \cdot a} = \frac{2 \cdot \sqrt{a}}{2 \cdot a}$$

$$SR_1 = \frac{1}{\sqrt{a}}$$

$$SR_2 = -\frac{1}{\sqrt{a}}$$

$$\text{where } a = (\sigma_1^2 / X^2) + (\sigma_2^2 / Y^2) - (\sigma_1 \sigma_2 / X^2) + (\sigma_{12}^2 / S^2) = FI$$

Therefore, for the HILL quadratic failure criteria, the SR can be directly derived from the FI:

$$SR = \frac{1}{\sqrt{FI}}$$

## HOFFMAN Failure Criteria

Unlike the Hill criteria, the Hoffman failure criteria takes into account the differences in tensile and compressive strengths in the longitudinal and transverse directions of the ply.

$$FI = [1 / (X_T \cdot X_C)] \sigma_1^2 + [1 / (Y_T \cdot Y_C)] \sigma_2^2 + [1 / (X_T \cdot X_C)] (\sigma_1 \sigma_2) + (\sigma_{12}^2 / S^2) \\ + [(1 / X_T) - (1 / X_C)] \sigma_1 + [(1 / Y_T) - (1 / Y_C)] \sigma_2$$

defining:

$$F_1 = (1 / X_T) - (1 / X_C)$$

$$F_2 = (1 / Y_T) - (1 / Y_C)$$

$$F_{11} = 1 / (X_T \cdot X_C)$$

$$F_{22} = 1 / (Y_T \cdot Y_C)$$

$$F_{66} = 1 / (S^2)$$

$$FI = F_{11}\sigma_1^2 + F_{22}\sigma_2^2 - F_{11}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2 + F_1\sigma_1 + F_2\sigma_2$$

Now introduce applied stress as a ratio, R times applied stress equated to a FI of unity.

$$1.0 = F_{11}SR^2\sigma_1^2 + F_{22}SR^2\sigma_2^2 - F_{11}SR^2(\sigma_1\sigma_2) + F_{66}SR^2\sigma_{12}^2 + F_1SR\sigma_1 + F_2SR\sigma_2$$

Arrange in quadratic form

$$[F_{11}\sigma_1^2 + F_{22}\sigma_2^2 - F_{11}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2]R^2 + [F_1\sigma_1 + F_2\sigma_2]R - 1 = 0$$

$$SR_1 = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \quad \text{and} \quad SR_2 = \frac{-b - \sqrt{b^2 - 4ac}}{2a}$$

where:

$$a = [F_{11}\sigma_1^2 + F_{22}\sigma_2^2 - F_{11}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2]$$

$$b = [F_1\sigma_1 + F_2\sigma_2]$$

$$c = -1.0$$

This is now a quadratic equation that does not simplify in the same way as the Hill criteria; such that there is no direct relationship between FI and SR. The roots of the equation have to be determined to obtain the SR.

## TSAI-Wu Failure Criteria

The Tsai-Wu failure criterion is very similar to the Hoffman criteria, but has a different value for the coefficient associated with the interaction of the direct stresses. In the Hoffman criterion it is the same as the coefficient associated with the direct longitudinal stresses.

For the Tsai-Wu criterion:

$$FI = F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{12}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2 + F_1\sigma_1 + F_2\sigma_2$$

where:

$$F_1 = (1/X_T) - (1/X_C)$$

$$F_2 = (1/Y_T) - (1/Y_C)$$

$$F_{11} = 1/(X_T \cdot X_C)$$

$$F_{22} = 1/(Y_T \cdot Y_C)$$

$$F_{66} = 1/(S^2)$$

$$F_{12} = -0.5\sqrt{F_{11} \cdot F_{22}}$$

Now introduce applied stress as a ratio R times applied stress equated to a FI of unity.

$$1.0 = F_{11}SR^2\sigma_1^2 + F_{22}SR^2\sigma_2^2 + F_{12}SR^2(\sigma_1\sigma_2) + F_{66}SR^2\sigma_{12}^2 + F_1SR\sigma_1 + F_2SR\sigma_2$$

Arrange in quadratic form

$$[F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{12}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2]SR^2 + [F_1\sigma_1 + F_2\sigma_2]SR - 1 = 0$$

$$SR_1 = \frac{-b + \sqrt{b^2 - 4ac}}{2c} \quad \text{and} \quad SR_2 = \frac{-b - \sqrt{b^2 - 4ac}}{2c}$$

where:

$$a = [F_{11}\sigma_1^2 + F_{22}\sigma_2^2 - F_{12}(\sigma_1\sigma_2) + F_{66}\sigma_{12}^2]$$

$$b = [F_1\sigma_1 + F_2\sigma_2]$$

$$c = -1.0$$

This is a quadratic equation that does not simplify in the same way as the Hill criteria such that there is no direct relationship between FI and SR. The roots of the equation have to be determined to obtain the SR.

## Maximum Stress (Strain)

Failure Index FI = Calculated Stress / Allowable Stress

By substituting the Allowable stress by RF\*Allowable Stress and setting FI to unity:

$$SR = 1/(FI)$$

## Transverse Shear Stress

For this case, the Strength Ratio is simply the inverse of the failure index.

## User Interface

A new parameter (PARAM,SRCOMPS,YES/NO) requests Strength Ratio (SR) output. SR Output requires specification of a failure theory and allowable stress/strain values. Note that SRs will be computed for plies with an output request (PCOMP-SOUTi field).



## 5.7 Additional Laminate Options for the PCOMP Entry

### Introduction

The PCOMP Bulk Data entry is used to provide input data for the properties of an n-ply composite material laminate. As such, it is widely used in aerospace applications where a composite structure is built up from a number of discrete sheets of material, such as a carbon-epoxy fiber. The utility of the entry was enhanced in MSC.Nastran 2001 by providing additional options to characterize the way the laminate input was to be interpreted. (See pps. 91-94 the *MSC.Nastran 2001 Release Guide* for descriptions of the MEM and BEND options for the LAM field of the PCOMP entry). This capability is extended further in MSC.Nastran 2004 to provide two additional options for the LAM field:

1. The SMEAR options smears the ply effects while ignoring stacking sequence.
2. The SMCORE option enables the modeling of a skin/core combination where the ply effects are smeared in the skin, the core inertial terms are treated exactly, and the core stiffness terms are ignored.

### Benefits

The primary motivation behind the additional PCOMP features is to support the use of composite materials in a preliminary design stage, where stacking sequence effects are considered secondary and would impede the development of a high quality candidate design. This is particularly useful when the composite description is used with an automated design procedure such as SOL 200 in MSC.Nastran or a client in-house procedure. Currently, stacking sequence impacts the stiffness of the laminate and therefore, the results. In an automated design context, it is hoped that the effects on the results are small, but this cannot be guaranteed. The SMEAR option is a compromise solution that includes the bending effects by assuming that the plies are uniformly distributed through the laminate and membrane/bending coupling effects are ignored. The SMCORE option is an alternative refinement that allows a simple modeling of a frequently encountered sandwich panel design. The stacking sequence of the plies in the face sheet are again ignored and a uniform distribution is assumed across the face sheet, but now the offset due to the known core thickness can be included.

### Theory

MSC.Nastran develops mass and stiffness data from PCOMP input in a two-step process. First, the PCOMP input data are considered together with the material data referenced by MIDi entries to produce PSHELL/MAT2 combinations that will lead to

the required stiffness results. Then these spawned data are used in the actual stiffness and mass calculations. The spawned PSHELL has four MIDis, identifying the MAT2s to be used for membrane, bending, transverse shear, and membrane-bending coupling. The SMEAR and SMCORE options use these MIDis in the following manner:

## SMEAR

The spawned PSHELL has MID1=MID2 with MID3, MID4, 12I/T\*\*3, and the TS/T fields set as blanks. This results in a bending term given as

$$[D] = \frac{T^2[A]}{12} \quad \text{Eq. 5-3}$$

## SMCORE

Computation of the membrane and bending stiffness matrices is performed using the following derivation. Note that membrane-bending coupling is ignored.

### Definitions:

$$\begin{aligned} t_{face} &= T_1 + T_2 + \dots + T_{N-1} \text{ (total thickness of both face sheets)} \\ t_{core} &= T_N \text{ (total core thickness)} \end{aligned} \quad \text{Eq. 5-4}$$

### Membrane Stiffness Matrix:

$[A]$  is computed using method utilized by LAM=BLANK assuming that core stiffness (layer N) is zero.

### Face Sheet Properties:

$$\mu_{xy} = \frac{-[A]_{12}^{-1}}{[A]_{11}^{-1}} \quad \text{Eq. 5-5}$$

$$\mu_{yx} = \frac{-[A]_{12}^{-1}}{[A]_{22}^{-1}} \quad \text{Eq. 5-6}$$

$$E_x = \frac{A_{11}(1.0 - \mu_{xy}\mu_{yx})}{t_{face}} \quad \text{Eq. 5-7}$$

$$E_y = \frac{A_{22}(1.0 - \mu_{xy}\mu_{yx})}{t_{face}} \quad \text{Eq. 5-8}$$

$$G_{xy} = \frac{A_{33}}{t_{face}} \quad \text{Eq. 5-9}$$

Moment of inertia:

$$I_{xx} = I_{yy} = \frac{t_{face}\left(t_{core} + \frac{t_{face}}{2}\right)^2}{4} + \frac{t_{face}^3}{48} \quad \text{Eq. 5-10}$$

Note, this collapses to  $t_{face}^3/12$  if  $t_{core}$  is zero.

**Bending Stiffness Matrix:**

$$D_{11} = \frac{E_x I_{xx}}{(1.0 - \mu_{xy}\mu_{yx})} \quad \text{Eq. 5-11}$$

$$D_{22} = \frac{E_y I_{yy}}{(1.0 - \mu_{xy}\mu_{yx})} \quad \text{Eq. 5-12}$$

$$D_{12} = D_{21} = \frac{D_{11}\mu_{xy}E_y}{E_x} \quad \text{Eq. 5-13}$$

$$D_{33} = G_{xy} I_{xx} \quad \text{Eq. 5-14}$$

$$D_{13} = D_{31} = D_{23} = D_{32} = 0 \quad \text{Eq. 5-15}$$

## Input

The format of the PCOMP entry is unchanged, but the description of the LAM field is expanded and additional Remarks have been added.

Field	Contents
LAM	Laminate Options (Character or Blank, default=Blank). See Remarks 3. and 4.
“Blank”	All plies must be specified and all stiffness terms are developed

Field	Contents
"SYM"	Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1 or TN) should be half the actual thickness.
"MEM"	All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.
"BEND"	All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.
"SMEAR"	All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4, and TS/T and $12I/T^{**3}$ terms are set to zero).
"SMCORE"	All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below, with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.

## Remarks

3. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR, and SMCORE options provide special purpose stiffness calculations. [SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared.] SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.
4. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.

## Output

Standard element response data are recovered for elements that employ the new LAM options with the key provision that the SMEAR and SMCORE options result in element results that correspond to the equivalent homogenous element and not the ply-by-ply results. This is equivalent to using PARAM NOCOMPS -1 and only the homogeneous results are produced regardless of the user input specification of the NOCOMPS parameter.

## Guidelines and Limitations

Special procedures must be employed when using the PCOMP with the SMEAR and SMCORE options in an automated design task in SOL 200. The FSD option in SOL 200 does not support the SMEAR or SMCORE options because the resizing of a ply layer in FSD is driven by a response in that layer. The SMEAR/SMCORE options do not produce layer responses; therefore, FSD resizing is precluded. For the standard mathematical programming of SOL 200, composite stress, strain and force results are not available for the PCOMP with SMEAR or SMCORE. Instead, the homogeneous element results must be invoked. This is done by using the standard STRESS, STRAIN, FORCE response types (as opposed to the CSTRESS, CSTRAIN, CFAILURE response types) and the PTYPE can be set to either PCOMP or PSHELL. The PID for the PSHELL is identical to that of the PCOMP. The rules for the design of properties are not changed by these new options in that the user can invoke a lamina thickness or orientation on the PCOMP entry from a DVPREL1 or DVPREL2. Invoking the thickness of the equivalent PSHELL that is derived from a PCOMP is not allowed and produces a fatal error.

## Examples (PSTIFF1, PSOFT1, PMEM1, PSMEAR, and PSMCORE)

This example demonstrates an application of the new options in the modeling of wing skins and compares them with results obtained when the LAM option is SYM or MEM.

### Input File

The overall model is a simple, built-up cantilever with a span of 10 units, a constant chord of 4 units, and a constant core depth of 1 unit. A tip load totaling 600 units is applied on the grids at the span tip. Webs and spars are metallic and modeled using CQUAD4 elements. The composite wing skins are modeled with ten CQUAD4s on the top surface and ten on the bottom.

Five models are used for comparison:

- PSTIFF1 represents a layup that has a high bending stiffness in the direction of bending in the skins. Each skin, top and bottom, has the following 8 ply layup:  
0/45/-45/90 (symmetric) with a 0.0125 unit ply thickness
- PSOFT1 minimizes bending stiffness with the following 8 ply layup:  
90/45/-45/0 (symmetric) with a 0.0125 unit ply thickness

- PMEM1 uses the MEM option without bending stiffness and has the following 4-ply layup:  
0/45/-45/90 (unsymmetric) with a 0.025 unit ply thickness
- PSMEAR uses the SMEAR option and has a layup identical to the PSOFT model.
- PSMCORE uses the SMCORE option and adds a small (.005) core thickness to the layup of the PSOFT1 and PSMEAR models.

### Results

**Table 5-1** compares selected results from the five modeling options. Strains are in terms of micro strains. The strain results are for an element at the root of the bottom skin. It is seen that the **PMEM** is the softest because it has no bending stiffness and **PSTIFF** is the stiffest while PSMEAR and PSMCORE give intermediate results.

**Table 5-1 Results for PCOMP Comparisons**

Result	PSTIFF	PSOFT	PMEM	PSMEAR	PSMCORE
Tip Deflection	0.6703	0.6719	0.6724	0.6711	.6709
von Mises Strain	4929	4940	4945	4935	4933
von Mises Curvature	4218	4779	N/A	4549	N/A
0 Fiber Strain	6570	6361	6326	N/A	4529
45 Fiber Strain	2321	2319	2244	N/A	N/A
-45 Fiber Strain	2291	2293	2244	N/A	N/A
90 Fiber Strain	-1835	-1892	-1838	N/A	N/A

<b>Note:</b>	With the MEM SMEAR and SMCORE options, it is not necessary to create a symmetric layup. This is why PMEM and PSMEAR have four layers of 0.025 unit thickness plies, whereas, PSTIFF and PSOFT have eight plies of 0.125 unit thickness plies. PSMCORE has an additional layer with thickness .005 representing the core.
--------------	--

## 5.8 Temperature-Dependent Composites

### Introduction

MSC.Nastran 2004 updates the smeared laminate properties with the temperature-dependent material properties of the individual plies, and calculates the thermal strain as an integral rather than a secant. In addition, temperature dependence has been added for shell orthotropic materials, and thermal strains may be input directly. These enhancements apply to nonlinear static analysis.

### Benefits

Prior to MSC.Nastran 2004, the smeared laminate properties for a composite material were calculated only once, at the beginning of the analysis, using the reference temperature with no gradient through the thickness. The thermal strain was calculated using a secant approach with the coefficient of thermal expansion at the initial temperature and the load temperature. For a nonlinear analysis, where the temperature may change appreciably, and especially where thermal buckling may occur, the new smeared laminate properties calculations provide a more realistic simulation.

### Theory

#### Temperature Distribution

The temperature through the shell thickness is assumed to be linear:

$$T(\xi, \eta, \zeta) = T(\xi, \eta) + \zeta \cdot T', \quad -\frac{t}{2} \leq \zeta \leq \frac{t}{2}$$

where  $T(\xi, \eta)$  is the membrane temperature defined by the weighted average of the grid temperatures from the TEMP and/or TEMPD Bulk Data entries, or the element temperature from the TEMPP1 Bulk Data entry; and  $T'$  is the temperature gradient from the TEMPP1 Bulk Data entry.

Currently the smeared laminate properties are calculated using the reference temperature from the PCOMP Bulk Data entry, which has no gradient defined through the thickness. This implies that all the plies are at the same temperature, which is not a valid assumption in certain cases.

The temperature at the midplane of the  $i$ th ply will be calculated as:

$$T_i = \bar{T} + \left( z_0 + \sum_{j=1}^{i-1} t_j + \frac{1}{2}t_i \right) \cdot T'$$

where  $\bar{T}$  is the reference plane temperature;  $z_0$  is the bottom surface offset from the reference plane; and  $t_i$  are the ply thicknesses defined on the PCOMP Bulk Data entry. This temperature will be used to calculate the temperature-dependent material properties of each ply.

## Coefficient of Thermal Expansion

### Secant Approach

The coefficient of thermal expansion (in one dimension) is defined as the normalized change in length for a given change in temperature:

$$\alpha = \frac{1}{L} \frac{\Delta u}{\Delta T}$$

such that the thermal strain is given by:

$$\varepsilon = \alpha \Delta T$$

which is the secant approach. This can be computed in several ways, depending on the case control and material dependence.

For linear analysis using neither TEMP(MAT) nor TEMP(INIT):

$$\varepsilon = \alpha(T_{load} - T_{ref})$$

For linear analysis using TEMP(INIT) with no MATTi:

$$\varepsilon = \alpha(T_{load} - T_{init})$$

For linear analysis using TEMP(MAT) with MATTi:

$$\varepsilon = \alpha_{mat}(T_{load} - T_{ref})$$

For linear analysis using TEMP(INIT) with MATTi:

$$\varepsilon = \alpha_{init}(T_{load} - T_{init})$$



For nonlinear analysis using TEMP(INIT) with MATTi (TEMP(INIT) is required and TEMP(MAT) is not allowed). The expression is more complex, in an attempt to better approximate the temperature-dependent properties:

$$\varepsilon = \alpha_{load}(T_{load} - T_{ref}) - \alpha_{init}(T_{init} - T_{ref})$$

## Integral Approach

This secant approximation is adequate for small changes in temperature; however, for temperature-dependent properties, a derivative approximation is more accurate:

$$\alpha(T) = \frac{1}{L} \frac{\partial u}{\partial T}$$

such that the thermal strain is given by:

$$\varepsilon = \int_{T_{init}}^{T_{load}} \alpha(T) dT$$

which is the integral approach.

Temperature dependence for each material property, such as the coefficient of thermal expansion, may be defined as a linear function or a power series, and may include offsets and scale factors. These are input on the TABLEMi Bulk Data entries, and may be integrated analytically for each interval of the table.

## Input

For the temperature-dependent composites, two new parameters have been added. The COMPMATT parameter controls whether the smeared laminate properties are updated with the temperature-dependent material properties of the individual plies. It has a default value of NO and must be changed to YES to invoke the enhancement. The EPSILONNT parameter controls whether the thermal strain is computed using an integral or a secant method. It has a default value of SECANT and must be changed to INTEGRAL to invoke the enhancement.

A new MATT8 Bulk Data entry has been added for temperature-dependent shell orthotropic material properties, and thermal strains may be input directly by using a negative TABLEMi ID for the coefficients of thermal expansion on the MATTi Bulk Data entry and the referenced TABLEMi Bulk Data entry. All other bulk data entries and case control remain the same.

## Output

The output for the temperature-dependent composites has not changed; rather, the way in which it is calculated has changed.

For the strains, the thermal strains will be calculated using the smeared coefficients of thermal expansion at the current load temperature in addition to using the appropriate secant or integral method. For the stresses and forces, the smeared material properties will be calculated at the proper temperature. For the ply strains and stresses, the thermal strain will be calculated using the coefficients of thermal expansion at the current load temperature and appropriate method for the specific ply, and the ply properties will be calculated at the proper temperature, rather than using the reference temperature on the PCOMP Bulk Data entry.

In addition, if the parameter COMPMATT is set, the composite elements will be included in the nonlinear stress/strain output.

## Guidelines and Limitations

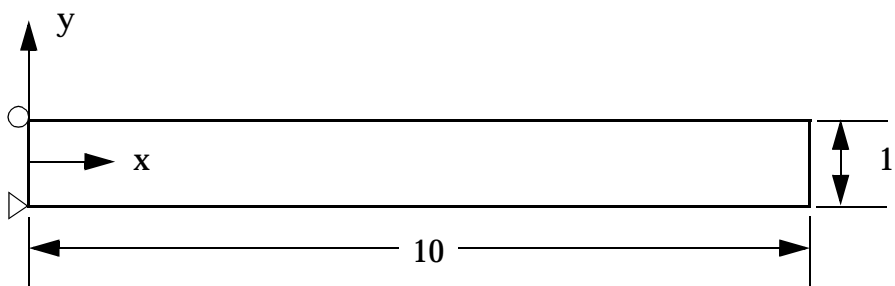
Composite capabilities are currently available with the QUAD4, QUAD8, TRIA3, and TRIA6 h-elements, but nonlinear capabilities are currently available with only the linear elements. Therefore only the QUAD4 and TRIA3 element types will be supported. (Note that although smeared laminate properties may be applied to other two-dimensional elements as well, ply-by-ply post-processing is not performed on those elements.)

The capabilities will be implemented for SOL 106, Nonlinear Statics. They will not be implemented for SOL 101, Linear Statics, nor SOL 129, Nonlinear Transient Response.

## Examples

(TDCRN\_\*.DAT)

In order to illustrate the new enhancements, a very simple problem will be solved for each permutation of the parameters. The problem consists of a simply-supported cantilever beam, ten units long by one unit wide, undergoing a uniform free expansion from  $T_{init} = 100$  to  $T_{load} = 200$  with  $T_{ref} = 50$ , as shown in the following figure:



Two independent versions of the model will comprise each test file: one will reference a PSHELL, and the other will reference a PCOMP with a single ply of the same thickness and orientation as the shell. The material is isotropic, although a MAT8/MATT8 will be used to demonstrate the new MATT8.

The coefficient of thermal expansion is temperature-dependent, as shown in the following table:

	Temperature (T)	CTE ( $\alpha$ )
$T_{ref}$	50	0.0005
$T_{init}$	100	0.001
$T_{load}$	200	0.005

Most of the models will be run in SOL 106, Nonlinear Statics, although one will be run in SOL 101, Linear Statics, for comparison purposes. (Note again that the enhancements are not implemented in SOL 101.)

The six example problems are listed in the following table, along with the thermal strains computed using the equations given above. For the final problem, the thermal strains, marked with an asterisk, are input directly on the TABLEM4 entry as the quadratic integral of the linear property variation of the coefficient of thermal expansion.

	Solution Sequence	Parameters		Thermal Strain ( $\epsilon_t$ )	
		COMPMATT	EPSILON T	PSHELL	PCOMP
TDCRN_L	101	n/a	n/a	0.10	0.05
TDCRN_NS	106	no	secant	0.70	0.05
TDCRN_YS	106	yes	secant	0.70	0.70
TDCRN_NI	106	no	integral	0.30	0.05

	Solution Sequence	Parameters		Thermal Strain ( $\epsilon_t$ )	
		COMPMATT	EPSILON T	PSHELL	PCOMP
TDCRN_YI	106	yes	integral	0.30	0.30
TDCRN_YIE	106	yes	integral	0.30*	0.30*

Results from the six example problems are listed in the following table, along with the axial displacement at the final load step.

	Solution Sequence	Parameters		Axial Displacement ( $u_x$ )	
		COMPMATT	EPSILON T	PSHELL	PCOMP
TDCRN_L	101	n/a	n/a	1.0	0.5
TDCRN_NS	106	no	secant	7.0	0.5
TDCRN_YS	106	yes	secant	7.0	7.0
TDCRN_NI	106	no	integral	3.0	0.5
TDCRN_YI	106	yes	integral	3.0	3.0
TDCRN_YIE	106	yes	integral	3.0*	3.0*

The axial displacements consist of the thermal strains multiplied by the length, as expected. The strains are those given above and the stresses are numerically zero.

## 5.9 Rigid Element Enhancements

### Introduction

Rigid elements in MSC.Nastran 2001 are restricted to small rotations. Furthermore, the differential stiffness for rigid elements is not calculated. In MSC.Nastran 2004, a new method for rigid elements is introduced to allow large rotations and to provide a differential stiffness. The new method for rigid elements is invoked with minor additional input. The existing user input of rigid elements does not change.

### Methods for Rigid Elements

In MSC.Nastran 2004, two methods for rigid elements are available -- the linear method and the Lagrange method.

#### Linear Methods

The linear method has been available since the inception of MSC.Nastran. The linear method uses the linear elimination technique. The rigid elements are not real elements, they are internally represented by a set of MPC equations. By using these MPC equations, the dependent degrees-of-freedom (the m-set) are eliminated from the solution set. The rigid elements linear method has the following limitations:

- Do not compute thermal load.
- Do not have a differential stiffness matrix; therefore, the solutions are incorrect for the buckling analysis or other solution sequences where the differential stiffness matrix is required.
- Use the small rotation theory in the geometrical nonlinear analysis so that the solutions are incorrect in this type of analysis.
- Use the elimination method for solution, resulting in very dense stiffness matrices. These dense matrices cannot take advantages of the sparse matrix algorithm.

The Lagrange method, which is discussed in the next sub-section, does not have the above limitations.

## Lagrange Method

In MSC.Nastran 2004, a new method called the Lagrange method is introduced. With the Lagrange method, the rigid elements become “real” finite elements, similar to, for example, a QUAD4 element. Instead of using MPC equations, the element stiffness matrix is computed for each rigid element. All the limitations for the linear method are removed, i.e., the rigid elements

- Include thermal load effect.
- Include the differential stiffness.
- Support for both the small rotation and the large rotation. Use the large rotation theory in the geometrical nonlinear analysis (PARAM,LGDISP,1).
- Take advantages of the sparse matrix algorithm if the augmented Lagrange multiplier method, defined below, is used.

For each Lagrange rigid element, a number of Lagrange multiplier degrees-of-freedom are created internally by MSC.Nastran. For example, one to six Lagrange multiplier degrees-of-freedom are created for the RBAR and one Lagrange multiplier degree-of-freedom is created for the RROD. For each Lagrange rigid element, the independent degrees-of-freedom, the dependent degrees-of-freedom, and the Lagrange multiplier degrees-of-freedom are left in the solution set (the I-set). Two solution methods are available:

### Augmented Lagrange Multiplier Method

For this method, the solution is obtained with the independent degrees-of-freedom, the dependent degrees-of-freedom, and the Lagrange multipliers degrees-of-freedom left in the solution set. Thus, the sparse characteristic of the stiffness matrix is maintained and sparse matrix algorithms can be used.

### Lagrange Elimination Method

For this method, additional operations of eliminating both the dependent and Lagrange multiplier degrees-of-freedom from the solution set are performed after the global stiffness matrix is assembled. The solution is performed on the independent set (the n-set), creating dense matrices after elimination. Once again, these dense matrices cannot take advantage of the efficiency features of sparse methods.

## Method Selection

For most problems, the augmented Lagrange multiplier method is preferred. The tangent stiffness matrix using this method, however, is not positive definite, and can potentially pose numerical difficulties. The Lagrange elimination method is introduced as a backup for eigenvalue problems.

For linear rigid elements, the dependent degrees-of-freedom are eliminated and placed in the mp-set. For the augmented Lagrange multiplier rigid elements, both the dependent degrees-of-freedom and the internally created Lagrange multipliers are left in the l-set. For the Lagrange elimination method, the dependent degrees-of-freedom are placed in the mr-set. Both the dependent degrees-of-freedom and the Lagrange multipliers are eliminated. The mp-set and the mr-set are subsets of the m-set. Please see “[Degree-of-Freedom Set Definitions](#)” on page 778 of the *MSC.Nastran Quick Reference Guide*.

The Lagrange elements are only available for linear static analysis (SOL 101), normal modes analysis (SOL 103), buckling analysis (SOL 105), and general nonlinear analysis (SOL 400). The Lagrange elimination method is only recommended for eigenvalue analysis, it is not recommended in nonlinear static analysis for the following reasons:

- For nonlinear analysis, the tangent stiffness matrix must be updated at each nonlinear iteration for the Lagrange elimination method. (See “[The Lagrange Elimination Method](#)” on page 251.) This implies that only the full Newton-Raphson method can be used, which conflicts with the general iteration solution scheme employed in MSC.Nastran.
- The denseness of the tangent stiffness matrix after elimination poses an inefficiency problem for the sparse solver.
- For nonlinear analysis, only stiffness matrix decomposition is required and the eigenvalue analysis is not required. For stiffness matrix decomposition, the method for solving a non-positive definite matrix is well established, and hence removes the need for a backup method for the Lagrange multiplier method.

For MSC.Nastran 2004, SOL 400 has additional limitations. Please see “[Limitations in SOL 400](#)” on page 257.

## Theory

In this section, we introduce the theoretical formulation of rigid elements. Rigid elements use large rotation theory, and hence, warrant a discussion of large rotation. In the following sections, the method for representing large rotation for rigid elements

is presented first, followed by the solution methods for the linear elimination method, the Lagrange multiplier method, and the Lagrange multiplier method with elimination. Also, only the solution methods for the nonlinear analysis are discussed.

## Rotation Vector

Mathematically, large rotations may be represented in many ways. The most commonly used methods are Euler angles, Gimbal angles, Rodrigues parameters, quaternion (Euler parameters), and rotation vector methods. MSC.Nastran implemented the Gimbal angles and the rotation vector method into the geometric nonlinear analysis. The Gimbal angles have inherent limitations. They are non-invariant parameters, which means that their values are intimately associated with the coordinate system used. On the other hand, the rotation vector gives a simple geometric meaning to rotations and is an invariant parameter. Thus, the Lagrange elements use the rotation vector for representing large rotations, as described in the next paragraph. Please note that the Lagrange elements do not have the option of using the Gimbal angles.

The most general motions of a rigid body can be represented by a translation of a base point and rotation about the base point. Therefore, the motion of a GRID point rigid body can be decomposed into a translation of the GRID point and a rotation defined by:

$$\Psi = \psi \mathbf{p}$$

where:

$$\psi = \|\Psi\| = (\Psi^T \Psi)^{\frac{1}{2}} \quad \text{Eq. 5-16}$$

$$\mathbf{p} = \frac{\Psi}{\psi} = \begin{Bmatrix} p_x \\ p_y \\ p_z \end{Bmatrix}$$

Thus  $\psi$  is the magnitude or the angle of the rotation and  $\mathbf{p}$  is the direction of the rotation.

It is well known that large rotations cannot be treated as vectors. Consequently, the assumption that rotations are linear, which is one of the basic assumptions of linear analysis, does not provide a unique solution when a series of large rotations is applied



in different sequences. However, the effect of large rotation can be expressed in terms of the rotation matrix  $\mathbf{R}(\Psi)$ . Any vector  $\mathbf{v}$  can be rotated into new position  $\mathbf{v}_r$  given by the equation:

$$\mathbf{v}_r = \mathbf{R}(\Psi)\mathbf{v}$$

$$\mathbf{R}(\Psi) = \begin{bmatrix} \cos \psi + p_x^2(1 - \cos \psi) & -p_z \sin \psi + p_x p_y(1 - \cos \psi) & p_y \sin \psi + p_x p_z(1 - \cos \psi) \\ p_z \sin \psi + p_x p_y(1 - \cos \psi) & \cos \psi + p_y^2(1 - \cos \psi) & -p_x \sin \psi + p_y p_z(1 - \cos \psi) \\ -p_y \sin \psi + p_x p_z(1 - \cos \psi) & p_x \sin \psi + p_y p_z(1 - \cos \psi) & \cos \psi + p_z^2(1 - \cos \psi) \end{bmatrix} \quad \text{Eq. 5-17}$$

The rotational matrix is an orthogonal matrix, having the property:

$$\mathbf{R}\mathbf{R}^T = \mathbf{I} \quad \text{or} \quad \mathbf{R}^{-1} = \mathbf{R}^T$$

A very important matrix associated with the rotation vector  $\Psi$  is the skew-symmetric matrix  $\hat{\Psi}$ , defined by:

$$\hat{\Psi} = \begin{bmatrix} 0 & -\psi_z & \psi_y \\ \psi_z & 0 & -\psi_x \\ -\psi_y & \psi_x & 0 \end{bmatrix} \quad \text{Eq. 5-18}$$

The rotation vector  $\Psi$  and its skew-symmetric matrix have the relationship:

$$\hat{\Psi}\Psi = 0 \quad \text{and} \quad \hat{\Psi}\mathbf{v} = \Psi \times \mathbf{v} \quad \text{for any vector } \mathbf{v} \quad \text{Eq. 5-19}$$

$\Psi$  is the axial vector of the skew-symmetric matrix  $\hat{\Psi}$ . In term of the skew-symmetric matrix and the unit vector  $\mathbf{p}$  of rotation, the rotation matrix can be rewritten as:

$$\mathbf{R}(\Psi) = \mathbf{p}\mathbf{p}^T + \cos \psi(\mathbf{I} - \mathbf{p}\mathbf{p}^T) + \sin \psi \hat{\mathbf{p}} \quad \text{Eq. 5-20}$$

where  $\hat{\mathbf{p}}$  is the skew-symmetric matrix of the unit vector  $\mathbf{p}$ .

## Composite Rotations -- Left Rotation or Right Rotation

A composite rotation is the successive application of two or more rotations. For large rotation, this series of rotations is no longer additive. Furthermore, they are not commutative, i.e., the order of application of the rotation is important. For example, in a geometric nonlinear analysis, an incremental rotation  $\Delta\theta_g$  is computed for a load increment or a nonlinear iteration. The composite rotation of  $\Delta\theta_g$  and the previous rotation  $\Psi_i$  can be expressed by the equation:

$$\mathbf{R}_{i+1}(\Psi_{i+1}) = \Delta\mathbf{R}_l(\Delta\theta_g)\mathbf{R}_i(\Psi_i) \quad \text{Eq. 5-21}$$

Let us examine the coordinate system used for [Eq. 5-21](#). Both  $\Psi_{i+1}$  and  $\Psi_i$  are computed in terms of the global coordinate system at the GRID point. Note that the global coordinate system at any GRID point is fixed in space. Here we must be very careful about the coordinate system used for the incremental rotation  $\Delta\theta_g$ . In order for [Eq. 5-21](#) to be true, we must compute the incremental rotation in terms of the same coordinate system used by  $\Psi_{i+1}$  and  $\Psi_i$ , i.e., the global coordinate system at the GRID point. The corresponding incremental rotation matrix  $\Delta\mathbf{R}_l$  computed is called **the left rotation**.

Another way to compute the incremental rotation matrix would be to rotate the global coordinate system at the GRID point by  $\Psi_i$ , i.e.

$$\mathbf{m}_x = \mathbf{R}_i(\Psi_i)\mathbf{g}_x \quad \text{Eq. 5-22}$$

where  $\mathbf{g}_x$ ,  $x = 1, 2, 3$ , are the unit vectors of the global coordinate system  $\mathbf{g}$  at the GRID point and  $\mathbf{m}_x$  are the unit vectors of the rotated coordinate system  $\mathbf{m}$ . The coordinate system  $\mathbf{m}$  is called the material coordinate system at the GRID point. Please note that the material coordinate system is **not** fixed in the space and it is rotated with the GRID point. Now, let us compute the incremental rotation  $\Delta\theta_m$  in terms of the material coordinate system. Then, the composite rotation of the incremental rotation and the previous rotation can be expressed by:

$$\mathbf{R}_{i+1}(\Psi_{i+1}) = \mathbf{R}_i(\Psi_i)\Delta\mathbf{R}_r(\Delta\theta_m) \quad \text{Eq. 5-23}$$

The rotation matrix  $\Delta\mathbf{R}_r$  so computed is called **the right rotation**. It can be shown that the right rotation and the left rotation are related by the equations:

$$\Delta\theta_m = \mathbf{R}_i^T \Delta\theta_g \quad \text{Eq. 5-24}$$

$$\Delta\mathbf{R}_r = \mathbf{R}_i^T \Delta\mathbf{R}_l \mathbf{R}_i \quad \text{Eq. 5-25}$$

The right rotation has the following advantages:

- The representation of inertia forces is greatly simplified.
- The stiffness properties of each elastic member may be described in a rigorous manner, including the geometric stiffening effects.

To improve the geometric nonlinear analysis, the rigid elements are formulated in terms of the right rotation approach. However, all the existing elements, CQUAR4, CTRIA3, and CBEAM, are formulated in terms of the left rotation approach. In order to make the existing elements compatible with the rigid elements, the existing elements are modified in terms of the right rotation approach.

## The Linear Elimination Method

For the linear elimination method, all rigid elements are reduced using MPC constraint equations given by:

$$\mathbf{u}_m = \mathbf{G}_m \mathbf{u}_n \quad \text{Eq. 5-26}$$

where  $\mathbf{u}_m$  are the dependent degrees-of-freedom,  $\mathbf{u}_n$  are the independent degrees-of-freedom, and  $\mathbf{G}_m$  is the constraint matrix. During the solution process, the dependent degrees-of-freedom are eliminated from the solution set (the I-set). This method is valid only for small rotation because the relationship given by Eq. 5-26 is linear. Also, the thermal loads and the differential stiffness matrix are not computed. For large rotation, the relationships between dependent degrees of freedom and independent degrees of freedom are nonlinear. This elimination method can be modified to solve the nonlinear rigid element problem. However, we use the augmented Lagrange multiplier method instead, which is discussed in the next sub-section.

## The Augmented Lagrange Multiplier Method

The equilibrium equation for a static, nonlinear solution of a constrained, structural model using the Lagrange multiplier method is given by:

$$\begin{aligned} \mathbf{F} + \mathbf{H}^T \lambda &= \mathbf{p} \\ \Phi(\mathbf{u}) &= \mathbf{q} \end{aligned} \quad \text{Eq. 5-27}$$

where  $\mathbf{F}$  is the internal force vector,  $\lambda$  is a vector of Lagrange multipliers,  $\mathbf{p}$  is the external force vector,  $\mathbf{u}$  is the displacement vector,  $\Phi$  is a vector of nonlinear constraints on the displacements generated by the rigid elements,  $\mathbf{q}$  is the external enforced displacements or effect due to thermal loads, and  $\mathbf{H}$  is the displacement gradient given by:

$$\mathbf{H} = \frac{\partial}{\partial \mathbf{u}} (\Phi(\mathbf{u}) - \mathbf{q}) = \frac{\partial \Phi}{\partial \mathbf{u}} \quad \text{Eq. 5-28}$$

The explicit form for  $\Phi$  and  $\mathbf{H}$  are dependent on the individual rigid element type. For example, the forms are different for the RBAR and the RBE3.

The equilibrium for Eq. 5-27 is obtained by minimizing the functional:

$$\Pi = \Pi_k + k(\Phi - \mathbf{q})^T \lambda + \frac{1}{2}(\Phi - \mathbf{q})^T p(\Phi - \mathbf{q}) \quad \text{Eq. 5-29}$$

The first term  $\Pi_k$  of Eq. 5-29 contains the regular strain energy and external work done to the system. For rigid elements, this term is zero. The second term is a contribution due to the Lagrange multipliers, where  $k$  is a scale factor introduced to balance the magnitude of the Lagrange multipliers with other terms in the stiffness matrix. The last term is the added penalty function term and  $p$  is the penalty function, which is added to reduce numerical difficulty during matrix decomposition.

The constrained equilibrium equation can also be solved by the **penalty function method**. For this method, the magnitude of the penalty function must be very large in comparison to the tangential stiffness matrix in order to obtain an approximation solution of the original constrained equation. When the penalty function approaches infinity, the solution becomes the exact solution of the original constrained equation. The shortcoming of this method is that, when the penalty function approaches infinity, the tangential stiffness matrix becomes ill conditioned. On the other hand, in the augmented Lagrangian method as given by Eq. 5-29, the penalty function  $p$  needs not to be large. It only needs to be similar in magnitude to the other terms in the tangential stiffness matrix for enforcing the positive definiteness of the stiffness matrix.

For the nonlinear analysis, the next step is to derive the Newton-Raphson iteration equation from Eq. 5-29. Let  $(\mathbf{u}^*, \lambda^*)$  be an approximation solution to Eq. 5-29. By using the Newton-Raphson method, a corrected solution can be obtained in the form:

$$(\mathbf{u}, \lambda) = (\mathbf{u}^* + \Delta \mathbf{u}, \lambda^* + \Delta \lambda) \quad \text{Eq. 5-30}$$

$$\mathbf{K}_{ee} \begin{Bmatrix} \Delta \mathbf{u} \\ \Delta \lambda \end{Bmatrix} = \begin{Bmatrix} \mathbf{e}_u \\ \mathbf{e}_\lambda \end{Bmatrix} \quad \text{Eq. 5-31}$$

where  $\mathbf{K}_{ee}$  is the tangential stiffness matrix and the right hand side is the error vector. The first and second variations of Eq. 5-29 are

$$\delta \Pi = \delta \mathbf{u}^T \mathbf{H}^T (p(\Phi - \mathbf{q}) + k\lambda) + \delta \lambda^T k(\Phi - \mathbf{q}) \quad \text{Eq. 5-32}$$

$$d(\delta \Pi) = \delta \mathbf{u}^T \mathbf{H}^T (p \mathbf{H} d\mathbf{u} + k d\lambda) + \delta \lambda^T k \mathbf{H} d\mathbf{u} + \delta \mathbf{u}^T \frac{\partial}{\partial \mathbf{u}} (\mathbf{H}^T (p(\Phi - \mathbf{q}) + k\lambda)) d\mathbf{u} \quad \text{Eq. 5-33}$$

From Eq. 5-33, we obtain the corresponding element tangential stiffness matrix as:

$$\mathbf{K}_{ee} = \mathbf{K}_{ee}^e + \mathbf{K}_{ee}^d = \begin{bmatrix} \mathbf{H}^T p \mathbf{H} & k \mathbf{H}^T \\ k \mathbf{H} & 0 \end{bmatrix} + \begin{bmatrix} \frac{\partial}{\partial \mathbf{u}} (\mathbf{H}^T (p(\Phi - \mathbf{q}) + k\lambda)) & 0 \\ 0 & 0 \end{bmatrix} \quad \text{Eq. 5-34}$$

In the above equation, the first term is the element stiffness and the second term is the differential stiffness for the rigid element. Next, we need to compute the error vector for the rigid element in a nonlinear iteration. For the approximate solution  $(\mathbf{u}^*, \lambda^*)$ , the error vector can be obtained from Eq. 5-32 as

$$\begin{Bmatrix} \mathbf{e}_u \\ \mathbf{e}_\lambda \end{Bmatrix} = \begin{Bmatrix} -\mathbf{H}^T(\mathbf{u}^*)(k\lambda^* + p(\Phi(\mathbf{u}^*) - \mathbf{q})) \\ k(\mathbf{q} - \Phi(\mathbf{u}^*)) \end{Bmatrix} \quad \text{Eq. 5-35}$$

With the tangential stiffness matrix and the error vector defined, Eq. 5-30 and Eq. 5-31 are used to obtain a better approximation. This process continues until the solution converges.

## The Lagrange Elimination Method

The stiffness matrix for the Lagrange multiplier method given by Eq. 5-34 is not positive definite. For most problems, this method is appropriate. However, for certain types of problems, this method may create numerical difficulties. Therefore, the Lagrange multiplier method with elimination, called the Lagrange elimination method, is implemented as a backup method for the Lagrange multiplier method. For reasons given in “Bulk Data Entries” on page 208, the Lagrange elimination method is implemented only for the linear static analysis, the normal modes analysis, and the buckling analysis. However, for completeness, the theory given below is still based on the nonlinear analysis.

The Lagrange elimination method is summarized as follows:

- Compute the rigid element tangential stiffness matrix and error vector by using the Lagrange multiplier method as given by Eq. 5-34 and Eq. 5-35 with the penalty function  $p=0$ .
- Assemble the element stiffness and error vector into global stiffness and error vector, which will create a system equation similar to Eq. 5-31.
- Eliminate the Lagrange multipliers and the dependent degrees-of-freedom from the tangential stiffness matrix and the error vector.
- Construct and solve the Newton-Raphson equation in terms of the independent degrees-of-freedom.

The method of elimination is derived as follows. Let  $g$  denote the total displacement degrees of freedom for the structural model,  $m$  the dependent degrees-of-freedom, and  $n$  the independent degrees-of-freedom. Then the displacement gradient defined by Eq. 5-28 can be partitioned into m-set and n-set as

$$\mathbf{H}_{mg} \Delta \mathbf{u}_g = \begin{bmatrix} \mathbf{H}_{mm} & \mathbf{H}_{mn} \end{bmatrix} \begin{Bmatrix} \Delta \mathbf{u}_m \\ \Delta \mathbf{u}_n \end{Bmatrix} \quad \text{Eq. 5-36}$$

and the constraint equation for the dependent degrees-of-freedom can be computed as

$$\begin{aligned} \Delta \mathbf{u}_m &= \mathbf{G}_{mn} \Delta \mathbf{u}_n \\ \mathbf{G}_{mm} &= -\mathbf{H}_{mm}^{-1} \mathbf{H}_{mn} \end{aligned} \quad \text{Eq. 5-37}$$

Now, the global tangential stiffness matrix in g-set, similar to Eq. 5-31, Eq. 5-34, and Eq. 5-35 can be expanded into m-set and n-set as

$$\begin{bmatrix} \bar{\mathbf{K}}_{nn} & \mathbf{K}_{nm} & \mathbf{G}_{mn}^T \\ \mathbf{K}_{nm}^T & \mathbf{K}_{mm} & -\mathbf{I} \\ \mathbf{G}_{mn} & -\mathbf{I} & 0 \end{bmatrix} \begin{Bmatrix} \Delta \mathbf{u}_n \\ \Delta \mathbf{u}_m \\ \Delta \lambda_m \end{Bmatrix} = \begin{Bmatrix} \mathbf{p}_n - \mathbf{F}_n(\mathbf{u}_g^*) - \mathbf{G}_{mn}^T(\mathbf{u}_g^*) \lambda_m^* \\ \mathbf{p}_m - \mathbf{F}_m(\mathbf{u}_g^*) + \lambda_m^* \\ -\mathbf{H}_{mm}^{-1}(\mathbf{q}_m - \Phi(\mathbf{u}_g^*)) \end{Bmatrix} = \begin{Bmatrix} \bar{\mathbf{e}}_n \\ \mathbf{e}_m \\ \bar{\mathbf{q}}_m \end{Bmatrix} \quad \text{Eq. 5-38}$$

The right hand side of above equation is the error vector. Elimination of m-set displacements and the Lagrange multipliers from the above equation yields the tangent stiffness matrix in terms of n-set independent degrees-of-freedom, as:

$$\mathbf{K}_{nn} \Delta \mathbf{u}_n = \mathbf{e}_n \quad \text{Eq. 5-39}$$

$$\mathbf{K}_{nn} = \bar{\mathbf{K}}_{nn} + \mathbf{K}_{nm} \mathbf{G}_{mn} + \mathbf{G}_{mn}^T \mathbf{K}_{nm}^T + \mathbf{G}_{mn}^T \mathbf{K}_{mm} \mathbf{G}_{mn} \quad \text{Eq. 5-40}$$

$$\mathbf{e}_n = \bar{\mathbf{e}}_n + \mathbf{G}_{mn}^T \mathbf{e}_m + (\mathbf{K}_{nm} + \mathbf{G}_{mn}^T \mathbf{K}_{mm}) \bar{\mathbf{q}}_m \quad \text{Eq. 5-41}$$

$$\Delta \mathbf{u}_m = \mathbf{G}_{mn} \Delta \mathbf{u}_n - \bar{\mathbf{q}}_m \quad \text{Eq. 5-42}$$

$$\Delta \lambda_m = \mathbf{K}_{nm}^T \Delta \mathbf{u}_n + \mathbf{K}_{mm} \Delta \mathbf{u}_m - \mathbf{e}_m \quad \text{Eq. 5-43}$$

Eq. 5-39, Eq. 5-40, and Eq. 5-41 can be used in the Newton-Raphson iterations, and Eq. 5-42 and Eq. 5-43 can be used to recover the displacements in the m-set and the Lagrange multipliers. In general, the error vector given by Eq. 5-39 is dependent on the tangential stiffness matrix. In order to compute the correct error vector, the stiffness matrix must be updated at each nonlinear iteration. This means only the full Newton-Raphson method can be used to obtain the solution for the Lagrange elimination method.

## Input

### Case Control Command RIGID

Control Case command RIGID selects the type of rigid element. It has the following format:

$$\text{RIGID} = \left\{ \begin{array}{c} \text{LINEAR} \\ \text{LAGR} \\ \text{LGELIM} \end{array} \right\}$$

LINEAR will select the linear rigid elements, LAGR will select the Lagrange rigid element with the Lagrange multiplier method, and LGELIM will select the Lagrange element with the Lagrange elimination method.

If RIGID command does not exist in the user Case Control file, the linear rigid element will be used for all solution sequences except SOL 400. For SOL 400, the default rigid element type is the Lagrange rigid element with the Lagrange multiplier method.

### Parameters LANGLE, LMFACT, PENFN, and ORIGID

The parameter LANGLE selects the method to represent the larger rotations, having the following values:

LANGLE,1 -- use the Gimbal angle method.

LANGLE,2 -- use the left rotation method.

LANGLE,3 -- use the right rotation method.

See sub-section, “[Composite Rotations -- Left Rotation or Right Rotation](#)” on page 247 for a description of the rotation methods. LANGLE=1 or 2 is the existing method. LANGLE=3 is the new method implemented for the Lagrange rigid elements

and is the only method available for the Lagrange rigid elements. Therefore, if Lagrange rigid elements are present in a structural model, MSC.Nastran will automatically use `LANGLE=3`, and ignore user input on the `LANGLE` parameter.

The parameters `LMFACT` and `PENFN` are the scale factor  $k$  and penalty function  $p$  (see [Eq. 5-29](#)) for the Lagrange rigid element with the Lagrange multiplier method. The purpose of `LMFACT` and `PENFN` is to make the values of the stiffness matrix for the Lagrange rigid elements approximately equal in relative magnitude to those of the other elements in a structural model. Too small a value produces inaccurate results and too large a value poses numerical difficulties. Usually, the same value is assigned to both `LMFACT` and `PENFN`. Under special requirements, the user may assign different values to `LMFACT` and `PENFN`. For example, if `LMFACT`  $\neq$  0.0 and `PENFN` = 0.0, then the solution method becomes the pure Lagrange multiplier method instead of the augmented Lagrange multiplier method. However, the user must exercise caution when assigning different values to `LMFACT` and `PENFN`. The default value is 1.0e+5 for all solution sequences except SOL 400. For the nonlinear solution sequence SOL 400, MSC.Nastran will compute the appropriate default value in nonlinear iterations.

The parameter `ORIGID` is the offset for the identification number of the Lagrange multiplier. MSC.Nastran internally generates the Lagrange multiplier degrees-of-freedom for the Lagrange elements. The identification number for the Lagrange multiplier is `ORIGID+MGID`, where `MGID` is the grid point identification number of the dependent grid point of the rigid element. The default value for `ORIGID` is 101000000 if there are no other internally generated grid points. This value is modifiable using system cell 178.

## Bulk Data Entries

Existing rigid elements -- `RBAR`, `RBE1`, `RBE2`, `RBE3`, `RROD`, and `RTRPLT` -- can be used as either the linear rigid element or the Lagrange element, that is selected by the Case Control command, `RIGID`. In addition, three new Bulk Data entries, `RBAR1`, `RTRPLT1`, and `RJOINT` are discussed in next two sub-sections.

For each Lagrange rigid element, a number of Lagrange multiplier degrees-of-freedom are created internally by MSC.Nastran. The number of the Lagrange multiplier degrees-of-freedom is equal to the dependent degrees-of-freedom defined by the rigid element. The identification numbers of Lagrange multipliers are determined by the parameter `ORIGID`.

All of the rigid elements described above can either be the linear rigid element or the Lagrange rigid element. However, the input rules are not the same for these two types of rigid elements. The following paragraphs discuss the differences.



Except for the RBE3 and RROD, the major difference in the input format between the linear rigid element and the Lagrange rigid element is the selection of the independent degrees-of-freedom. This can be illustrated by the Bulk Data entry for RBAR:

Table 5-2

RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	
------	-----	----	----	-----	-----	-----	-----	-------	--

The independent degrees-of-freedom are selected by CNA and CNB. For the linear rigid element, the independent degrees-of-freedom can be assigned to both CNA and CNB: for example, “CNA=1236, CNB=34”, as long as the total number equals to six and they can jointly represent any general rigid body motion. However, for the Lagrange rigid element, all six independent degrees-of-freedom must be assigned to a single grid point, i.e., “CNA=123456, CNB=blank”, or “CNA=blank, CNB=123456”. The same rule applies to the RBE1, RBE2, and RTRPLT elements.

For the RBE3, the REFC degrees-of-freedom can be any combination of integers from 1 through 6 for the linear rigid element. For the Lagrange rigid element, RFEC must be 123, 456, or 123456.

For the RROD element, the user must select one dependent degree-of-freedom by inputting either CMA or CMB for the linear rigid element. However, for the Lagrange rigid element, the user can leave both fields blank and let MSC.Nastran select the best component as the dependent degree-of-freedom. In fact, this is the recommended method.

For all rigid elements, a new field ALPHA is the thermal coefficient of expansion. For the Lagrange rigid elements, if ALPHA is given and the thermal loads are requested by the Case Control command TEMPERATURE(INITIAL) and TEMPERATURE(LOAD), the thermal load effect will be computed for the rigid elements. The temperature loads are taken as the average temperature given by the independent grid point and the dependent grid point. For example, the temperature load for the BAR element is taken as the average temperature of grid points GA and GB. For the linear rigid element, no temperature effect is computed and the ALPHA field is ignored.

RBAR1 and RTRPLT1

RBAR1 and RTRPLT1 are two new Bulk Data entries that are alternative input formats for RBAR and RTRPLT. The Bulk Entry for RBAR1 is shown below:

Table 5-3

RBAR1	EID	GA	GB	CMB	ALPHA				
-------	-----	----	----	-----	-------	--	--	--	--

GA is the independent grid point and all six degrees-of-freedom are independent. The dependent degrees-of-freedom are selected by CMB of the dependent grid point GB. This is a much simpler input format than RBAR and is the preferred input format for the Lagrange rigid element.

The format for RTRPLT1 is shown below:

Table 5-4

RTRPLT1	EID	GA	GB	GC	CMB	CMC	ALPHA		
---------	-----	----	----	----	-----	-----	-------	--	--

RTRPLT1 defines one independent grid point GA and two dependent grid points GB and GC with dependent degrees-of-freedom CMB and CMC, respectively. Between RTRPLT1 and RTRPLT, RTRPLT1 is the preferred input format for the Lagrange rigid element.

RJOINT

RJOINT is a new rigid element. The format for RJOINT is as follows:

Table 5-5

RJOINT	EID	GA	GB	CB					
--------	-----	----	----	----	--	--	--	--	--

GA is the independent grid point and all six degrees-of-freedom are independent. GB is the dependent grid point. The length between points GA and GB must be zero. Because its length is zero, the thermal load effect is not applicable.

If CB=123456 or is left blank, then the grid point GB is constrained to move with the grid point GA, and two the grid points move as a single point. If any degree-of-freedom is released on CB, then RJOINT becomes a mechanical joint. A mechanical joint is a mechanical system that has two bodies jointed at a point. The two bodies can rotate relatively about one, two, or three axes of a local coordinate system at that point. For example:

- Hinge - A hinge is a mechanical joint that rotates freely about one axis about the local coordinate system. It can be simulated by RJOINT with one rotational degree of freedom released, i.e., CB = 12356, 12346, or 12345.
- Universal joint - A universal joint is a mechanical joint that rotates freely in two axes. It can be simulated by RJOINT with two rotational degrees-of-freedom released, i.e., CB = 1234, 1235, or 1236.
- Spherical joint - A spherical joint is a mechanical joint that rotates freely about all three axes. It can be simulated by a zero length RJOINT with all rotational degrees of freedom released, i.e. CB = 123.

- **Prismatic joint** - A prismatic joint is a mechanical system with two blocks that are constrained to have the same rotations, but translate relative with each other along a local axis. It can be simulated by RJOINT with one translational degree-of-freedom released, i.e., CB = 23456, 13456, 12456.
- **Cylindrical joint** - A cylindrical joint is a mechanical system that allows two grid points to have relative translation along a moving axis and, at the same, have relative rotation about the same axis. It can be simulated by RJOINT with one translational degree of freedom and one rotational degree of freedom released, i.e. CB = 2356, 1346, 1245.

Guidelines for the RJOINT element:

- The theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB. If different global coordinate systems are used, the degrees-of-freedom to be released are determined by the coordinate system at GB.
- For linear or nonlinear static analysis, the joints must be constrained by other elements in the structural model. Otherwise, a singular stiffness matrix may be produced, leading to failure during decomposition.

## Limitations in SOL 400

As described in sub-section, “**Composite Rotations -- Left Rotation or Right Rotation**” on page 247, the Lagrange rigid elements are formulated in terms of the right rotation method and the existing elements, CQUAD4, CTRIA3, and CBEAM, are formulated in the left rotation method. In order to make these existing elements compatible with the Lagrange rigid elements, the existing elements must be modified and formulated with the right rotation method. This work is under development. Therefore, the pre-release of SOL 400 has the following limitation:

- If Lagrange rigid elements are used in a model that contains CQUAD4, CTRIA3, or CBEAM elements, then the problem will converge only for 2-D motion. The out-of-plane motion may need to be constrained in this case. Full 3-D motion is supported with Lagrange rigid elements if the model does not contain the above elements.

## Examples

In this section, four simple examples are included to illustrate the usage of Lagrange rigid elements. These four examples are:

- Linear static analysis with thermal loads

- Nonlinear geometric analysis with thermal loads
- Buckling analysis
- Normal modes analysis with differential stiffness

### Example 1: Linear Static Analysis with Thermal Loads (nlrgd11a.dat)

To illustrate the effect of thermal loads on the Lagrange rigid elements in a static analysis, a simple beam is modeled by a RBAR1 element with a length of 1.0. At left end of the beam, a very soft rotational spring is connected to the normal rotation and the remaining five degrees-of freedom are fixed. At the right end, all six degrees-of-freedom are free with a vertical force of 100.0 is acting on it. This beam is subjected to a thermal load, which will double its length to 2.0. This problem is solved by the Lagrange multiplier method (RIGID=LAGR) with the linear static analysis (SOL 101).

The input file for this problem is shown below.

```
ID RIGID,NLRGD10A
SOL 101
CEND
TITLE = LINEAR STATIC ANALYSIS - THERMAL LOAD ON RBAR
SUBTIT= LAGRANGE MULTIPLIER METHOD
RIGID = LAGR
TEMP(INIT) = 10
SUBCASE 1
  TEMP(LOAD) = 20
  LOAD = 100
  SPC = 10
  DISP = ALL
  SPCF = ALL
  FORC = ALL
BEGIN BULK
FORCE,100,2,0,500.0,0.0,1.0,0.0
CELAS2,101,1000.0,1,6
GRID,1,,1.0,0.0,0.0
GRID,2,,2.0,0.0,0.0
RBAR1,3,1,2,123456,2.0-2
TEMP,10,1,0.0
TEMP,10,2,0.0
TEMP,20,1,50.0
TEMP,20,2,50.0
SPC1,10,12345,1
ENDDATA
```

Results for nlrgd11a.dat:

Description	Value
Vertical displacement	0.5
Horizontal displacement	1.0

Description	Value
Angle of rotation	28.6°
Initial beam length	1.0
Final beam length	2.06

These results show that horizontal thermal expansion is correctly computed as 1.0. However, due to the soft rotational spring on the left end of beam, this problem is a large displacement problem. Therefore, the vertical and horizontal displacements, angle of rotation, and the final beam length are incorrect.

**Example 2: Geometric Nonlinear Analysis with Thermal Loads (nlrgd11b.dat)**

This problem illustrates the use of Lagrange rigid elements in geometric nonlinear analysis. The model and loading for this problem is identical to nlrgd11a, except that the geometric nonlinear analysis (SOL 400 and LGDISP,1) is used. Note that the RIGID Case Control command is not needed for SOL 400 because the default for SOL 400 is RIGID=LAGR.

The input file for this problem is shown below.

```
ID RIGID,NLRGD11B
SOL 400
CEND
TITLE = GEOMETRIC NONLINEAR ANALYSIS - THERMAL LOADS ON RBAR
SUBTIT= LAGRANGE MULTIPLIER METHOD
TEMP(INIT) = 10
NLPARM = 10
SUBCASE 1
    TEMP(LOAD) = 20
    LOAD = 100
    SPC = 10
    DISP = ALL
    SPCF = ALL
    FORC = ALL
BEGIN BULK
NLPARM,10,10
PARAM,LGDISP,+1
.
.
.
ENDDATA
```

Results for nlrgd11b.dat:

Description	Value
Vertical displacement	1.344
Horizontal displacement	0.481
Angle of rotation	42.22°
Initial beam length	1.0
Final beam length	2.0

Now the final beam length is 2.0, which is expected, and all above results are correct when large displacement theory is used.

### Example 3: Buckling Analysis (nlrgd10d.dat)

To illustrate the use of Lagrange rigid elements in a buckling analysis, two simple columns are modeled by RBAR elements and are subjected to compressive loads of 100.0 at the top of the column. At bottom of the columns, they are connected to a single grid point and are constrained by very soft rotational spring in both horizontal directions. Also, at the bottom, the three translations and axial rotation are fixed. The buckling analysis (SOL 105) and the Lagrange elimination method (RIGID=LGELIM) are used to obtain the solution.

The input file for this problem is shown below.

```
ID RIGID,NLRGD10D
SOL 105
CEND
TITLE = BUCKLING ANALYSIS - RBAR
SUBTI = LAGRANGE ELIMINATION METHOD
DISP = ALL
SPC = 10
RIGID = LGELIM
SUBCASE 1
$ STATIC PRELOAD CASE
  LOAD = 100
SUBCASE 2
$ BUCKLING CASE
  METHOD = 10
BEGIN BULK
EIGRL,10,,10
FORCE,100,4,0,-100.0,1.0,0.0,0.0
FORCE,100,41,0,-100.0,1.0,0.0,0.0
CELAS2,101,100.0,3,5
CELAS2,102,120.0,3,6
GRID, 3,,2.0,0.0,0.0
GRID, 4,,4.0,0.0,0.0
GRID, 41,,4.0,0.0,0.0
```

```

RBAR, 4,3,4
RBAR, 3,3,41,123456, , ,123456
SPC1,10,1234,3
ENDDATA

```

The buckling load factors calculated are 0.25 and 0.30. For SOL 105, currently only the Lagrange elimination method (RIGID=LGELIM) is supported (see error 1-11842201).

### Example 4: Normal Modes Analysis with Differential Stiffness (nlrgd14c.dat)

This problem illustrates the effect of the differential stiffness matrix with Lagrange rigid elements for normal modes analysis. The model used in this problem is similar to that of nlrgd10a, except that concentrated masses are added to the top of the columns and tensile forces of 100.0 units are applied. The normal modes analysis (SOL 103) using the Lagrange multiplier method (RIGID=LAGR) is used to obtain the solution.

Two problems are investigated--one without and one with the effects of the differential stiffness matrix included.

The input file for the problem including the differential stiffness effect is shown below. For the problem without the differential stiffness, just remove subcase 1.

```

ID RIGID,NLRGD14C
SOL 103
CEND
TITLE = NORMAL MODES ANALYSIS WITH DIFFERENTIAL STIFFNESS - RBAR
SUBTI = LAGRANGE MULTIPLIER METHOD
DISP = ALL
SPC = 10
RIGID = LAGR
SUBCASE 1
$ STATIC PRELOAD CASE
  LOAD = 100
SUBCASE 2
$ NORMAL MODES CASE
  STATSUB = 1
  METHOD = 10
BEGIN BULK
EIGRL,10,,,10
CONM2,201,4,,10.
CONM2,202,41,,10.
FORCE,100,4,0,+100.0,1.0,0.0,0.0
FORCE,100,41,0,+100.0,1.0,0.0,0.0
CELAS2,101,100.0, 3,5
CELAS2,102,120.0, 3,6
GRID, 3,,2.0,0.0,0.0
GRID, 4,,4.0,0.0,0.0
GRID, 41,,4.0,0.0,0.0

```

```
RBAR, 4,3,4
RBAR, 3,3,41,123456, , ,123456
SPC1,10,1234,3
ENDDATA
```

Results for nlrgd11b.dat:

Description	Value
Eigenvalues without preload	1.25, 1.50
Eigenvalues with preload	6.25, 6.50

From the above results, clearly, the differential stiffness can affect the results substantially.



## 5.10 QUADR and TRIAR Elements

### Introduction

Traditional Mindlin shell elements, such as QUAD4, have 5 degrees-of-freedom per grid point--three translations and two bending rotations. The stiffness for the rotational degree-of-freedom normal to the element (the drilling degree-of-freedom) is zero. This creates modeling difficulties that may eventually lead to poor solutions. Many methods--such as AUTOSPC and K6ROT--have been devised to remedy this deficiency. None of them, however, supply physical stiffness in the drilling degree-of-freedom to resist the drilling moments.

Shell elements (the old QUADR/TRIAR) with drilling degrees-of-freedom were introduced in MSC.Nastran Version 66. However, these elements have not been widely used in the production environment because they lack capabilities such as the differential stiffness, coupling in membrane and bending, and support of composite materials, etc. Also, the old QUADR/TRIAR formulations did not yield accurate solutions for curved shell models.

The new QUADR/TRIAR formulation removes the limitations of the old QUADR/TRIAR mentioned above.

Various examples are run comparing results using the QUADR, QUAD4, and QUAD8. Overall, the new QUADR provides the most accurate results and is the recommended element.

### Benefits

The results for the new QUADR are compared with the QUAD4 and the old QUADR.

As compared to the QUAD4, the new QUADR has the following benefits:

- It has six degrees-of-freedom at each grid point as opposed to the QUAD4, which has five degrees-of-freedom. This removes the necessity of using such methods as the AUTOSPC and K6ROT--which sometimes cast doubts on results--to account for the normal rotational DOF in the QUAD4.
- When using AUTOSPC or K6ROT for the QUAD4, the stiffness at the drilling degree-of-freedom is not an actual physical stiffness. Therefore, drilling loads cannot be correctly transferred between elements, which leads to modeling difficulties. For QUADR, the stiffness at the drilling degree-of-freedom is an actual physical stiffness and, therefore, the drilling loads can be transferred correctly between elements. Example 3 demonstrates the QUADR's ability to correctly resist the drilling moments.

- As demonstrated by Examples 2 and 6, the membrane behavior of the new QUADR is far superior to that of the QUAD4 and approaches that of the QUAD8.

Many capabilities available to the QUAD4 are not included in the old QUADR. For the new QUADR, all the capabilities in QUAD4 have been incorporated into the QUADR. In comparison with the old QUADR, the current QUADR has the following benefits:

- An improved theoretical formulation to make the current QUADR a more robust element (see the **Theory** Section.)
- Coupled with the shell normal, the current QUADR is well suited for modeling shell problems (see the Section on **Shell Models and Shell Normal**.)
- The rotational mass is implemented for the drilling degree-of-freedom (see the Section **Mass Properties**.)
- Consistent surface loads and edge loads are implemented (see the Section **Consistent Surface and Edge Loads**.)
- The differential stiffness matrix is implemented for the new QUADR so that buckling and other analyses requiring the differential stiffness can be performed.
- Coupling between bending and membrane is implemented.
- Composite material is implemented for layered stress/strain output.
- Offset is allowed for the new QUADR.
- The new QUADR is implemented into the solution sequence SOL 200, optimization.
- New QUADR is supported in heat transfer analysis.
- A System cell 370 is available for conversions between QUAD4/TRIA3 and QUADR/TRIAR.
- Nonlinear material and geometric analysis will be implemented in a future version.

## Theory

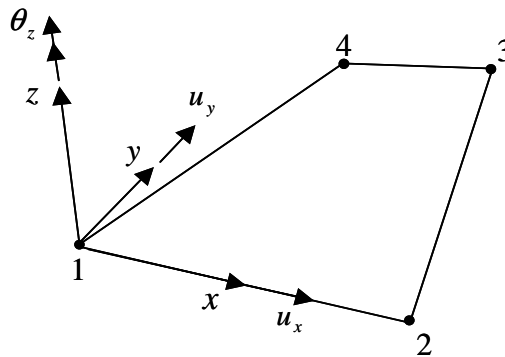
Allmann (reference 14.) and Cook (reference 16.) were the first to propose a successful membrane element with drilling dof. On the other hand, Hughes and Brezzi (reference 18.) presented a variational formulation, first suggested by Reissner (reference 13.). This variational principle uses the drilling degrees of freedom as an independent rotation field, i.e., it is based on the separate kinematic variables of

displacements and rotations. Ibrahimbegavic, Taylor and Wilson (reference 19.) combined the Allmann type of interpolation and the variational principle to construct a quadrilateral shell element. In MSC.Nastran, membrane elements with drilling degrees-of-freedom were first introduced in 1988, which is described in (reference 17.). The shells with drilling degrees-of-freedom in the current version of MSC.Nastran are based on the work in references 17. and 19. A summary of the theory for the membrane portion of the shell element is given below.

The Allman type of interpolation fields assumes the normal displacements to be quadratic and the tangential displacements to be linear. For example, along the edge from grid 1 to 2 (see Figure 5-4), the displacement in the middle of the edge is

$$\mathbf{u}_m = \frac{1}{2}(\mathbf{u}_1 + \mathbf{u}_2) + \frac{1}{8}l_1(\theta_2 - \theta_1)\mathbf{n}_1 \quad \text{Eq. 5-44}$$

where all bold letters indicate vectors with  $x$  and  $y$ -components.  $\mathbf{u}_1$  and  $\mathbf{u}_2$  are the displacements at grids 1 and 2, respectively;  $l_1$  is the length of the edge;  $\theta_1$  and  $\theta_2$  are the drilling rotations at grid 1 and 2, respectively;  $\mathbf{n}_1$  is the vector normal to the edge in the  $x$ - $y$  plane; and  $\mathbf{u}_m$  is the interpolated displacement vector in the middle of the edge.



**Figure 5-4 In-Plane displacements and drilling rotation**

Hughes and Brezzi presented two variational formulations: one of the mixed type and the other is the displacement type, which is the one adopted here. The displacement type variational formulation states that the function to be minimized for elements with drilling degrees-of-freedom is

$$\Pi = \frac{1}{2} \int_A \boldsymbol{\varepsilon}^T \mathbf{G} \boldsymbol{\varepsilon} t dA + \frac{1}{2} \gamma' G \int_A (\Omega - \theta)^2 t dA \quad \text{Eq. 5-45}$$

$$\Omega = \frac{1}{2} \left( \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right) \quad \text{Eq. 5-46}$$

where:

$\gamma$  = the non-dimensional constant, for most problems  $\gamma = 1.0$

$G$  = the shear modulus,  $A$  and  $t$  are the area and thickness

$\Omega$  = the rotational field given by [Eq. 5-46](#)

$\theta$  = the interpolated drilling field

$\varepsilon$  = the strain vector

$\mathbf{G}$  = the stress-strain relationship defined by  $\sigma = \mathbf{G}\varepsilon$

The first term is the normal strain energy and the second term is the contribution due to the drilling degrees-of-freedom.

In the variational formulation given by [Eq. 5-45](#), the drilling field and the displacement field are interpolated independently. For the drilling field, the interpolation function used is the linear QUAD4 shape function. For the displacement field, the Allman type interpolation functions given by [Eq. 5-44](#) are used. They are given by the following equations:

$$\theta(\xi, \eta) = \sum_{i=1}^4 N_i(\xi, \eta) \theta_i \quad \text{Eq. 5-47}$$

$$\mathbf{u}(\xi, \eta) = \sum_{i=1}^4 N_i(\xi, \eta) \mathbf{u}_i + \sum_{i=1}^4 N_{i+4}(\xi, \eta) \frac{l_i}{8} (\theta_{i+1} - \theta_i) \mathbf{n}_i c_r + N_0(\xi, \eta) \mathbf{u}_0 \quad \text{Eq. 5-48}$$

where:

$$i = 1, 2, 3, 4$$

$$\theta_5 = \theta_1$$

The  $N_i$ s in above equations are shape functions for QUAD4 and QUAD8 elements. They are defined by:

$$\begin{aligned} N_i &= \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta) & \text{for } i = 1, 2, 3, 4 \\ N_i &= \frac{1}{2}(1 - \xi^2)(1 + \eta_i \eta) & \text{for } i = 5, 7 \\ N_i &= \frac{1}{2}(1 + \xi_i \xi)(1 - \eta^2) & \text{for } i = 6, 8 \end{aligned} \quad \text{Eq. 5-49}$$

The  $N_0$  in [Eq. 5-48](#) is a bubble function or an auxiliary strain function (reference [17.](#)). The  $\mathbf{u}_0$  is the displacement corresponding to the bubble function or the auxiliary function. The  $\mathbf{n}_i$  is the outward normal vector to the element boundary and  $l_i$  is its length.

With the shape functions defined by [Eq. 5-47](#) and [Eq. 5-48](#), and the variational principle defined by [Eq. 5-45](#), the standard finite element theory can be used to obtain the element forces and stiffness.

With the above formulation, the singularity in the 6<sup>th</sup> degree-of-freedom of an element such as QUAD4 is replaced with a true stiffness. In addition, the drilling rotation becomes a true load carrying degree-of-freedom. We will show that the membrane behavior is significantly improved. In fact, the accuracy of QUADR approaches that of the QUAD8.

## Shell Models and Shell Normal

The shell element described in the previous **Theory** Section is for a flat element. In using flat shell elements for modeling a curved shell, local normals of adjacent shell elements have different directions. In MSC.Nastran, an algorithm has been developed to generate a unique normal at a grid point. The unique normal is the average normal at each grid point so that the angles between the average normal and all the local normals of the adjacent shell elements are minimized. This unique normal is called the **Shell Normal** (references [20.](#) and [21.](#)), which is requested by the PARAM, SNORM, or the Bulk Data entry SNORM.

The old QUADR/TRIAR (reference [17.](#)) formulation, which requires the shell normal in order to give a good solution for many shell problems, is not as robust as the new formulation. The examples cited in references [20.](#) and [21.](#) are based on the old formulation.

The new QUADR/TRIAR, formulation is more robust, is less sensitive to the shell normal, and is well suited for solving shell problems. The following list illustrates the effect of the shell normal to the solution of shell problems. Some of the problems are the MacNeal/Harder test problems (reference 15.)

- Twist beam (MacNeal/Harder test, see example below) – no effect, excellent solution.
- Scordelis-Lo roof (MacNeal/Harder test, see example below) – no effect, excellent solution.
- Spherical shell (MacNeal/Harder test, see example below) – the solution is slightly stiffer with the shell normal than without the shell normal. The solution for the coarse model with shell normal converges rapidly to that of the fine mesh without shell normal.
- Cylindrical shell with internal pressure (see example below) and without the shell normal, solution converges with the fine mesh. With the shell normal, the solution is excellent even with a coarse mesh.
- Pinched cylinder (not presented) – almost no effect, excellent solution. The pinched cylinder is a cylinder with two rigid end diaphragms. This cylinder is subject to a pair of concentrated forces that pinch the cylinder.
- Raasch hook (see example below) – no effect, excellent solution.
- We can conclude from the results that the shell normal is beneficial for membrane-dominant shell problems, and unfavorable for bending-dominant shell problems. To provide good solutions for a wide class of shell problems, the default value for PARAM, SNORM is set to 20.0. Also, from the above results, we can conclude that, with or without the shell normal, the new QUADR/TRIAR is well suited to modeling shell problems.

SNORM should be turned off if there are offsets in the QUADR element.

## Mass Properties

In MSC.Nastran, two mass formulations are available--the lumped mass and the coupled mass. Lumped mass is the default formulation. Coupled mass formulation is requested by the parameter PARAM, COUPMASS, 1.

For the QUADR/TRIAR, the mass properties are as follows:

- Lumped mass – only the translational masses are computed. No rotational mass is computed for the drilling degrees-of-freedom.

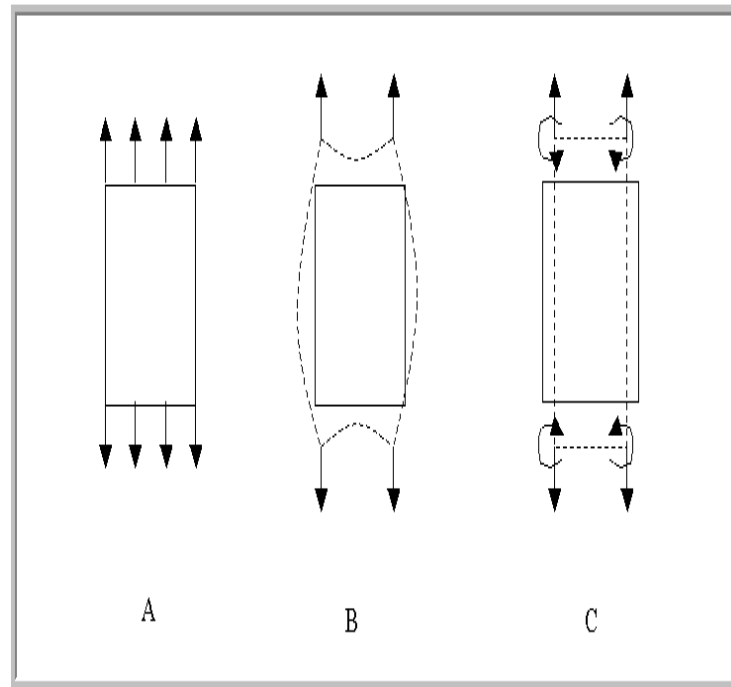
- Coupled mass – in addition to the translational masses, the rotational masses are computed for the drilling degrees-of-freedom.

## Consistent Surface and Edge Loads

PLOAD4, the most commonly used surface load, may have both normal and tangential components relative to the surface of an element. For the QUADR/TRIAR element, the consistent loads due to the surface load contain both concentrated forces at the connected grid points and the drilling moments at the drilling degrees-of-freedom. For the new QUADR/TRIAR, the consistent surface loads have been implemented.

For QUADR/TRIAR, because of the presence of drilling degrees-of-freedom, it is important to compute the consistent grid point forces for any distributed load along the edge of the element. This point is best illustrated by the following example. A single QUADR element is loaded with uniform tensile edge load along the opposite edge as shown in [Figure 5-5A](#). If we lump the uniform load into grid point forces, as shown in [Figure 5-5B](#), then we will get the non-uniform displacement solution, instead of the correct uniform displacement solution. This error can be corrected if we compute the consistent grid point forces for the distributed edge loads. The consistent grid point forces consist of the lumped grid forces and a pair of drilling moments, as shown in [Figure 5-5C](#).

For QUADR/TRIAR only, the consistent edge loads have been implemented. The user can request the consistent edge loads using the PLOAD4 Bulk Data entry.



**Figure 5-5 Consistent Edge Loads**

## User Interfaces

### Parameters SNORM and Bulk Data Entry SNORM

The parameter SNORM is used for requesting unique shell normals for all grid points connected to the shell elements, such as QUADR, in a model. The default shell normal value is PARAM, SNORM, 20.0. For relationships between a shell model and the shell normal, please see the discussions in the **Shell Normal and Shell Models Section**.

The Bulk Data entry, SNORM requests a local shell normal at a grid point. It will override the shell normals requested by PARAM, SNORM, and can also be used to turn off the local default shell normal.

### Bulk Data Entries CQUADR, CTRIAR, and PSHELL

The QUADR/TRIAR elements are requested by the existing CQUADR, CTRIAR, and PSHELL Bulk Data entries. For the new QUADR/TRIAR, there is no modification to the PSHELL entry. For the CQUADR and CTRIAR entries, two new fields are added:

- ZOFFS – this entry is used to input the offsets from the surface of the grid points to the element plane.



- TFLAG – this flag allows the user to input the relative values for a variable grid point thickness, instead of absolute values. See Section 5.5 for further details.

Bulk Data Entry PLOAD4

The PLOAD4 entry is used to define a pressure load for a 3-D or 2-D element. For the QUADR/TRIAR only, this entry is extended to accommodate consistent edge loads. See the **Consistent Surface and Edge Loads** Section for further details. Two new fields have been added to PLOAD4 for this purpose:

- SORL – selects whether the load is a surface or an edge load.
- SORL=SURF selects the surface load and SORL=LINE selects the consistent edge load. SORL=SURF is the default.
- LDIR – defines the direction of the consistent edge load (SORL=LINE). It has the value of X, Y, Z, TANG, or NORM. See “PLOAD4” on page 1623 of the *MSC.Nastran Quick Reference Guide* for definitions. The default is NORM.

For example, the uniform extensive edge loads shown in Figure 5-5 can be specified by the following PLOAD4 entries:

1	2	3	4	5	6	7	8	9	10
PLOAD4	10	1	20.0						
					LINE				

The edge loads have a magnitude of 20.0 and the direction is NORM; i.e., it is in the mean plane of the element, normal to the edge, and pointing outward from the element.

NASTRAN System (370) – QRMETH

The new default QUADR/TRIAR formulation is recommended. However, a new system cell 370 (QRMETH) is available for reverting back to the old formulation.

A companion version of the QUAD4/TRIA3 has also been developed along with the new QUADR/TRIAR. We will call this version the new formulation for QUAD4/TRIA3, and the existing QUAD4 and TRIA3 the old formulation. The performances of the old and the new formulations are similar.

The default for the QUADR/TRIAR uses the new formulation. The default for the QUAD4/TRIA3 uses the old formulation.

In addition to the selection of the new or old formulation for QUADR/TRIAR, QRMETH provides the following options:

- 0 – selects the new QUADR/TRIAR formulation.
- 1 – selects the old QUADR/TRIAR formulation.
- 2 – converts QUADR/TRIAR into QUAD4/TRIA3 using the new QUAD4/TRIA3 formulation.
- 3 – converts QUADR/TRIAR into QUAD4/TRIA3 using the old QUAD4/TRIA3 formulation.
- 4 – selects the new QUAD4/TRIA3 formulation.
- 5 – converts QUAD4/TRIA3 into QUADR/TRIAR using the new QUADR/TRIAR formulation.

The default is for QRMETH is 0.

## Converting a QUAD4 model to QUADR

The following examples will illustrate that the QUADR is superior to the QUAD4. The user may want to convert a QUAD4 model to QUADR by using system cell QRMETH. However, the following should be noted:

- Using QRMETH=5 will convert all QUAD4/TRIA3 in the model to QUADR/TRIAR.
- At the boundaries of the model, the drilling degrees-of-freedom must be constrained if the user wants fixed boundaries. For example, in the example **Static Analysis of Straight Beam with Coarse Mesh**, the drilling degrees-of-freedom must be constrained at the left end.
- At the internal grid points of a model, the user must remove any SPC or the PS field on the GRID Bulk Data entries which constrain the drilling degrees-of-freedom because the QUADR/TRIAR supply stiffness for these degrees-of-freedom. On the other hand, if the user uses K6ROT or AUTOSPC to manage the drilling degrees-of-freedom, these will be converted automatically with QRMETHD=5. No user action is required.

## Examples

### Example 1 - Static Analysis of Straight Beam with Coarse Mesh

For membrane behavior, the performance of the QUADR element is substantially better than that of the QUAD4 element. This can best be illustrated by the static analysis of a straight beam with coarse mesh, shown in [Figure 5-6](#). The load is an in plane shear load acting at the tip of the beam. This test is taken from the MacNeal-Harder tests (reference [15](#).) and is designed to exhibit in plane behavior of shell elements for different element shapes. The results for this test are shown in [Table 5-6](#). The displacements shown are at the tip of the beam, normalized to the theoretical value.

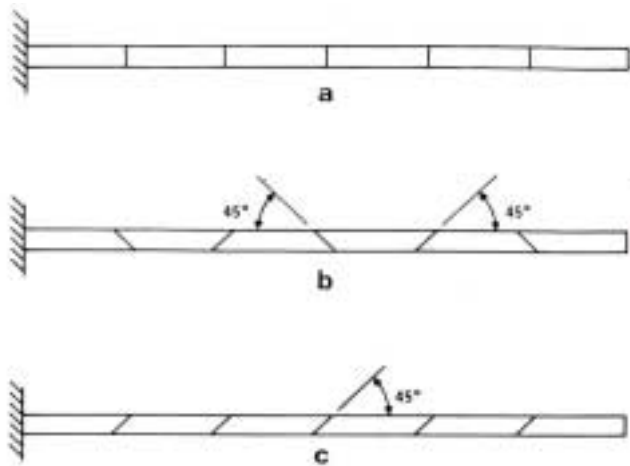


Table 5-6 Normalized displacements for in-plane shear load at tip

Element	Rectangle	Trapezoid	Parallelogram
QUAD4	0.9929	0.0515	0.6232
QUADR	0.9926	0.9613	0.9491

We see that the QUADR provides excellent results for both regular and irregular element shapes, while QUAD4 fails for the irregular shapes. We may argue that, in real life, we would not use an element mesh as coarse as the one in this test. A fine element mesh, will be discussed in next sub-example.

### Example 2 - Normal Modes Analysis of Straight Beam with Fine Mesh

For this example, a refined mesh of Example 1 is used in a normal mode analysis. [Table 5-7](#) summarizes the results of the first mode using various element shapes.

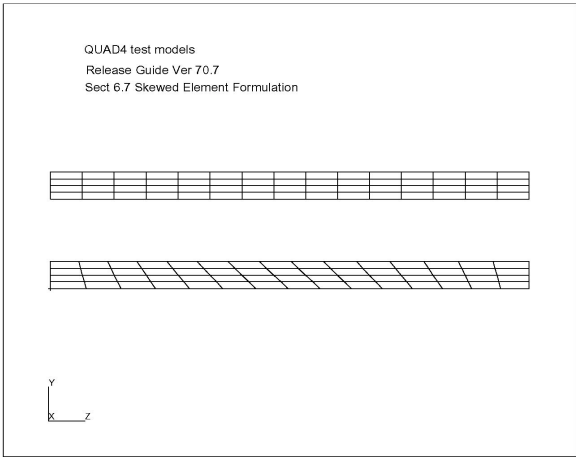


Figure 5-6 Straight beam with fine mesh.

Table 5-7 In-Plane Bending Frequency of Straight Beam with Fine Mesh

Element	Rectangle (hz)	Trapezoid (hz)	Parallelogram (hz)
QUAD4	9.39	15.89	9.41
QUADR	9.39	9.42	9.39

Once again, the QUAD4 element exhibits locking effects for trapezoidal shapes. Even for finer meshes, the results of the QUADR elements are still noticeable better than the results of the QUAD4 element in the case of distorted element shapes.

Example 3 - Static Analysis of Straight Beam with Drilling Moments

This example shows that the QUADR element provides the true physical stiffness for the drilling degrees-of-freedom and hence, is able to resist the actual drilling loads. A straight beam is modeled by six QUADR elements and is subject to a pair of end drilling moments as shown in Table 5-7.

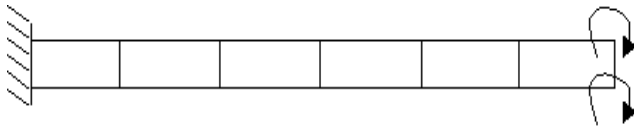


Figure 5-7 Straight Beam Subject to Tip Drilling Moments

The tip displacements of the straight beam for a static analysis are

- Theory = -5.40e-2
- Finite element solution = -5.46e-2

The above results show that the QUADR correctly predicts the response due to the applied drilling moments.

### Example 4 - Static Analysis of Cylindrical Shell with Internal Pressure

The purpose of this example is to illustrate the benefits of the shell normal in a shell structure modeled by QUADR elements. The shell normal is discussed in the **Shell Models and Shell Normal** Section. A cylinder of height 1000.0, radius 1000.0, and thickness 10.0, is subject to internal pressure of 1.0, as shown in **Figure 5-8**. The cylinder is free at both edges. Due to symmetry, only half of the height and a quarter of the cylinder is modeled. The radial displacement at the free edge is given in **Table 5-8**.

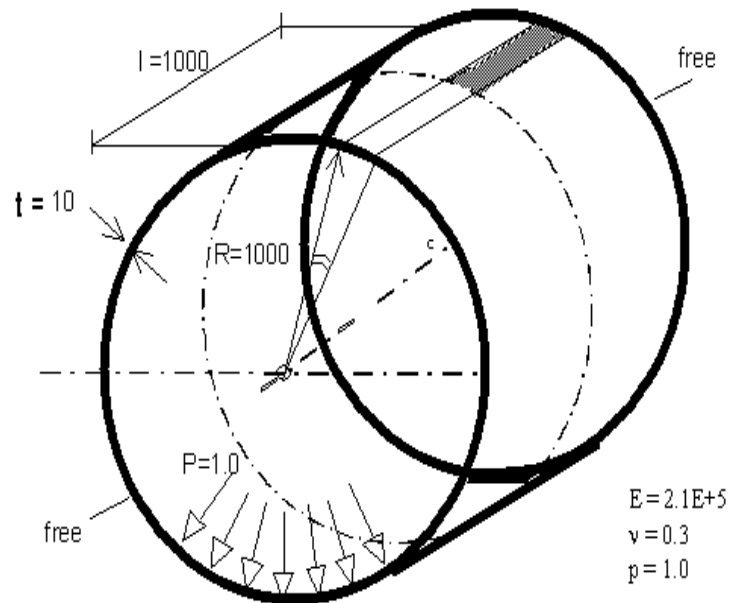


Figure 5-8 Cylindrical Shell with Internal Pressure

Table 5-8 Radial Displacement at Free Edge for Pressurize Cylinder

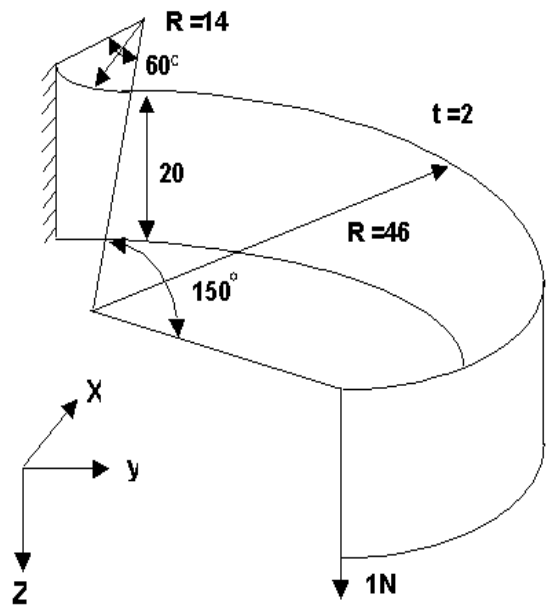
Mesh	Theory	With SNORM	No SNORM
4x4	0.4762	0.4670	0.6733
8x8	0.4762	0.4739	0.5251
16x16	0.4762	0.4756	0.4882

From [Table 5-8](#), the results with the shell normal are substantially better than those without the shell normal.

**Example 5 - Raasch Hook**

The Raasch hook is a curved strip clamped at one end and loaded at the other end with an in-plane shear load as shown in [Figure 5-9](#). A static analysis is performed on the Raasch hook for cases with and without the shell normal. The tip displacements in the direction of the shear force are summarized in [Table 5-9](#) using the new QUADR with various mesh sizes.

From [Table 5-9](#), we see excellent results for the new QUADR, with or without the shell normal. Using the old QUADR, the solution converges to the incorrect results if shell normals are not used as shown in references [20](#). and [21](#).



**Figure 5-9 Raasch Hook**

**Table 5-9 Tip Displacement for Raasch Hook**

Mesh	Theory	With SNORM	No SNORM
1x9	5.012	4.600	4.601
3x17	5.012	4.890	4.888
5x34	5.012	4.916	4.915
10x68	5.012	4.970	4.971
20x136	5.012	5.011	5.012

### Example 6 - MacNeal and Harder Tests

MacNeal and Harder (reference 15.) suggested a series of test problems for evaluating the accuracy of a finite element. For comparison, we have run these test problems for QUADR, QUAD4 and QUAD8. The results are summarized in Table 5-10 and a score is given to each test problem based on how well it compares to the theoretical solution. The scoring criteria are shown:

- A – error is less than and equal to 2%.
- B – error is between 2% and 10%.
- C – error is between 10% and 20%.
- D – error is between 20% and 50%.
- F – error is greater than 50%.

From Table 5-10, we see that QUADR obtained the best scores among the three elements. It does not have a score below B. Between QUADR and QUAD4, the results of QUADR are better than those of the QUAD4. Between QUADR and QUAD8, the accuracy of the QUADR approaches that of QUAD8 for membrane problems, and is better than that of QUAD8 for bending problems.

For QUADR, the tests are run with the default value of the shell normal equal to 20.0. Except for the spherical shell (14), the shell normal has no effect on the tests. For the spherical shell, it will score an A if shell normal is turned off (PARAM, SNORM, 0.0).

Table 5-10 MacNeal and Harder Tests for QUADR, QUAD4, and QUAD8

MacNeal/Harder Test Description	Element Loading		Element Shapes	QUADR	QUAD4	QUAD8
	In-Plane	Out-of-Plane				
1. Patch Test	X		Irregular	A	A	C
2. Patch Test		X	Irregular	A	A	D
3. Straight Beam, Extension	X		All	A	A	A
4. Straight Beam, Bending	X		Regular	A	A	A
5. Straight Beam, Bending	X		Irregular	B	F	B
6. Straight Beam, Bending		X	Regular	A	A	A

**Table 5-10 MacNeal and Harder Tests for QUADR, QUAD4, and QUAD8**

MacNeal/Harder Test Description	Element Loading		Element Shapes	QUADR	QUAD4	QUAD8
	In-Plane	Out-of-Plane				
7. Straight Beam, Bending		X	Irregular	A	A	A
8. Straight Beam, Twist	X	X	All	B	B	B
9. Curved Beam	X		Regular	A	C	A
10. Curved Beam		X	Regular	B	B	B
11. Twisted Beam	X	X	Regular	A	A	A
12. Rectangular Plate (N=4)		X	Regular	A	B	B
13. Scordelis-Lo Roof (N=4)	X	X	Regular	B	B	A
14. Spherical Shell (N=8)	X	X	Regular	B	A	C
15. Thick Walled Cylinder	X		Regular	B	B	B



## 5.11 CWELD Element Enhancements

### Introduction

The CWELD element was introduced in MSC.Nastran 2001. The CWELD can connect congruent as well as non congruent meshes. In this release, we have extended the connectivity options to handle spot welds which connect more than one element per sheet and to handle connections other than surface patches.

### Benefits

In MSC.Nastran 2001 the CWELD was limited to connect one element per shell sheet in patch-to-patch type connections using the ELEMID and GRIDID format, (see [Figure 5-10](#)). The connectivity could be unsymmetric so that results became unsymmetric for a model with symmetric mesh, loads, and boundary conditions. Two new patch-to-patch type connections are introduced, format ELPAT and PARTPAT, which can connect more than one element per sheet, see [Figure 5-11](#). The new formats preserve symmetry. In addition, the two element connectivity could miss elements if the mesh is so fine that the spot weld area spans over more than one element, see [Figure 5-12](#). The situation of meshes with element sizes smaller than spot weld sizes may occur in automotive models for crash analysis. The new formats can connect a maximum of 3x3 elements per shell sheet.

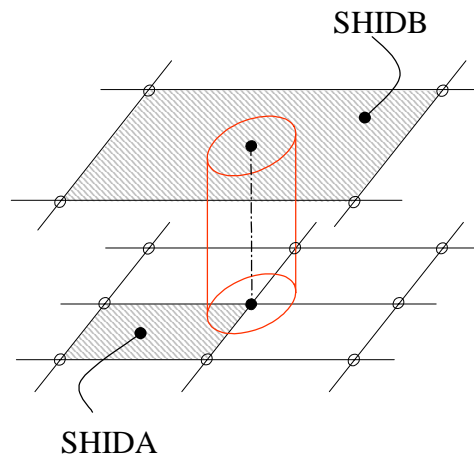
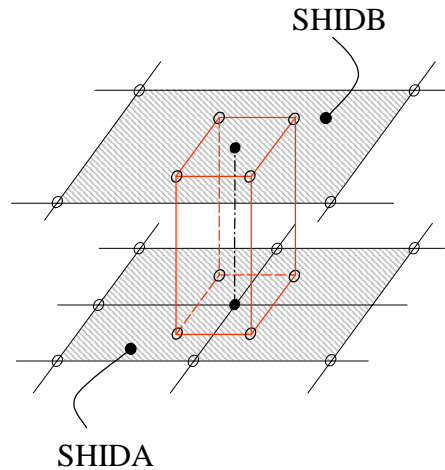
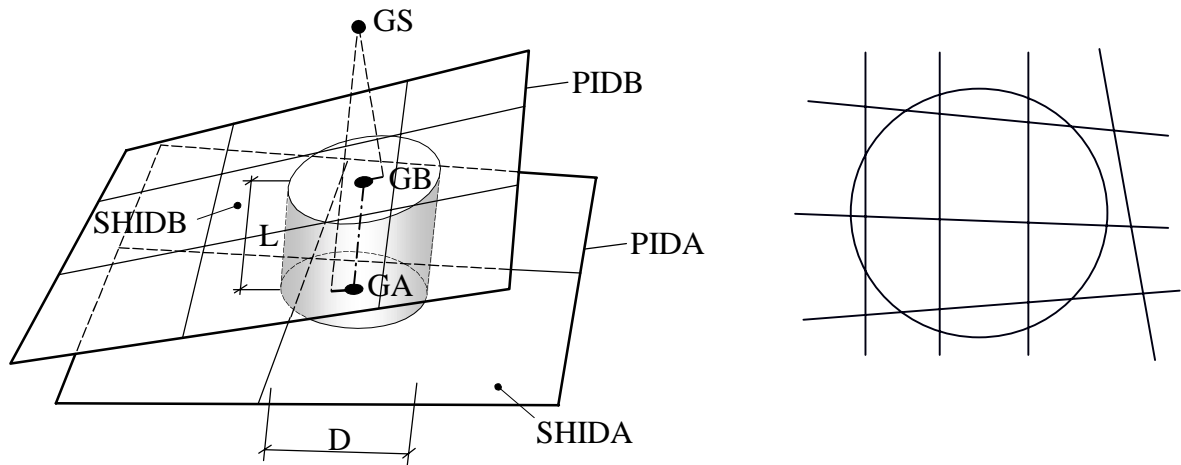


Figure 5-10 Connected elements for format ELEMID



**Figure 5-11 Connected elements for format ELPAT**



**Figure 5-12 Spot weld covering a patch of elements.**

In addition, the CWELD element in MSC.Nastran 2001 was restricted to handling connectivities of shell elements or surface patches made up of grid point sequences. In MSC.Nastran 2004, the point-to-patch type connectivity has been extended so that any element type, load or boundary condition can be connected to the CWELD. The element can now be used to model bearings of large masses, connectivities to springs, gaps, etc., see [Figure 5-13](#).

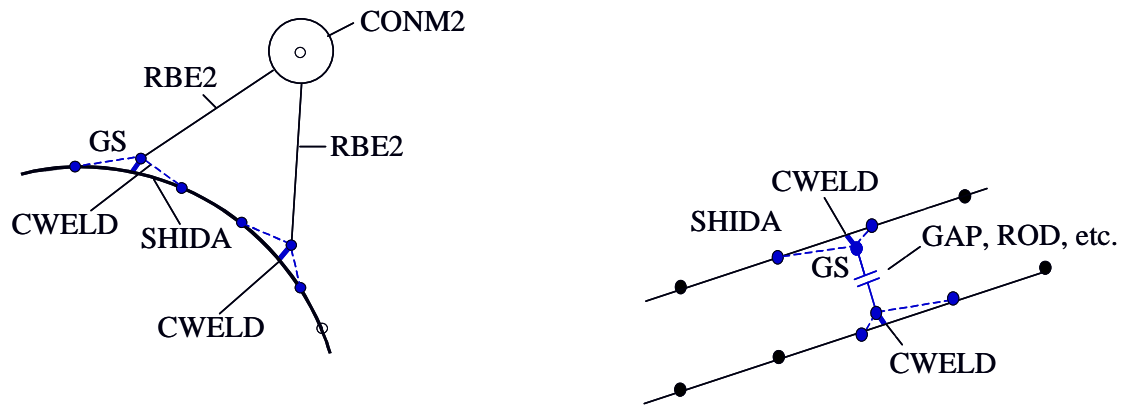


Figure 5-13 Modeling options using the point-to-patch type connectivity.

## Theory

### Finding CWELD End Points GA and GB

If the CWELD end points GA and GB are not defined, then they are created from the locator point GS as follows. For the patch-to-patch type connectivity, the user supplied grid point GS is projected on to shell A and B according to [Figure 5-14](#). The end points GA and GB are created by a normal projection of GS to shell A and B if GS lies between shell A and B. A two step projection is used if GS lies outside of shell A and B. First, the auxiliary points GA' and GB' are created by a normal projection of GS to shell A and B. From the midpoint GC' of section GA',GB' a second normal projection is done that produces the final end points GA and GB. All the points described in this paragraph do not have degrees of freedom attached.

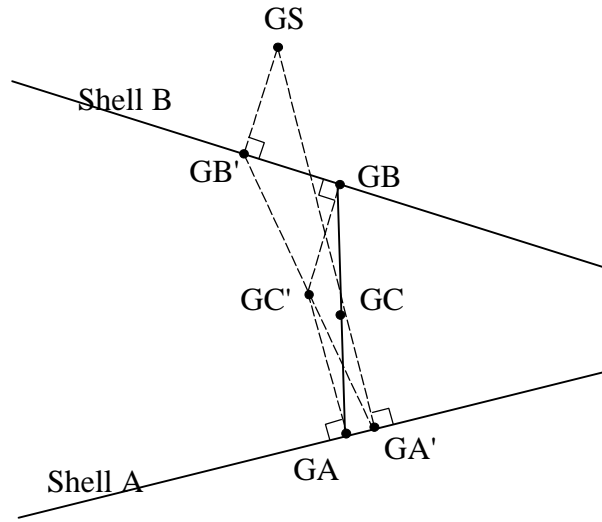


Figure 5-14 Projection of GS to determine end points GA and GB.

### Element Coordinate System

The element x-axis points from GA to GB.

$$e_1 = \frac{x_B - x_A}{\|x_B - x_A\|} \quad \text{element x-axis}$$

In case of zero length, the normal of shell A is taken. All vector components are in basic unless noted otherwise.

Find the smallest component  $j$  of  $e_1$

$$e_1^j = \min_{i=1,2,3} \left\{ e_1^i \right\}.$$

In case of two equal components we take the one with the smaller  $i$ . The corresponding basic vector

$$b_j, \text{ e.g., for } j=3, b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

provides a good directional choice for  $e_2$ . In addition, the vector  $e_2$  must be orthogonal to  $e_1$ .

$$\tilde{e}_2 = b_j - \frac{e_1^T b_j}{e_1^T e_1} e_1 \quad e_2 = \frac{\tilde{e}_2}{\|\tilde{e}_2\|} \text{ element y-axis}$$

and  $e_3$  is just the cross product

$$e_3 = e_1 \times e_2 \quad \text{element z-axis}$$

The final transformation matrix is

$$T_{be} = \begin{bmatrix} | & | & | \\ e_1 & e_2 & e_3 \\ | & | & | \end{bmatrix}$$



**Figure 5-15 Element Coordinate System.**

## Element Stiffness

The elastic part of the CWELD is a short beam from GA to GB with 2x6 degrees of freedom. The element is a shear flexible Timoshenko type beam. The Young's and shear modulus is taken from the material defined on the PWELD property entry. The cross sectional properties are calculated from the user supplied spot weld diameter D defined on the PWELD property entry,

$E$  = Young's modulus

$G$  = shear modulus =  $\frac{E}{2(1 + \nu)}$

$A$  = area =  $\pi D^2 / 4$

$$I_{zz} = I_{yy} \text{ moment of inertia} = \frac{\pi D^4}{64}$$

$$J = \frac{\pi D^4}{32}$$

The length  $L$  of the beam element is from end points GA to GB. If the user defines the type as SPOT on the PWELD property entry, then the effective element length  $L_e$  is always

$$L_e = \frac{1}{2}(t_A + t_B)$$

regardless of the true length  $L$ . The user supplied values for the Young's and shear modulus  $E$  and  $G$  are scaled by the ratio of true length to effective length

$$\tilde{E} = E \frac{L}{L_e} \quad \tilde{G} = G \frac{L}{L_e}$$

With the scaling, the spot weld stiffness is approximately constant for all elements. Extremely stiff elements from short lengths  $L$  and extremely soft elements from long lengths  $L$  are avoided. For zero length  $L$ , the beam element degenerates to 6 uncoupled springs. If the user does not define SPOT as the type, the true length is used if it is inside the range of

$$L_{DMIN} \leq \frac{L}{D} \leq L_{DMAX}$$

where  $L_{DMIN}$  and  $L_{DMAX}$  are user supplied parameters on the PWELD entry. By default  $L_{DMIN} = 0.2$  and  $L_{DMAX} = 5.0$ . If the user does not want the true length adjusted at all,  $L_{DMIN} = 0$  and  $L_{DMAX} = 1.e+12$  may be specified.

### Constraints Connecting GA and GB to Shell Grids

The end points GA and GB are connected to the shell grids of shell A and B, respectively, using constraint equations. The 3 translations at grid GA are connected to the 3 translations of the shell grid points using the interpolation functions of the corresponding shell surface. The 3 rotations at grid GA are connected to the 3 translations of the shell grid points using Kirchhoff conditions.

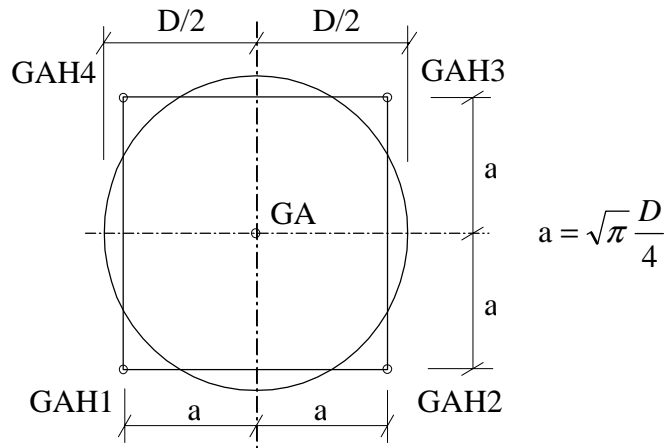
$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_A = \sum N_I(\xi_A, \eta_A) \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_I$$

$$\begin{aligned}\theta_x^A &= \frac{\partial w}{\partial y} = \sum N_{I,y} \cdot w_I \\ \theta_y^A &= -\frac{\partial w}{\partial x} = -\sum N_{I,x} \cdot w_I \\ \theta_z^A &= \frac{1}{2} \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) = \frac{1}{2} (\sum N_{I,x} \cdot v_I - \sum N_{I,y} u_I)\end{aligned}$$

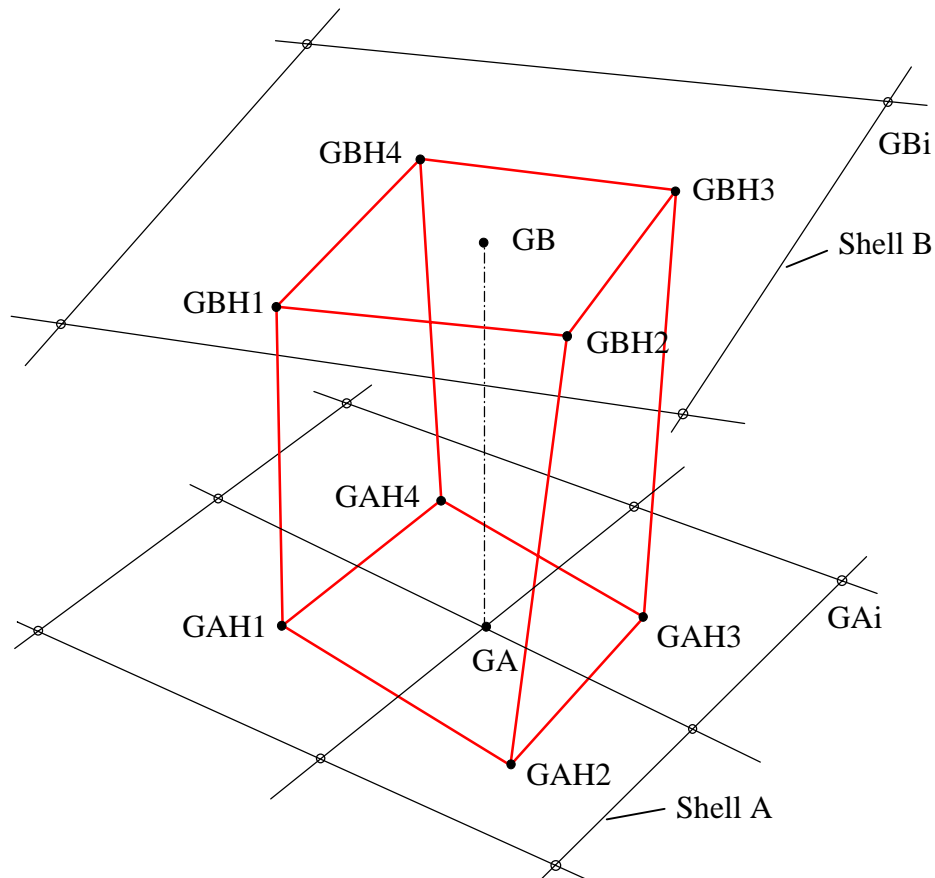
For the formats GRIDID and ELEMID on the CWELD entry, the grid points I are grid points of shell elements or grid points of solid element faces. For the new formats ELPAT and PARTPAT, the grid points I are auxiliary points GAHI and GBHI, I=1,4, constructed according to [Figure 5-16](#) and [Figure 5-17](#). The auxiliary points are connected to shell element grids with a second set of constraints

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_I = \sum_K G_{IK} \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_K$$

where  $G_{IK}$  is a coefficient matrix derived from RBE3 type constraints. The grid points K are shell grid points.

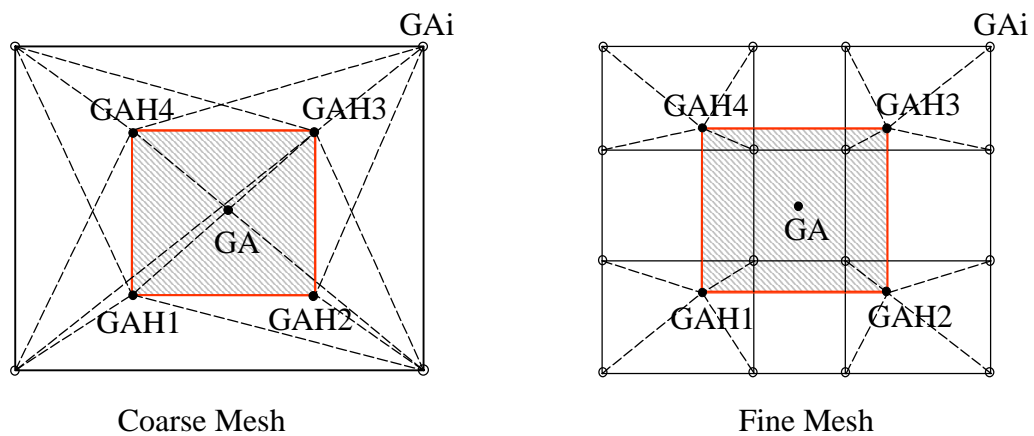


**Figure 5-16** Equivalent square cross sectional area for a spot weld of diameter D



**Figure 5-17 Auxiliary points GAH1 and GBH1 for formats ELPAT and PARTPAT**

With the new formats ELPAT and PARTPAT, the element can connect from one to 3x3 elements on shell A and B, respectively, see [Figure 5-18](#).



**Figure 5-18 Connectivity for formats ELPAT and PARTPAT**



With the constraint equations above, the degrees of freedom of grid points GA, GB and the auxiliary points GAHI, GBHI are condensed out on the element level. The final stiffness matrix of the CWELD consists of translational degrees of freedom of grid points on shell A and B, respectively. There are no extra degrees of freedom generated except for the case where the user requests explicit constraints by setting MSET=ON on the PWELD entry. The flag MSET=ON works only for the formats ELEMID and GRIDID.

Inputs

There are two new formats added to the three existing formats. The first new format, ELPAT, has the same input parameters as the format ELEMID, except for the new character string ELPAT.

CWELD, ELID, PWID, GS, , , ELPAT, , , +  
+, SHIDA, SHIDB

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

For the second new format, PARTPAT, the user defines shell A and B by property id PSIDA and PSIDB.

CWELD, ELID, PWID, GS, , , PARTPAT, , , +  
+, PSIDA, PSIDB

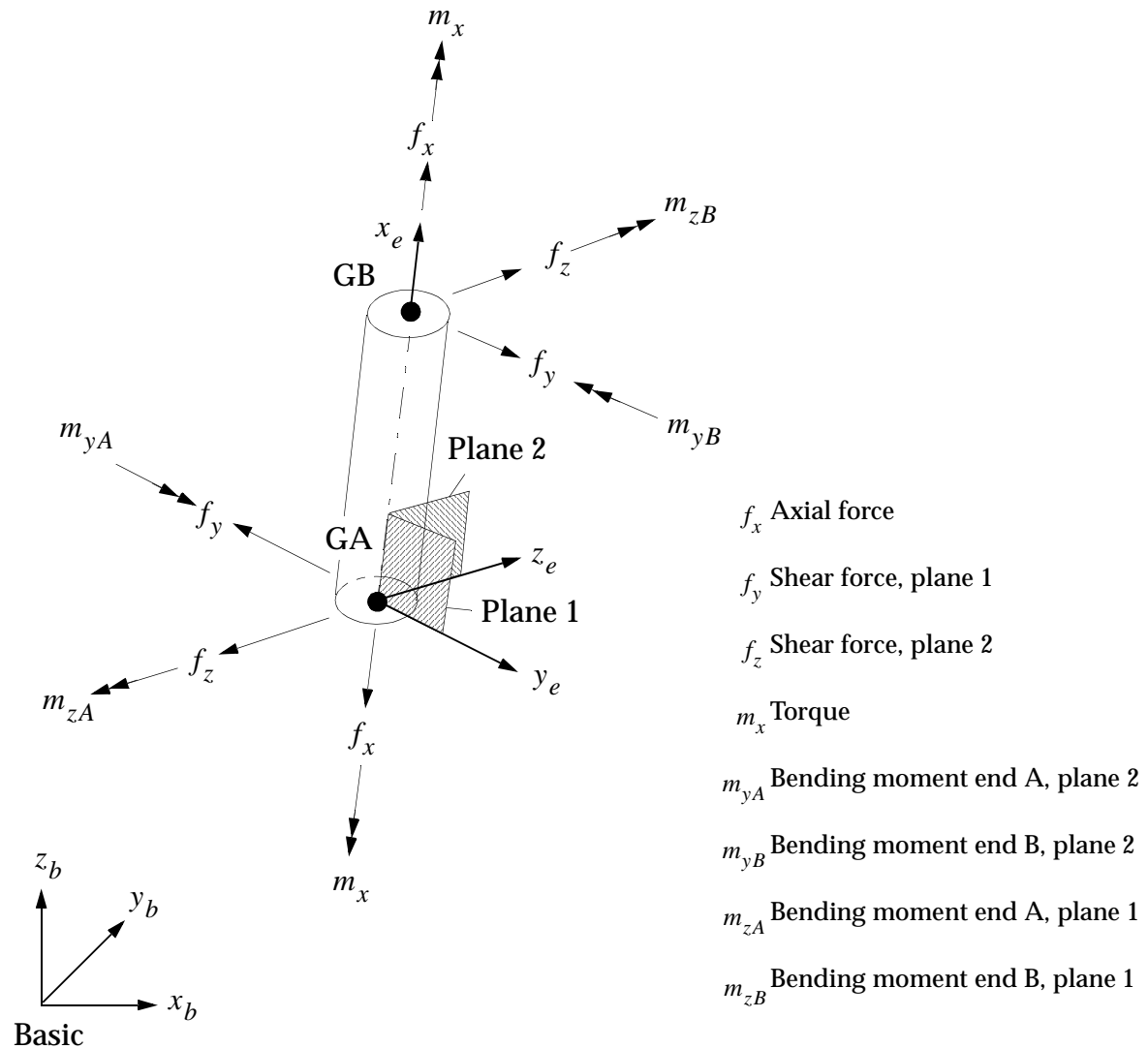
With PARTPAT, all shell elements are connected which fall into the cross sectional area of the spot weld.

Outputs

The force output remains the same as in MSC.Nastran 2001. For internal processing, the following element types are created depending on the user specified formats.

Element Name	Element Number	User Specification
CWELD	200	ALIGN ELEMID, GRIDID with MSET=ON
CWELDC	117	ELEMID, GRIDID with MSET=OFF
CWELDP	118	ELPAT, PARTPAT

The output record of the CWELD forces is identical to the CBAR output record. The output is in the element coordinate system, see the explanations above how the element coordinate system is constructed. The forces are output at the end points GA and GB, see [Figure 5-19](#).



**Figure 5-19 CWELD Element Forces.**

## Guidelines and Limitations

Finding the connectivity is not straight forward for a complex topology. The user has an option to do a check out run first. In addition, the user may choose to let the program move GS in case a projection can not be found. Furthermore, the user may choose to reduce the diameter so that the spot weld can be placed near edges and

corners. All parameters for the connectivity search are lumped on a new SWLDPRM Bulk Data entry. It is recommended to use the defaults first. If there are problems, it is recommended that the check out option be used next and request diagnostic prints.

The CWELD element is not designed to provide accurate stresses at the spot weld location. The stresses and spot weld forces are mesh dependent. The mesh dependency is more pronounced when the mesh size difference between shell A and B is large, for example, when the element connects a coarse mesh on shell A with a fine mesh on shell B.

A couple of limitations exist which will be removed in future versions.

- The CWELD is not available for nonlinear material and it stays linear in geometric nonlinear analysis.
- The element does not take preloads or initial stresses.
- Temperature strains are not available in the element.

The MSET option on the PWELD entry is only available for the formats ELEMID and GRIDID. The MSET default was ON in MSC.Nastran 2001, that means external m-set constraints were generated by default. In MSC.Nastran 2004, the default is MSET=OFF, that means the constraints are eliminated on the element level and there are no external m-set constraints generated. For the new formats ELPAT and PARTPAT, all constraints are eliminated on the element level, there are no external constraints generated.

## Example

A symmetric hat profile with symmetric mesh, loads, and boundary conditions is used to demonstrate the difference between the old ELEMID and the new ELPAT formats, see [Figure 5-20](#). For the old ELEMID format, it may happen that the connectivity is defined in an unsymmetric manner, if the spot weld locator point GS is on a corner of an element, see [Figure 5-21](#). Then the format ELEMID produces unsymmetric results. With the new ELPAT format, even with a user defined unsymmetric connectivity, the internal connectivity is symmetric and the results are symmetric, see the following print out.

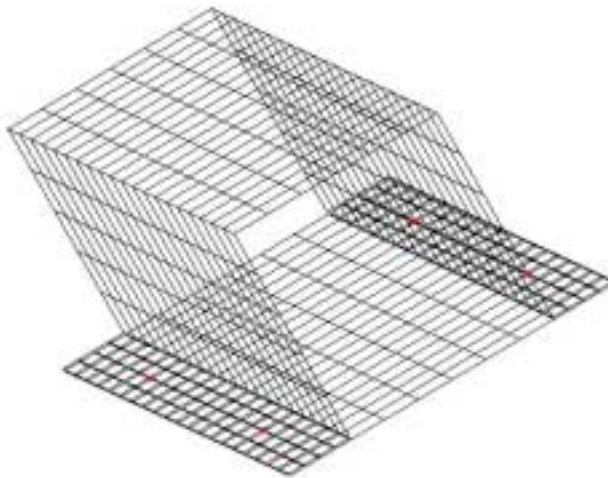


Figure 5-20 Hat Profile

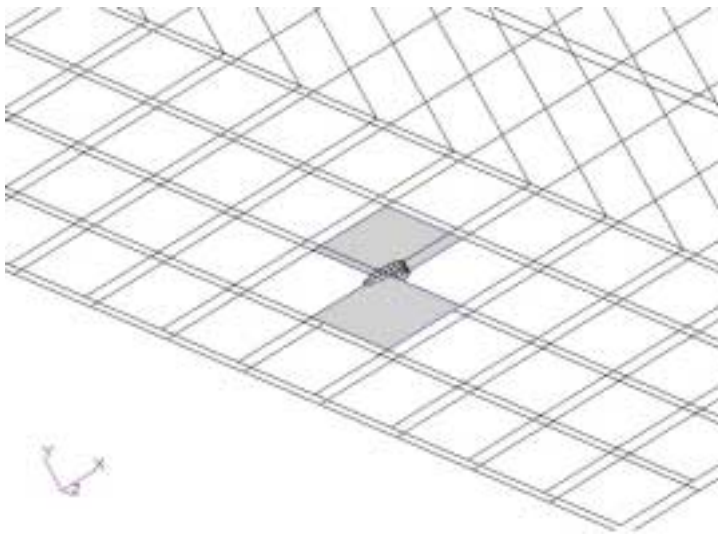


Figure 5-21 Unsymmetric Connectivity for Format ELEMID

The following is the echo of the input for the formats ELEMID and ELPAT:

```
$ Weld definition with format ELEMID
PWELD      10      1      3.
CWELD      761     10     10001    ELEMID
           259     22
CWELD      762     10     10002    ELEMID
           299     62
CWELD      763     10     10003    ELEMID
           639     402
```

```
CWELD      764      10      10004      ELEMID
           679      442
```

\$ Weld definition with format ELPAT

```
PWELD      10      1      3.
CWELD      761      10      10001      ELPAT
           259      22
CWELD      762      10      10002      ELPAT
           299      62
CWELD      763      10      10003      ELPAT
           639      402
CWELD      764      10      10004      ELPAT
           679      442
```

Below is the print out for the element forces. The forces are not symmetric for the format ELEMID, see the first part with the title CWELDC. The forces come out symmetric for the format ELPAT, see the second part with the title CWELDP.

1	HAT SECTION WELDED BEAM, NON-SYM ELEMID OPTION				MAY 16, 2003	MSC.NASTRAN	5/15/03	PAGE	16
	MZ								
0							SUBCASE 3		
	FORCES IN WELD ELEMENTS ( CWELDC )								
ELEMENT	BEND-MOMENT END-A			BEND-MOMENT END-B		- SHEAR -		AXIAL	
ID	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX	
761	-1.422846E-02	1.642372E-01	-8.496036E-03	-3.994170E-01	-5.732428E-03	5.636542E-01	-5.534064E-04	4.985318E-02	
762	-1.146957E-02	1.216688E-01	-1.872424E-02	-4.326421E-01	7.254668E-03	5.543109E-01	8.758141E-05	4.411347E-02	
763	-2.167158E-02	-1.589983E-01	-1.698017E-02	4.059372E-01	-4.691408E-03	-5.649355E-01	5.010240E-04	4.672464E-02	
764	-2.205327E-02	-1.208619E-01	-3.047966E-02	4.344850E-01	8.426391E-03	-5.553468E-01	-1.337890E-04	4.600000E-02	

1	HAT SECTION WELDED BEAM, NON-SYM ELPAT OPTION				MAY 16, 2003	MSC.NASTRAN	5/15/03	PAGE	16
	MZ								
0							SUBCASE 3		
	FORCES IN WELD ELEMENTS ( CWELDP )								
ELEMENT	BEND-MOMENT END-A			BEND-MOMENT END-B		- SHEAR -		AXIAL	
ID	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX	
761	-9.314521E-04	1.277454E-01	4.733397E-03	-4.993157E-01	-5.664849E-03	6.270611E-01	-2.275192E-04	5.223577E-02	
762	9.314521E-04	1.277454E-01	-4.733397E-03	-4.993157E-01	5.664849E-03	6.270611E-01	2.275192E-04	5.223577E-02	
763	-9.314521E-04	-1.277454E-01	4.733397E-03	4.993157E-01	-5.664849E-03	-6.270611E-01	2.275192E-04	5.223577E-02	
764	9.314521E-04	-1.277454E-01	-4.733397E-03	4.993157E-01	5.664849E-03	-6.270611E-01	-2.275192E-04	5.223577E-02	

## 5.12 New K6ROT Default

### Introduction

MSC.Nastran's CQUAD4 and CTRIA3 shell elements are based on the Mindlin-Reissner shell theory. These shell elements do not have stiffness for the rotational degrees-of-freedom normal to the shell plane. This is often called the drilling degree-of-freedom. The rank deficiency is caught and constrained if AUTOSPC is turned on. The combination of defaults in MSC.Nastran 2001 (PARAM, SNORM, 20. and PARAM, K6ROT, 0.) has revealed that an intentional or unintentional loading of the rotation around the normal may cause mechanisms and false load transfers. Therefore, it has been decided to change the default from K6ROT from 0. to 100. in MSC.Nastran 2004.

### Benefits

The small penalty stiffness in the 6<sup>th</sup> degree-of-freedom makes the CQUAD4 and CTRIA3 shell elements more robust, cures all problems with mechanisms and does not affect accuracy.

### Theory

K6ROT specifies the scaling factor of the penalty stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The contribution of the penalty term to the strain energy functional is

$$\Pi_p = 10^{-6} \text{K6ROT} \frac{1}{2} G \int_A (\Theta_z - \Omega_z)^2 t dA$$

with

$$\Omega_z = \frac{1}{2} \left( \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right)$$

where A is the area of the shell element, t is the shell thickness, G is the in plane shear modulus, using the MID1 material identification number on the PSHELL Bulk Data entry. The inplane displacements  $u_x, u_y$  and the normal rotation  $\Theta_z$  are shown in [Figure 5-22](#). The penalty term is constructed so that it has little affect upon the membrane and bending modes of a flat shell. The patch test is passed. The normal rotation should not be interpreted as a true rotation. The penalty stiffness removes the

singularity in the normal rotation. A higher value than  $K6ROT=100$ . is not recommended because unwanted stiffening effects may occur. If  $K6ROT=0$ . is specified, the singularity can be suppressed with the parameter AUTOSPC.

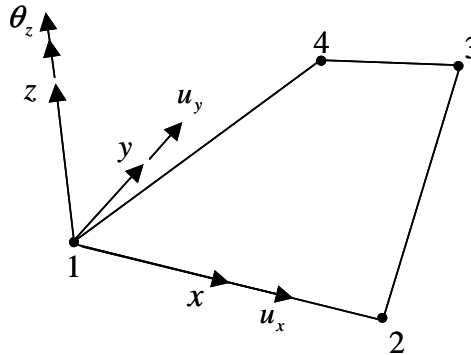


Figure 5-22 In plane displacements  $u_x$ ,  $u_y$  and normal rotation  $\Theta_z$

## Guidelines and Recommendations

- The a-set may increase in size when K6ROT is switched from 0. to 100.
- In 3D analysis, one of the three rotations is zero if AUTOSPC is turned on and  $K6ROT=0$ . The rotations have values in all three directions when  $K6ROT=100$ .
- We recommend shell normals when K6ROT is turned on. Without shell normals, K6ROT has unwanted effects on results, like stiffening and false load transfer in the 6<sup>th</sup> degree-of-freedom.
- If you want membrane only or plane strain, you should add `PARAM,K6ROT,0.` to the input file.
- If you used SPCs to get rid of expected singularities, you should either remove those SPCs or add `PARAM,K6ROT,0` to the input file.

## Examples

All MacNeal-Harder standard test problems passed with  $K6ROT=100$ . The results are either the same or the difference is not more than 0.1% in the displacements. A discussion about the influence of K6ROT can be found in Reference [24](#).





CHAPTER

6

## External Superelements

- Enhancements to External Superelements

## 6.1 Enhancements to External Superelements

Prior to MSC.Nastran 2004, the usage of external superelements involved a three-step procedure summarized under the description of the EXTOUT, EXTDROUT and EXTDR parameters in “[Parameter Descriptions](#)” on page 574 of the *MSC.Nastran Quick Reference Guide*. This procedure consists of the following steps:

- Step 1 -- External Superelement Creation  
Create the external superelements using EXTOUT parameter options.
- Step 2 – Superelement Assembly and Analysis  
Analyze the assembled configuration containing the external superelements created in Step 1 and save the results using EXTDROUT parameter options for subsequent operations in Step 3.
- Step 3 – External Superelement Data Recovery  
Using the results from Step 2, perform data recovery operations for the external superelements by specifying EXTDR parameter options.

The above procedure, although perfectly valid and still supported, has the following disadvantages:

- The setup of the data for Step 2 (the assembly/analysis run) requires manual preparation of data which is very cumbersome, inconvenient, and prone to errors.
- The recovery of the results for the external superelement requires the third step.

In order to avoid the above difficulties, a new procedure for the use of external superelements has been developed in MSC.Nastran 2004. This procedure makes use of the new Case Control command EXTSEOUT and consists of only the following two convenient steps.

- Step 1 – External Superelement Creation  
Create the external superelements using the new EXTSEOUT Case Control command.  
  
The EXTSEOUT command, whose description is given in the *MSC.Nastran Quick Reference Guide*, specifies the data to be saved for the external superelements, as well as the medium on which this data is to be saved. The EXTOUT parameter of the old procedure is no longer needed in this creation run.

This step is run as a standard MSC.Nastran job. The boundary points for the external superelements are defined by ASET/ASET1 entries. If the reduction involves component mode reduction, the generalized coordinates are specified via QSET/QSET1 entries. The fixity of the boundary points for component mode reduction is specified via the new BNDFIX/BNDFIX1 and BNDFREE/BNDFREE1 Bulk Data entries (or the existing BSET/BSET1 and CSET/CSET1 entries to which they are equivalent). By default, all boundary points are considered fixed for component mode reduction.

The output for the external superelements is generated in Step 2. (This output consists of displacements, velocities, accelerations, SPC forces and element stresses and forces.) However, in order for this output to be generated in Step 2, the output requests must be specified in the creation runs of Step 1. It should be emphasized that the only output requests for the external superelements that are honored in Step 2 are those that are specified in the creation runs of Step 1.

- Step 2 – Superelement Assembly, Analysis and Data Recovery

Analyze the assembled configuration containing the external superelements created in Step 1 and obtain the desired results, *including those for the external superelements*.

In this run, the output requests for all superelements, including those for the external superelements, may be specified. (However, as indicated earlier, the only output requests for the external superelements that are honored in Step 2 are those that were specified in the creation runs of Step 1.)

The setup of the Main Bulk Data for this step is the same as that for Step 2 of the old procedure. Thus, one SEBULK Bulk Data entry is required for each external superelement from Step 1 that is used in this run. Furthermore, if this run involves coincident points corresponding to the boundary points of one or more of the external superelements, appropriate SECONCT (or SEBNDRY) entries must be specified. These data can be generated automatically during the creation run of Step 1 by specifying the ASMBULK option for the EXTSEOUT Case Control command, thereby avoiding the need for manual preparation of the required data.

Unlike the old procedure, the new procedure does not normally need explicit definition of the data related to the external superelements in this run because such data are saved on the medium specified by the EXTSEOUT Case Control command in the creation run. There are, however, two cases in which appropriate data for the applicable external superelements must be explicitly specified in the assembly run after the

relevant BEGIN SUPER entries. The first case is when the DMIGPCH option has been employed in the creation run. In this case, the data required for the external superelement specification in the assembly run (including the BEGIN SUPER entry) are automatically generated and placed on the standard punch file during the creation run. These data may then be subsequently inserted into the assembly run input data. The second case is when the data in the creation run contain PLOTEL entries involving interior points of the external superelements. In this case, the tedious manual preparation of the data required for the external superelement specification in the assembly run may be avoided by requesting automatic generation of such data (including the BEGIN SUPER entry) by specifying the EXTBULK option for the EXTSEOUT Case Control command in the creation run. This data may then be subsequently inserted into the assembly run input data.

## CHAPTER

# 7

# Coupled Fluid-Structure Analysis

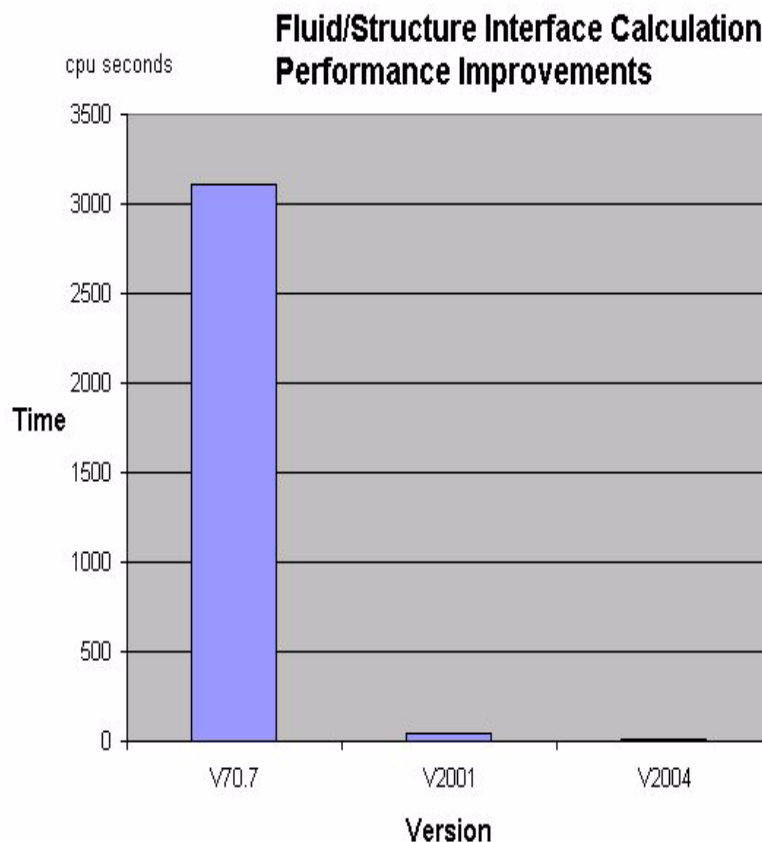
- New Body In White Method
- Direct Input of Interface [A] Matrix
- Acoustic Modal Participation Factors
- Acoustic XY Plots, Random, and Restarts
- Acoustic Source Change
- Efficiency Enhancement in Virtual Mass
- Modifications to the Computation of Fluid and Structural Modes in SOL 103

## 7.1 New Body In White Method

### Introduction

A new method (BW) of calculating fluid structure interface has been implemented. Speed improvements are such that an automobile model that consists of 120,000 - 150,000 shell elements and 15,000 - 20,000 fluid elements now takes only about 60 seconds for calculating the fluid/structure interface matrix, which is a 1000 times speed improvement.

For instance, the following is a typical performance increase for a medium sized model:



You can request a .pch file that represents fluid and structure "skin" (param,skinout,punch) in order to check interface accuracy. A utility program is provided for viewing and inspecting the completeness and accuracy of the "skin" coverage (see the following example).

Two methods are available on the ACMODL entry for calculating the interface [A] matrix.

The format for the new BW method is:

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	SKNEPS	DSKNEPS	
	INTOL	ALLSSET	SRCHUNIT						

## Example

ACMODL							.25		

For the BW method, no ACMODL entry is necessary if defaults are used (suggested for first run). A special license is required for this entry.

To use the old CP method the METHOD field (field 7) must be specified as CP:

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD			

## Example

ACMODL						CP			
--------	--	--	--	--	--	----	--	--	--

In most cases, the BW (Body in White) method provides a good fluid/structure interface matrix. However, changes in the interface can be made by either adding or deleting structural elements in the interface calculation.

A set of utility programs are provided to aid in viewing of the elements in the interface.

## New BW Coupling Algorithm

The purpose of this algorithm is to overcome the difficulties of many structural elements of various sizes overlapping each other. Also, this algorithm should be somewhat insensitive to small inadvertent holes in the structure.

The algorithm proceeds as follows:

1. Use the current search algorithm to locate the fluid free faces and the corresponding structural element faces.

2. For a fluid free face and its list of structural element faces (that were determined by boxing normal to the fluid element) do as follows:
  - a. For each fluid free face establish a face coordinate system.
  - b. Determine the resultant pressure force for each grid on the fluid element by the relationship

$$R_i = \int_S [N_f] dS \{p_i = 1; p_j = 0\} \quad i = 1, N \text{ grid/elem}$$

- c. Resolve this resultant pressure force for a unit grid pressure to the grids of the fluid element by the expression (determined by virtual work)

$$\{F_i\} = \int_S \{N_f\}^T [N_f] dS \{p_i\}$$

- d. Using the origin of the free fluid face, determine the center of pressure  $(X_{P_i}, Y_{P_i})$ . The relationship will be of the form

$$X_{P_i} = \sum_j^{\text{grids}} \frac{F_i}{R_i} (X_j - X_0)$$

$$Y_{P_i} = \sum_j^{\text{grids}} \frac{F_i}{R_i} (Y_j - X_0)$$

- e. Using rigid relations to consider only a unit motion normal to the fluid face with the appropriate moment relationships, determine the resulting load distribution at the grids of each of the structural elements. The area of each structural element projected normal to the fluid element will be used as a weighting function. The expression is of the form:

$$\{F_j\} = [W][R]([R]^T[W][R])^{-1} \begin{Bmatrix} R_i \\ 0 \\ 0 \end{Bmatrix}$$

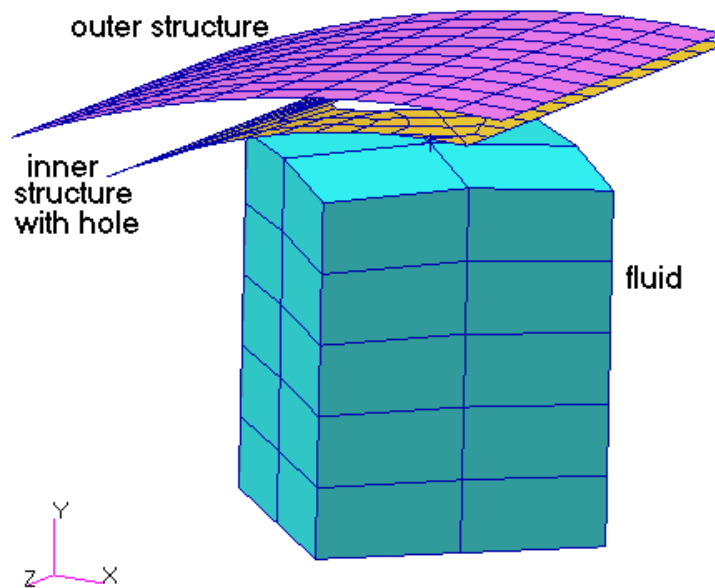
where  $\{F_j\}$  is the vector of resulting load distribution at the grids of each of the  $j$  structural elements.  $[W]$  is a diagonal weighting matrix.  $[R]$  is the rigid transformation matrix.



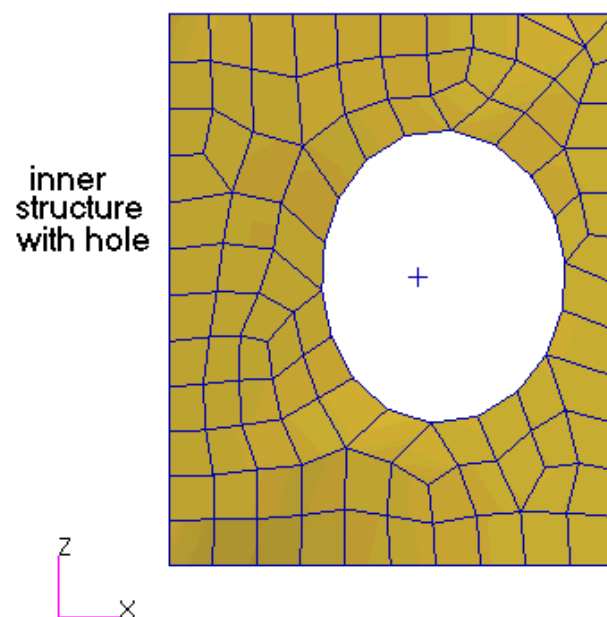
- f. Loop over each grid of the fluid element and accumulate the forces at the structural grids.
  - g. Note, with this algorithm, we do not worry if a structural element is only partially wetted by the fluid. We always require rigid body equilibrium.
3. Repeat for the next fluid element and its associated group of structural elements. Accumulate the forces at the structural grids.

### Example (acoustic1.dat)

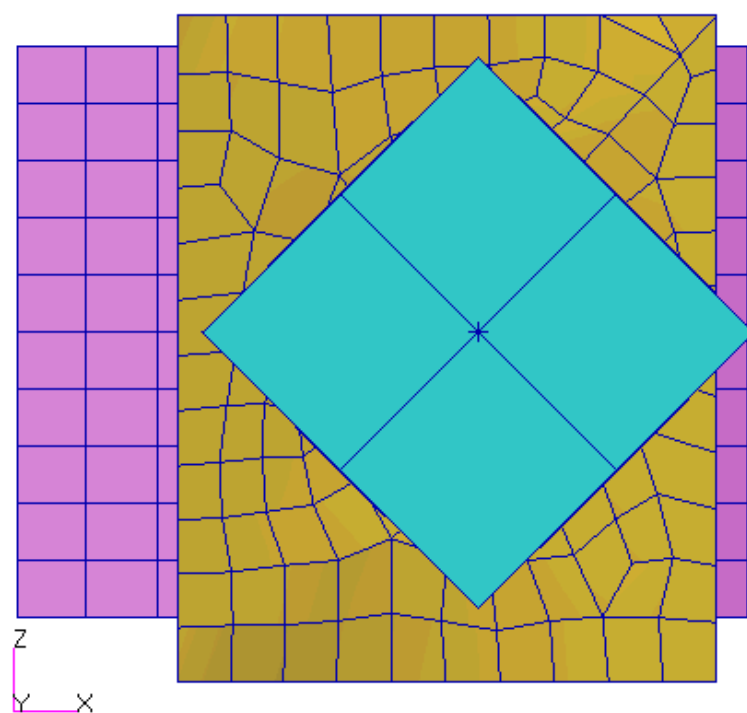
The model shown below, "acoustic1.dat", is a square column of fluid with a round top which interacts with an inner structural plate with a hole in it and an outer structural plate:



The inner plate has a hole as shown:



The relative location of the fluid face and the structural plates is shown:



When the model is run, and PARAM,SKINOUT,PUNCH is used, the fluid and structure interface elements and grids found by the search are written to the acoustic1.pch file shown:

```
$ *****
$ FSI - IDS OF FLUID ELEMENTS AT INTERFACE
$ *****
$ This bulk data entry may by referenced from the FSET field on the
$ ACMODL bulk data entry (with the ELEMENTS option) to remove unwanted
$ fluid faces from the fluid-structure interface.
SET1      1          5001      5001      5001      5002      5002      5003      5003
          5004
$ *****
$ FSI - IDS OF STRUCTURE ELEMENTS AT INTERFACE
$ *****
$ This bulk data entry may by referenced from the SSET field on the
$ ACMODL bulk data entry (with the ELEMENTS option) to remove unwanted
$ structure faces from the fluid-structure interface.
SET1      2          1041      1037      1038      1040      1039      1083      1094
          1092      1091      1090      1082      1095      1093      1063      1081
          1088      1064      1065      1062      1066      1053      1054      1055
          1087      1052      1056      1057      1086      1051      1096      1058
          1019      1028      1085      1073      1050      1072      1076      1084
          1020      1070      1059      1013      1077      1071      1060      1012
          1010      1061      1009      1008      1046      1047      1007      1006
          1044      1045
$ *****
$ FSI - IDS OF FLUID GRIDS AT INTERFACE
$ ***** $
This bulk data entry may by referenced from the FSET field on the
$ ACMODL bulk data entry (with the GRIDS option) to remove unwanted
$ fluid faces from the fluid-structure interface.
SET1      3          5001      5002      5003      5004      5005      5006      5007
          5008      5009      5010      5011      5012      5013      5016
$ *****
$ FSI - IDS OF STRUCTURE GRIDS AT INTERFACE
$ *****
$ This bulk data entry may by referenced from the SSET field on the
$ ACMODL bulk data entry (with the GRIDS option) to remove unwanted
$ structural faces from the fluid-structure interface.
SET1      4          1004      1005      1006      1007      1008      1009      1010
          1017      1018      1027      1028      1029      1030      1031      1032
          1040      1041      1042      1045      1047      1048      1049      1050
          1051      1051      1053      1054      1055      1056      1057      1058
          1059      1060      1061      1062      1063      1064      1065      1066
          1071      1072      1073      1074      1075      1078      1079      1080
          1081      1082      1084      1085      1086      1087      1088      1093
          1094      1095      1096      1097      1098      1099      1101      1102
          1103      1104      1105      1106      1107      1108      1109      1110
          1111      1112      1113      1114      1115      1116      1117      1118
          1119      1120      1121      1122      1123      1125      1126      1127
          1128      1129
```

These element and grid SET1 Bulk Data entry lists serve two purposes. First, they can be used in a finite element preprocessor as an accuracy check by graphically displaying which elements are used in the analysis, and second they can limit or expand what elements or grids are used in fluid/structure interface.

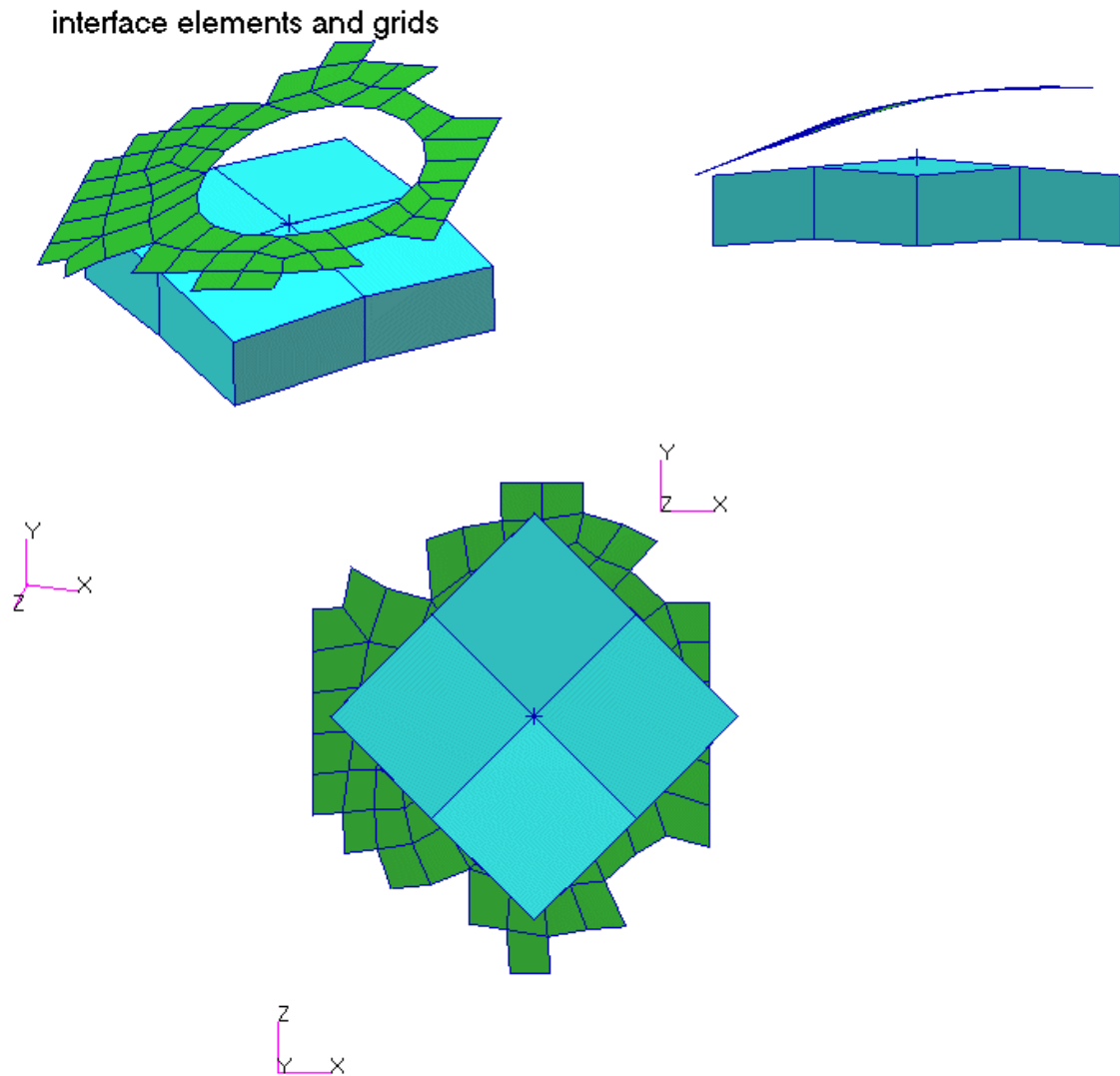
To check the fluid/structure interface, MSC supplies a perl program named "acnaspat.pl" which reads the above .pch file and writes a MSC.Patran session file. The perl script is located at "/msc2004/util". To run the script on UNIX, type, for instance:

```
/msc2004/util/acnaspat.pl acoustic1.pch
```

which creates a MSC.Patran session file named "acoustic1\_flstgrp.ses", which when run in MSC.Patran will create two groups:

acoustic1_fluid	the fluid interface elements and grids
acoustic1_structure	structure interface elements and grids

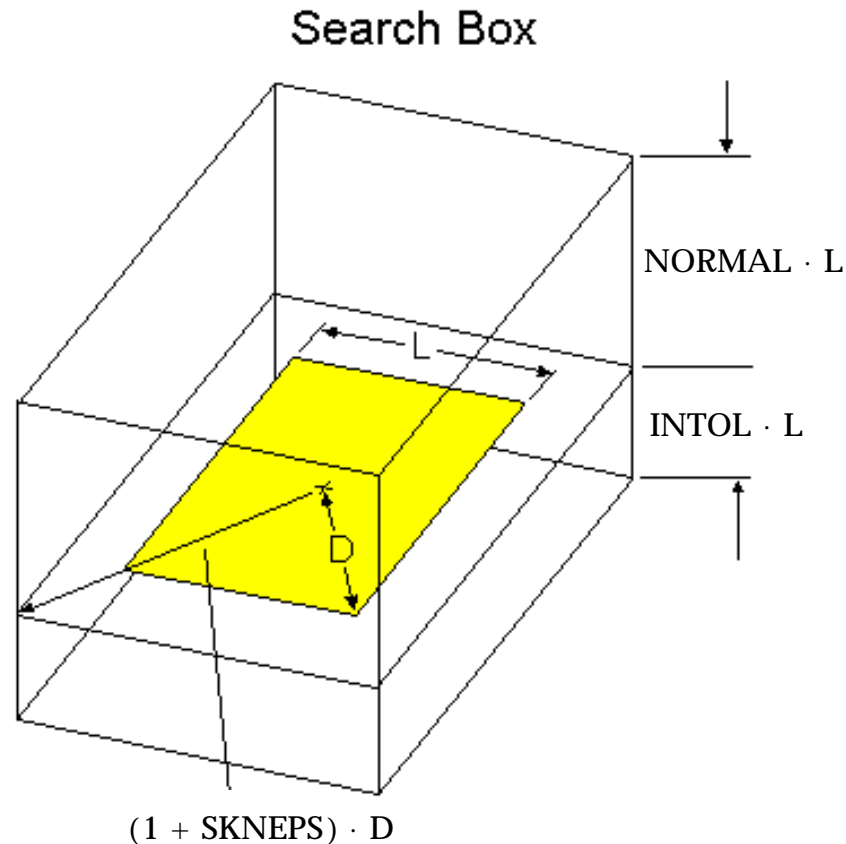
These two groups are shown below:



Two things are indicated:

- extra structure elements were selected that do not project directly onto the fluid faces,
- only elements from the inner structural plate were selected.

Which structural elements to be included in the fluid/structure interface can be adjusted in one of two ways. The first is by increasing the size of the search box by changing the defaults on the NORMAL, INTOL, and SKNEPS fields of the ACMODL Bulk Data entry.



- $\text{NORMAL} \cdot L$  - Search box height in positive normal direction from fluid face, where  $L$  is smallest fluid face side length,
- $\text{INTOL} \cdot L$  - height in negative normal direction,
- $(1 + \text{SKNEPS}) \cdot D$  - in-plane width added to fluid face dimensions where  $D$  is the distance from the center of the fluid face to the grid point.

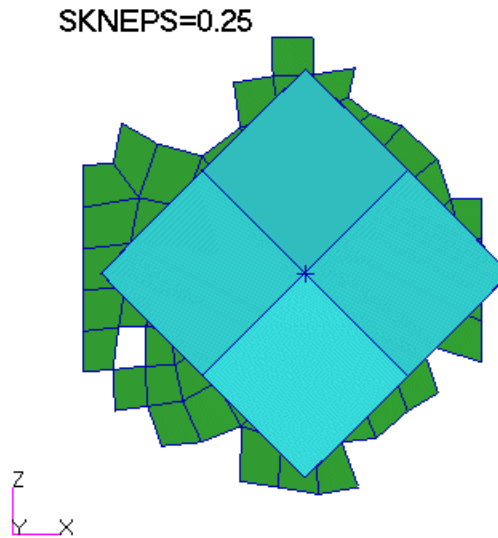
If the search cannot find any structural faces within the search box, it will widen the search box by using the value of the DSKNEPS field on the ACMODL Bulk Data entry.

Structural faces, whose centroids fall within the search box, are included in the fluid/structure interface. The effect of the structural face in the fluid/structure interface matrix is weighed according to areas, angles and distances between it and the fluid face that it found. (See “[New BW Coupling Algorithm](#)” on page 301.)

In this case, looking at the interface elements, the normal distance of the search box appears good, but too many structural elements on the sides of the fluid faces are being selected. By adding an ACMODL Bulk Data entry with SKNEPS decreased from the default of 0.5 to 0.25:

```
acmod1 , , , , , , 0.25
```

you get the following interface elements:



The structural elements are closer to what is wanted, but there are still too many.

The second way to control which elements are used in the fluid/structure interface is to use the INFOR="elements" or "grids" and the FSET and SSET fields on the ACMODL Bulk Data entry.

First the elements not wanted are removed from the "acoustic2\_struct" group in MSC.Patran. For this example, any structure element with less than 50% of it's area projected onto the fluid faces is removed. The final structural interface elements are shown:



- graphical selection
- searching by property or material type

Elm 1008 1013 1044 1045 1050:1059 1062:1064 1070 1072 1082 1085 1090 1091

```
./acpatnas.pl element1.txt
```

SET1	1	1008	1013	1044	1045	1050	1051	1052
	1053	1054	1055	1056	1057	1058	1059	1062
	1063	1064	1070	1072	1082	1085	1090	1091

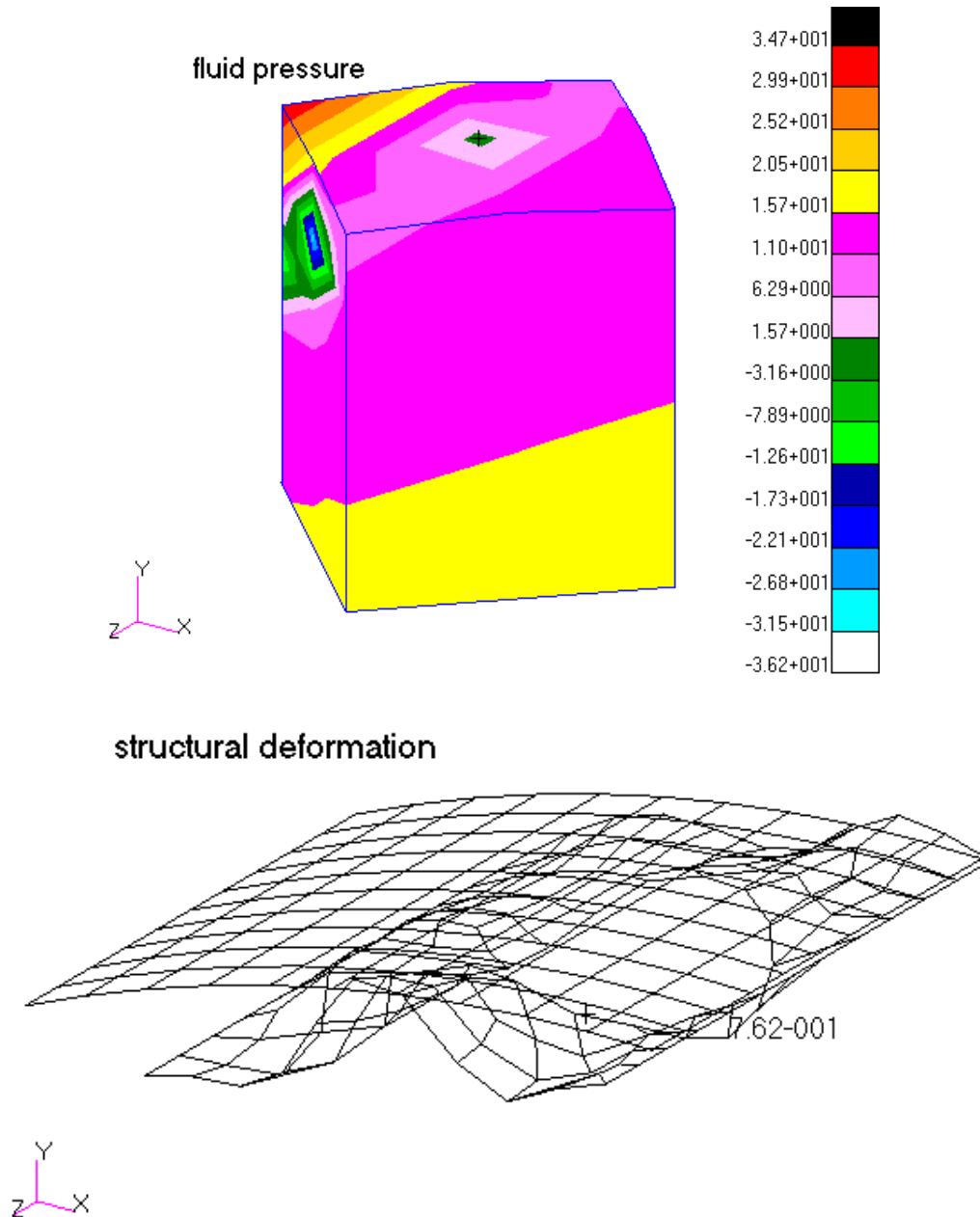


The SET1 SID is changed from "1" to "2" so as not to conflict with the fluid element SET1 from the "acoustic1.pch" file. When these two SET1 Bulk Data entries are put in the MSC.Nastran file along with:

```
acmod1,,elements,1,2
```

then only the above structural elements are used in the fluid/structure interface.

The following plots show the fluid pressure distribution and the structural deflection.



The model is loaded with acoustic power on the upper left hand corner of the fluid, thus the high pressure in that area. The effect of the hole in the inner structure can be seen as a slight reduction in fluid pressure at the hole.

The structural deformation shows that the outer structure is unaffected by the pressure, as expected.

This concludes the example.

### **Parallel Structural Element Meshes in the Search Box:**

As noted above, the outer parallel structural surface did not have any elements selected for inclusion in the interface, even though some were in the search box. This is because parallel surfaces are ignored by using a connectivity technique. The first structural element face found in the search box is the one closest to the searching fluid face. All elements connected to this first element are checked whether they are in the search box. Once an element is found to be outside of the box, it is no longer used for the connectivity check. This tends to eliminate parallel element meshes unless the connection is also in the search box, in which case the parallel elements must be manually removed using the SET1 technique above.

A problem occurs with this technique if a connectivity discontinuity occurs in the search box. Only one side of the discontinuity will be selected, causing elements to be missed in the interface search. The workaround for this is to use the ALLSET="YES" field on the ACMODL entry in which case all structural elements or grids in the SET1 entries are forced to be included in the fluid/structure interface whether the connectivity search finds them or not. You must also set INTER="DIFF", INFOR="ELEMENTS" or "GRIDS" and SSET must be properly filled out. This method forces an interface between the listed structural elements and the nearest fluid faces.

### **Absolute Versus Relative Search Box Dimensions:**

If SRCHUNIT="ABS", the box dimensions are equal to the value of NORMAL, INTOL, and SKNEPS, rather than times a fluid face dimension. This is useful when the distance between the structural interface and the fluid interface is relatively constant instead of relative to the fluid element size.

## 7.2 Direct Input of Interface [A] Matrix

You can input the [A] matrix directly using DMIG with the A2GG Case Control command. The format of the A2GG input is similar to the other G-type DMIG input (e.g., K2GG). By default, the A2GG fluid-structure coupling matrix will be added to the computed coupling matrix. To replace the computed coupling matrix with the selected A2GG matrix, set PARAM,ASCOUP,NO.

### Examples

- $A2GG = ADMIG$
- $A2GG = ADMIG1, ADMIG2, ADMIG3$
- $A2GG = 1.5*ADMIG1, .5*ADMIG2$

## 7.3 Acoustic Modal Participation Factors

### Introduction

Three new case control commands have been added to control the acoustic analysis output with a set of parameters:

- **FLSFSEL** - Fluid-structure superelement and frequency selection parameters
- **FLSPOUT** - Fluid-structure mode participation output requests parameters
- **FLSTCNT** - Miscellaneous fluid-structure control

The parameters can be either specified in these case control commands or as bulk data parameters. If bulk data parameters are used, then the parameter values may be replaced with numeric equivalents but usually are the same as the Case Control values. But check the “[Parameter Descriptions](#)” on page 574 of the *MSC.Nastran Quick Reference Guide* to make sure.

These commands must be above the subcase level in the Case Control Section.

Participation factors not printed are still available for plotting from the .op2 file or with NASPLOT.

## Case Control Command Summary

### FLSFSEL

Control for Fluid-Structure Frequency Selection

$$\begin{aligned} \text{FLSFSEL} \quad & \left[ \text{LFREQFL} = \left\{ \frac{0.0}{fl_1} \right\} \right], \left[ \text{HFREQFL} = \left\{ \frac{1. + 30}{fl_2} \right\} \right], \\ & \left[ \text{LFREQ} = \left\{ \frac{0.0}{fs_1} \right\} \right], \left[ \text{HFREQ} = \left\{ \frac{1. + 30}{fs_2} \right\} \right], \\ & \left[ \text{LMODESFL} = \left\{ \frac{0}{mf} \right\} \right], \left[ \text{LMODES} = \left\{ \frac{0}{ms} \right\} \right], \\ & \left[ \text{FLUIDSE} = \left\{ \frac{0}{seidf} \right\} \right] \end{aligned}$$

These are all previously existing parameters. Most are used to select which modes are used in the modal solution. The exception is the FLUIDSE parameter, which specifies the fluid superelement number.

**FLSPOUT****Control for Fluid-Structure Mode Participation Output**

$$\begin{aligned}
\text{FLSPOUT} \quad & \left[ \text{FLUIDMP} = \begin{Bmatrix} \text{ALL} \\ n_{\text{modes}} \\ \text{NONE} \end{Bmatrix}, \left[ \text{GRIDFMP} = \begin{Bmatrix} \text{setf} \quad \text{ALL} \\ \text{participations} \end{Bmatrix} \right] \right. \\
& \left[ \text{OUTFMP} = \begin{Bmatrix} \text{ALL} \\ p_{\text{highest}} \\ \text{NOPRINT} \end{Bmatrix}, \left[ \text{FEPS} = \begin{Bmatrix} 1. - 11 \\ \text{epsf} \end{Bmatrix} \right], \right. \\
& \left[ \text{ARF} = \begin{Bmatrix} 0.95 \\ \text{arf}_v \end{Bmatrix} \right], \\
& \left[ \text{STRUCTMP} = \begin{Bmatrix} \text{ALL} \\ m_{\text{modes}} \\ \text{NONE} \end{Bmatrix}, \left[ \text{OUTSMP} = \begin{Bmatrix} \text{ALL} \\ q_{\text{highest}} \\ \text{NOPRINT} \end{Bmatrix} \right] \right. \\
& \left[ \text{PANELMP} = \begin{Bmatrix} \text{ALL} \\ \text{setp} \text{participations} \\ \text{NONE} \end{Bmatrix}, \left[ \text{GRIDMP} = \begin{Bmatrix} \text{ALL} \\ \text{setg} \text{participations} \\ \text{NONE} \end{Bmatrix} \right] \right. \\
& \left[ \text{SEPS} = \begin{Bmatrix} 1. - 11 \\ \text{epss} \end{Bmatrix} \right], \left[ \text{ARS} = \begin{Bmatrix} 0.95 \\ \text{ars}_v \end{Bmatrix} \right] \\
& \left[ \text{PSORT} = \begin{Bmatrix} \begin{Bmatrix} \text{ABSOLUTE} \\ \text{REAL} \\ \text{IMAGINARY} \end{Bmatrix}, \begin{Bmatrix} \text{DESCENDING} \\ \text{ASCENDING} \end{Bmatrix} \end{Bmatrix}, \left[ \text{O2E} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \right] \right]
\end{aligned}$$

- **FLUIDMP=n, STRUCTMP=m, and PANELMP=setp**
  - Previously existing parameters
  - Fluid, structure and panel modal participation factor requests
  - First "n" and "m" modes
  - setp is a case control set referencing bulk data PANEL names
- **GRIDFMP=setf**
  - Set of fluid grids for output of modal participation factors

- **GRIDMP=setg**
  - Set of panel structural grids for output of panel grid participation factors
  - Default = "all", output of structural grid participation factors for all panel grids
  - FLUIDMP and STRUCTMP must both be specified for GRIDMP to become active
- **OUTFMP=p, OUTSMP=q**
  - highest fluid (p) and structure (q) participation factors output
- **FEPS=value, SEPS=value**
  - Fluid (FEPS) and structure (SEPS) participations below these values are not printed.
- **ARF=value, ARS=value**
  - Fluid (ARF) and structural (ARS) participation factors with values below ARF \* or ARS\* the maximum value for that mode will not be printed.
- **PSORT - Sorting options**
  - PSORT values must occur in pairs such as (ABSOLUTE, DESCENDING).
- **O2E=YES or NO**
  - Whether "alternate" form of participation factor plotting is available or not
  - "Regular" plotting is participation factor versus excitation frequency  
Use "MODE" "ptype" in XYPLOT command  
See XY plot section
  - "Alternate" plotting is participation factor versus natural frequency for each excitation frequency  
Use "FREQ" "ptype" in XYPLOT command  
See XY plot section
  - "NO", no "alternate" plotting is available, only "regular" (default)
  - "YES", "alternate" pots are available
  - O2E=YES will double op2 file size

**FLSTCNT**

Miscellaneous fluid-structure control parameters

$$\begin{aligned}
 \text{FLSTCNT} \quad & \left[ \text{ACSYM} = \left\{ \frac{\text{YES}}{\text{NO}} \right\} \right], \left[ \text{ACOUT} = \left\{ \frac{\text{PEAK}}{\text{RMS}} \right\} \right] \\
 & \left[ \text{PREFDB} = \left\{ \frac{1.0}{prp} \right\} \right], \left[ \text{ASCOUP} = \left\{ \frac{\text{YES}}{\text{NO}} \right\} \right] \\
 & \left[ \text{SKINOUT} = \left\{ \begin{array}{c} \text{NONE} \\ \text{PUNCH} \\ \text{PRINT} \\ \text{ALL} \end{array} \right\} \right]
 \end{aligned}$$

- **ACSYM**, **ACOUT**, **PREFDB**, and **ASCOUP** are existing parameters now accessible in the case control. See the *MSC.Nastran Quick Reference Guide* for more information on these.
- **SKINOUT** is a new parameter used to list the elements used in the fluid/structure interface. See the “**New Body In White Method**” on page 300, for more information on this parameter.



## Modal Participation Theory

The fluid and structure equations of motion are:

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix} \quad \text{Eq. 7-1}$$

where:

$M$  = mass

$B$  = damping

$K$  = stiffness

$u$  = displacement

$p$  = pressure

$s$  = structure

$f$  = fluid

$A$  = interface matrix

$P$  = load

The participation factors are obtained as follows:

$$\{u_s\} = [\Phi_s] \{\xi_s\} \quad \text{Eq. 7-2}$$

$$\{p\} = [\Phi_f] \{\xi_f\} \quad \text{Eq. 7-3}$$

where  $[\Phi_s]$  are the uncoupled, undamped structural modes and  $[\Phi_f]$  are the uncoupled, undamped, rigid-wall acoustic modes. The vectors  $\{\xi_s\}$  and  $\{\xi_f\}$  are the modal amplitudes. Substituting these relations into [Eq. 7-1](#) and pre-multiplying by the modal matrices, we get the equation

$$\begin{bmatrix} \Phi_s^T M_s \Phi_s & 0 \\ -\Phi_f^T A^T \Phi_s & \Phi_f^T M_f \Phi_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} + \begin{bmatrix} \Phi_s^T B_s \Phi_s & 0 \\ 0 & \Phi_f^T B_f \Phi_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} + \begin{bmatrix} \Phi_s^T K_s \Phi_s & \Phi_s^T A \Phi_f \\ 0 & \Phi_f^T K_f \Phi_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = \begin{Bmatrix} \Phi_s^T P_s \\ \Phi_f^T P_f \end{Bmatrix} \quad \text{Eq. 7-4}$$

or

$$\begin{bmatrix} m_s & 0 \\ -a^T & m_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} + \begin{bmatrix} b_s & 0 \\ 0 & b_f \end{bmatrix} \begin{Bmatrix} \dot{\xi}_s \\ \dot{\xi}_f \end{Bmatrix} + \begin{bmatrix} k_s & a \\ 0 & k_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = \begin{Bmatrix} Q_s \\ Q_f \end{Bmatrix} \quad \text{Eq. 7-5}$$

Define the following harmonic solution forcing frequency  $\omega$ .

$$\{\xi\} = \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = e^{i\omega t} \quad \text{Eq. 7-6}$$

$$\{\dot{\xi}\} = i\omega e^{i\omega t} \quad \text{Eq. 7-7}$$

$$\{\ddot{\xi}\} = -\omega^2 e^{i\omega t} \quad \text{Eq. 7-8}$$

For the bottom equation of [Eq. 7-5](#), we get:

$$\omega^2 [a]^T \{\xi_s\} + [-\omega^2 [m_f] + i\omega [b_f] + [k_f]] \{\xi_f\} = \{Q_f\} \quad \text{Eq. 7-9}$$

Define  $[Z_2]$  as

$$[Z_2] = [-\omega^2 [m_f] + i\omega [b_f] + [k_f]]^{-1} \quad \text{Eq. 7-10}$$

then

$$\{\xi_f\} = -\omega^2 [Z_2] [a]^T \{\xi_s\} + [Z_2] \{Q_f\} \quad \text{Eq. 7-11}$$

The fluid mode participation is defined as

$$[P_f] = [\Phi_f] [\{\xi_f\}] \quad \text{Eq. 7-12}$$

where  $[\{\xi_f\}]$  is the diagonalized vector of fluid modal amplitudes per excitation frequency.

The structure mode participation is defined as

$$[P_s] = -\omega^2 [\Phi_s] [Z_2] [a]^T [\{\xi_s\}] \quad \text{Eq. 7-13}$$

where  $[\{\xi_s\}]$  is the diagonalized vector of structural modal amplitudes per excitation frequency.

The fluid load participation is defined as

$$\{P_l\} = [\Phi_f][Z_2]\{Q_f\} \quad \text{Eq. 7-14}$$

The fluid-structure panel participation is defined as

$$[P_p] = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T [A]_{\text{panel}}^T [\Phi_s][\{\xi_s\}] \quad \text{Eq. 7-15}$$

The fluid-structure panel-grid participation is defined as

$$[P_g]_i = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T \left[ \left\{ A_b^T \right\} \right] [\Phi_s] [\xi_s] \quad \text{Eq. 7-16}$$

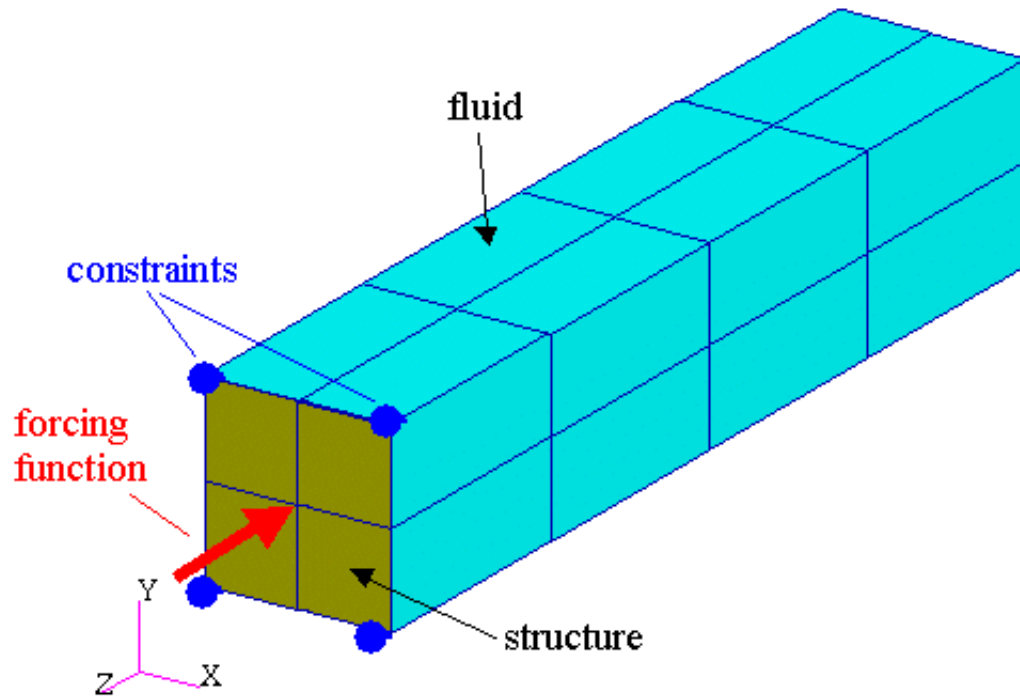
where

$$\left\{ A_b^T \right\}$$

are the columns extracted from the b-th boundary panel for panel grid i and  $[\Phi_s]$  are the rows of the structural modal matrix corresponding to panel grid i.

## Modal Participation Example (acoustic2.dat)

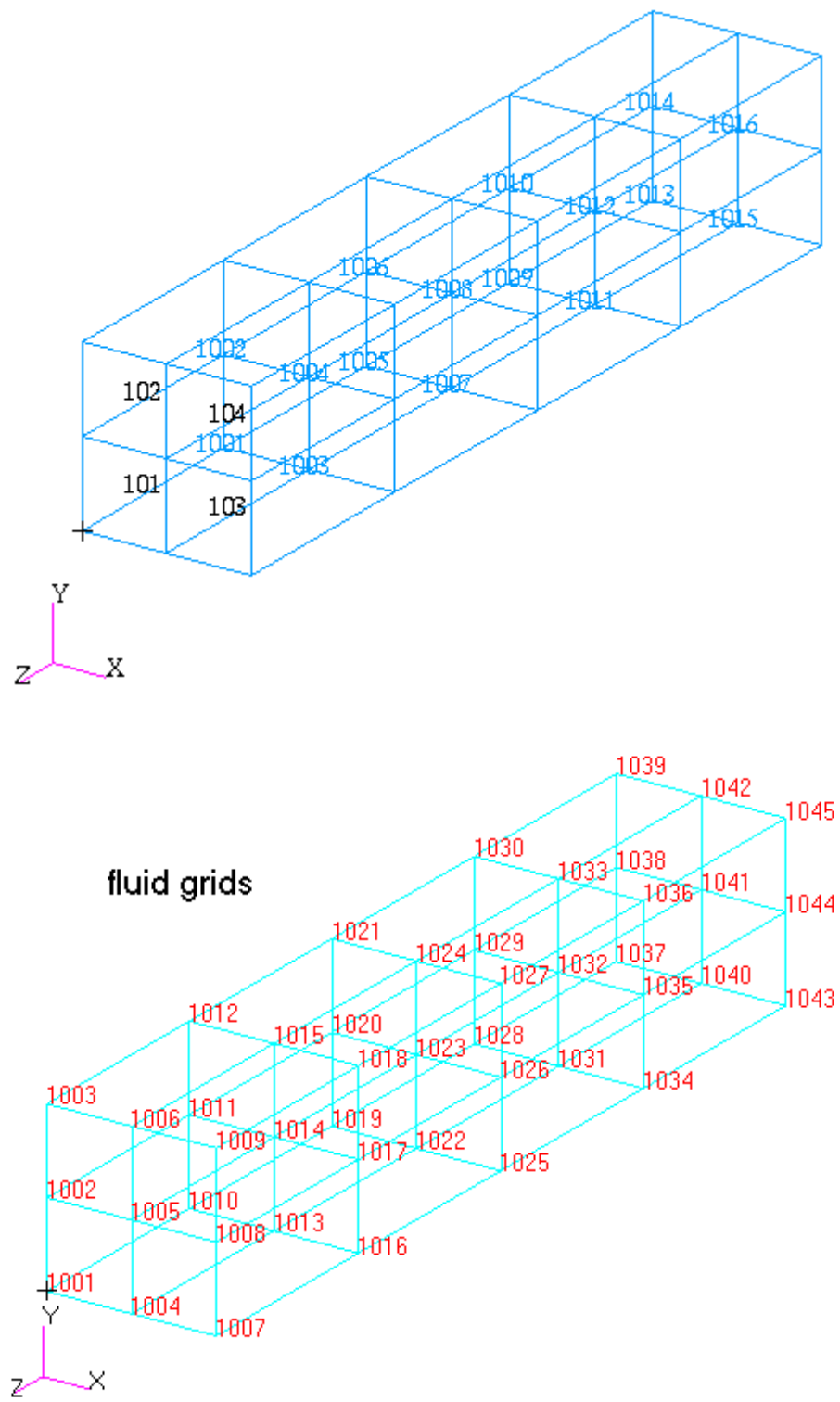
The problem is to find the modal participation factors in a damped frequency response analysis of a 20 in. x 20 in. x 100 in. air volume that has a 20 in. x 20 in. x 0.2 in. thick aluminum plate at one end loaded with a sinusoidal force at its center.

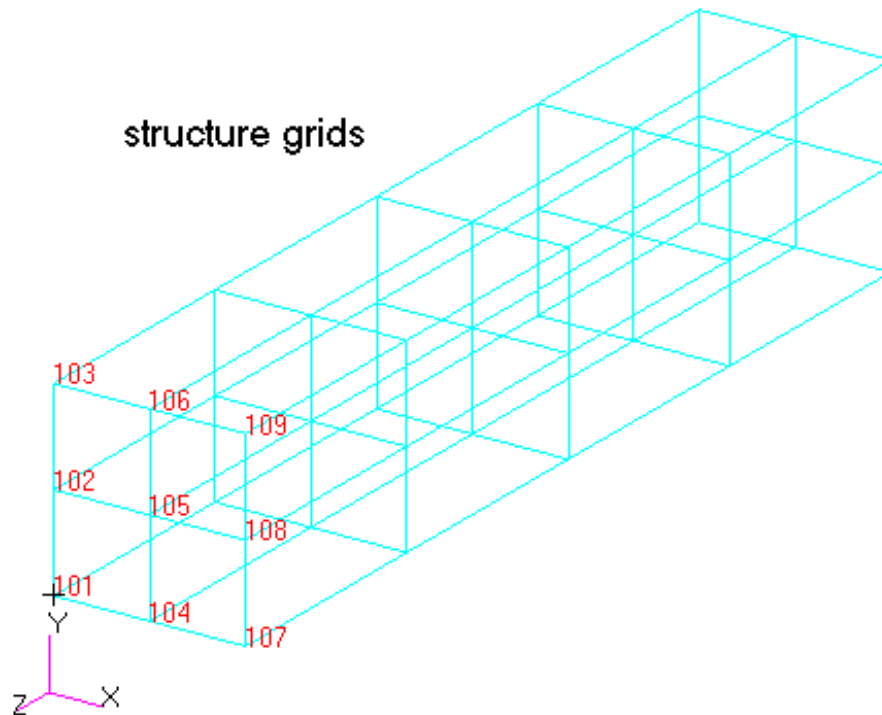


The following information will be found:

- fluid participation factors - these are the effects of the fluid mode on fluid grid pressures
- structural participation factors - these are the effect of the structural modes on fluid grid pressures
- panel participation factors - these are the effect of a group of structural grids on fluid grid pressures
- panel grid participation factors - these are the individual effects of each panel grid on fluid grid pressures

The model grid and elements are shown:





The modal frequency analysis uses five structural modes and 30 fluid modes. The case control commands to select fluid, structural, panel, and panel grid modal participation factors are:

```
flspout fluidmp=30 structmp=5 gridmp=all
```

Panel participation factors are automatically calculated when a STRUCTMP parameter is present for all the panels defined in the bulk data. This model has a panel defined with the following bulk data entries:

```
$
panel, leftside, 1
set1, 1, 101, 102, 103, 104, 105, 106
$
```

which are the grids that surround structural elements 101 and 102. The GRIDMP=all parameter on the FLSPOUT Case Control command requests participation factors for all of these panel grids.

A portion of the model is shown:

```

SOL      111
CEND
TITLE = Modal frequency response - fluid participation factors
DISP(sort2) = ALL
dload=10
spc=1
freq=15
method(structure)=98
method(fluid)=99
flstcnt prefdb=2.9e-9
flspout fluidmp=30 structmp=5 gridmp=all
BEGIN BULK
acmodlident
PARAM,POST,-1
PARAM    AUTOSPC YES
$
rload1,10,100,,,100100
darea,100,105,3,15.0
tabled1,100100
      0.      1.0      1000.      1.0      endt
freq1,15,10.,1.0,99
eigr1,98,,,5
eigr1,99,,,30
$
panel,leftside,1
set1,1,101,102,103,104,105,106
$
$ Structural model
$
$
PSHELL  1      1      .2      1      1
MAT1,1,1.+7,,,3,2.54-4, 0.01
CQUAD4  101      1      101      102      105      104      0.      0.
CQUAD4  102      1      102      103      106      105      0.      0.
CQUAD4  103      1      104      105      108      107      0.      0.
CQUAD4  104      1      105      106      109      108      0.      0.
GRID    101      0.      0.      0.
GRID    102      0.      10.      0.
GRID    103      0.      20.      0.
GRID    104      10.      0.      0.
GRID    105      10.      10.      0.
GRID    106      10.      20.      0.
GRID    107      20.      0.      0.
GRID    108      20.      10.      0.
GRID    109      20.      20.      0.
$
SPC      1      109      3      0.0
SPC      1      103      3      0.0
SPC      1      101      123      0.0
SPC      1      107      23      0.0
$
$ Fluid model

```

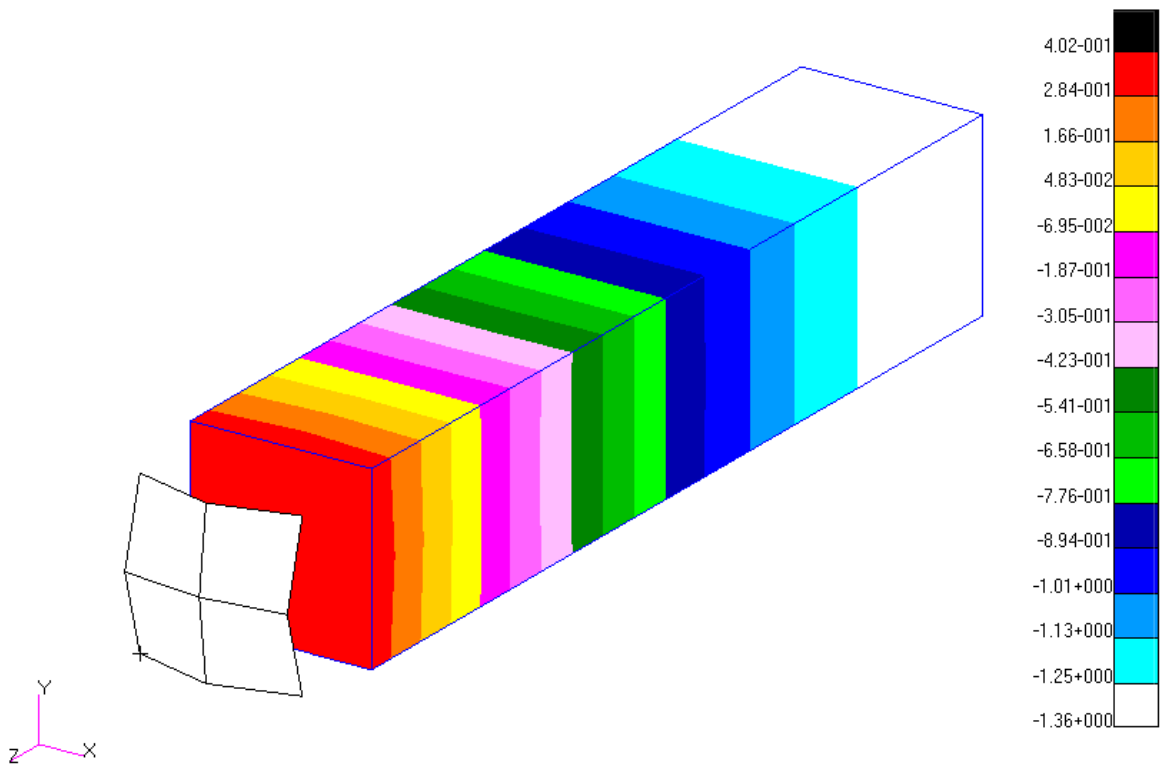
```

$
PSOLID 1001 1001
MAT10,1001,,1.21e-7,1.3e4,.02
CHEXA 1001 1001 1001 1002 1005 1004 1010 1011 + A
+ A 1014 1013
.
.
.
GRID 1044 20. 10. -100. -1
GRID 1045 20. 20. -100. -1
$
ENDDATA

```

Participation factors are available at all of the frequencies request, in this case 10 through 110 by 1. For illustrational purposes only the 39 hz results will be shown. This frequency is close to the first structural natural frequency.

A plot of the 39 hz response is shown below which is a composite of two pictures; a deformation plot of the structure and a pressure contour plot on the fluid elements, slightly offset from the structure for visual purposes:





The fluid modal participation factors for 39 hz are partially shown below:

MODAL PARTICIPATION FACTORS FOR SUBCASE				1 (DLOAD ID = 10)		
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = FLUID						
*** MODE PARTICIPATION FACTOR (MPF) ***				FLUID	MODE	FLUID
REAL	IMAGINARY	MAGNITUDE	PHASE	POINT	NUMBER	NATURAL FREQ. (HZ)
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1042	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1008	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1002	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1005	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1039	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1003	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1038	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1009	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1043	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1045	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1004	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1040	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1007	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1037	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1041	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1006	2	6.66808E+01
7.8079E-01	-6.6175E-01	1.0235E+00	319.72	1001	2	6.66808E+01
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1044	2	6.66808E+01
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1010	1	6.30946E-06
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1038	1	6.30946E-06
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1011	1	6.30946E-06
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1009	1	6.30946E-06
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1025	1	6.30946E-06
.						
.						
.						
-5.9512E-04	4.9336E-04	7.7303E-04	140.34	1024	29	7.71939E+02
-4.6513E-04	3.8560E-04	6.0418E-04	140.34	1004	28	7.53632E+02
-4.6513E-04	3.8560E-04	6.0418E-04	140.34	1044	28	7.53632E+02
4.6513E-04	-3.8560E-04	6.0418E-04	320.34	1008	28	7.53632E+02
4.6513E-04	-3.8560E-04	6.0418E-04	320.34	1002	28	7.53632E+02
-4.6513E-04	3.8560E-04	6.0418E-04	140.34	1038	28	7.53632E+02
-4.6513E-04	3.8560E-04	6.0418E-04	140.34	1006	28	7.53632E+02
4.6513E-04	-3.8560E-04	6.0418E-04	320.34	1042	28	7.53632E+02
4.6513E-04	-3.8560E-04	6.0418E-04	320.34	1040	28	7.53632E+02

The factors are ordered by magnitude. For the fluid participation factors, the ARF value on the FLSPOUT Case Control command is used to determine which factors will be printed. The default is ARF=0.95. This means only values above 95% of the highest value for each mode are printed. 328 factors were printed for 39 hz above, and also a similar number for each other 100 frequencies, totaling approximately 33,000 lines.

For the fluid modes, since participation factors are the modal solutions times the mode shapes, and since the first fluid mode is a rigid body mode, all of the participation factors for the first mode have the same value and thus are unaffected by ARF so they are all printed.

Also for this example, because of the simple shape of the model and the symmetry of the loading and constraints, various parts of the fluid model tend to have the same eigenvector value, so more than usual are printed.

If only the first maximum value is wanted from each of the 30 fluid modes, use ARF = 0.999:

*** MODE PARTICIPATION FACTOR (MPF) ***				FLUID	MODE	FLUID
REAL	IMAGINARY	MAGNITUDE	PHASE	POINT	NUMBER	NATURAL FREQ. (HZ)
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1042	2	6.66808E+01
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1010	1	6.30946E-06
1.6373E-01	-1.3631E-01	2.1304E-01	320.22	1008	3	1.43346E+02
-9.1451E-02	7.5979E-02	1.1890E-01	140.28	1045	4	2.32942E+02
3.8657E-02	-3.2104E-02	5.0249E-02	320.29	1024	5	2.86691E+02
7.1325E-03	-5.9130E-03	9.2648E-03	320.34	1045	27	7.53632E+02
-4.3311E-03	3.5905E-03	5.6258E-03	140.34	1032	30	7.71939E+02
2.5097E-03	-2.0806E-03	3.2600E-03	320.34	1019	25	7.30922E+02
2.4613E-03	-2.0404E-03	3.1971E-03	320.34	1004	26	7.30922E+02
-2.0102E-03	1.6665E-03	2.6112E-03	140.34	1041	23	7.19823E+02
-1.8705E-03	1.5507E-03	2.4297E-03	140.34	1008	24	7.19823E+02
-1.4622E-03	1.2122E-03	1.8994E-03	140.34	1007	22	7.16728E+02
-5.9823E-04	4.9595E-04	7.7708E-04	140.34	1024	21	7.16728E+02
-5.9512E-04	4.9336E-04	7.7303E-04	140.34	1011	29	7.71939E+02
-4.6513E-04	3.8560E-04	6.0418E-04	140.34	1004	28	7.53632E+02

Only 15 fluid mode values are shown because the other 15 have no values above the FEPS=1.0e-11 cutoff on the FLSPOUT Case Control command.

To list only fluid grid 1041 fluid participation values, use GRIDFMP:

*** MODE PARTICIPATION FACTOR (MPF) ***				FLUID	MODE	FLUID
REAL	IMAGINARY	MAGNITUDE	PHASE	POINT	NUMBER	NATURAL FREQ. (HZ)
-7.8079E-01	6.6175E-01	1.0235E+00	139.72	1041	2	6.66808E+01
-6.9499E-01	5.5365E-01	8.8856E-01	141.46	1041	1	6.30946E-06
1.6373E-01	-1.3631E-01	2.1304E-01	320.22	1041	3	1.43346E+02
-9.1451E-02	7.5979E-02	1.1890E-01	140.28	1041	4	2.32942E+02
3.8657E-02	-3.2104E-02	5.0249E-02	320.29	1041	5	2.86691E+02
-7.1325E-03	5.9130E-03	9.2648E-03	140.34	1041	27	7.53632E+02
4.3311E-03	-3.5905E-03	5.6258E-03	320.34	1041	30	7.71939E+02
2.5097E-03	-2.0806E-03	3.2600E-03	320.34	1041	25	7.30922E+02
2.4138E-03	-2.0011E-03	3.1354E-03	320.34	1041	26	7.30922E+02
-2.0102E-03	1.6665E-03	2.6112E-03	140.34	1041	23	7.19823E+02
1.4622E-03	-1.2122E-03	1.8994E-03	320.34	1041	22	7.16728E+02

The participations above show a wide variation of fluid mode inputs into the fluid solution. This is because 39 hz is a structural natural frequency. The single structural mode is translated at the fluid/structure interface into an equivalent set of fluid modes.

The structural fluid participation factors should show just one structural mode contributing to the majority of the fluid response because 39 hz is close to a structural natural frequency:

MODAL PARTICIPATION FACTORS FOR SUBCASE 1 (DLOAD ID = 10)							
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = STRUCTURE							
*** MODE PARTICIPATION FACTOR (MPF) ***				FLUID	MODE	STRUCTURE	
REAL	IMAGINARY	MAGNITUDE	PHASE	POINT	NUMBER	NATURAL FREQ. (HZ)	
-1.3650E+00	1.1229E+00	1.7675E+00	140.56	1037	1	3.96034E+01	
-1.3650E+00	1.1229E+00	1.7675E+00	140.56	1043	1	3.96034E+01	
-1.3650E+00	1.1229E+00	1.7675E+00	140.56	1039	1	3.96034E+01	
-1.3650E+00	1.1229E+00	1.7675E+00	140.56	1045	1	3.96034E+01	
-1.3648E+00	1.1228E+00	1.7673E+00	140.56	1040	1	3.96034E+01	
-1.3648E+00	1.1228E+00	1.7673E+00	140.56	1044	1	3.96034E+01	
-1.3648E+00	1.1228E+00	1.7673E+00	140.56	1038	1	3.96034E+01	
-1.3648E+00	1.1228E+00	1.7673E+00	140.56	1042	1	3.96034E+01	
-1.3647E+00	1.1226E+00	1.7671E+00	140.56	1041	1	3.96034E+01	
-1.5697E-05	2.0986E-04	2.1045E-04	94.28	1041	5	9.82372E+01	
-1.5689E-05	2.0976E-04	2.1034E-04	94.28	1040	5	9.82372E+01	
-1.5689E-05	2.0976E-04	2.1034E-04	94.28	1042	5	9.82372E+01	
-1.5689E-05	2.0976E-04	2.1034E-04	94.28	1038	5	9.82372E+01	
-1.5689E-05	2.0976E-04	2.1034E-04	94.28	1044	5	9.82372E+01	
-1.5682E-05	2.0965E-04	2.1024E-04	94.28	1039	5	9.82372E+01	
-1.5682E-05	2.0965E-04	2.1024E-04	94.28	1043	5	9.82372E+01	
-1.5682E-05	2.0965E-04	2.1024E-04	94.28	1037	5	9.82372E+01	
-1.5682E-05	2.0965E-04	2.1024E-04	94.28	1045	5	9.82372E+01	

Note that the only other mode besides structural mode 1 is mode 5 which has insignificant values.

The response for fluid grid 1041 at 39 hz is 1.76 psi:

POINT-ID = 1041					
COMPLEX ACOUSTIC PRESSURE RESULTS					
(MAGNITUDE/PHASE)					
FREQUENCY	TYPE	P	P(RMS)	DB	DB(A)
3.900000E+01	S	1.767275E+00	1.249652E+00	1.756981E+02	1.405334E+02
		140.5534	140.5534	140.5534	140.5534

Note that this is the value of the single structural modal participation factor, and is also the sum of the fluid factors (after accounting for phase).

The panel is made of the six grids that bound the left side of the structure. The participations for this panel are shown:

MODAL PARTICIPATION FACTORS FOR SUBCASE 1 (DLOAD ID = 10)							
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = PANEL : LEFTSIDE							
*** MODE PARTICIPATION FACTOR (MPF) ***							
REAL	IMAGINARY	MAGNITUDE	PHASE	FLUID POINT	MODE NUMBER	STRUCTURE NATURAL FREQ. (HZ)	
-1.1672E+00	9.6021E-01	1.5114E+00	140.56	1045	1	3.96034E+01	
-1.1672E+00	9.6021E-01	1.5114E+00	140.56	1043	1	3.96034E+01	
-1.1672E+00	9.6017E-01	1.5114E+00	140.56	1037	1	3.96034E+01	
-1.1672E+00	9.6017E-01	1.5114E+00	140.56	1039	1	3.96034E+01	
-1.1672E+00	9.6014E-01	1.5113E+00	140.56	1044	1	3.96034E+01	
-1.1671E+00	9.6009E-01	1.5113E+00	140.56	1038	1	3.96034E+01	
-1.1670E+00	9.6004E-01	1.5112E+00	140.56	1042	1	3.96034E+01	
-1.1670E+00	9.6004E-01	1.5112E+00	140.56	1040	1	3.96034E+01	
-1.1670E+00	9.5997E-01	1.5111E+00	140.56	1041	1	3.96034E+01	
2.9833E-06	-5.1878E-05	5.1964E-05	273.29	1003	5	9.82372E+01	
2.9833E-06	-5.1878E-05	5.1964E-05	273.29	1001	5	9.82372E+01	

Note that panel "leftside" has participations equaling 1.511 psi of the total 1.766 psi for the fluid grid 1041 response. This panel factor is the sum of the grid factors in the panel. They are shown below for 39 hz. Only structural mode 1 is shown since it only is significant. Also only fluid grid 1041 is shown:

MODAL PARTICIPATION FACTORS FOR SUBCASE				1 (DLOAD ID =	10)	
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = PANEL :						
LEFTSIDE AT GRID 102						
*** MODE PARTICIPATION FACTOR (MPF) ***						
REAL	IMAGINARY	MAGNITUDE	PHASE	FLUID POINT	MODE NUMBER	STRUCTURE NATURAL FREQ. (HZ)
1.9775E-01	-1.6267E-01	2.5606E-01	320.56	1041	1	3.96034E+01
MODAL PARTICIPATION FACTORS FOR SUBCASE				1 (DLOAD ID =	10)	
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = PANEL :						
LEFTSIDE AT GRID 104						
*** MODE PARTICIPATION FACTOR (MPF) ***						
REAL	IMAGINARY	MAGNITUDE	PHASE	FLUID POINT	MODE NUMBER	STRUCTURE NATURAL FREQ. (HZ)
1.9775E-01	-1.6267E-01	2.5606E-01	320.56	1041	1	3.96034E+01
MODAL PARTICIPATION FACTORS FOR SUBCASE				1 (DLOAD ID =	10)	
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = PANEL :						
LEFTSIDE AT GRID 105						
*** MODE PARTICIPATION FACTOR (MPF) ***						
REAL	IMAGINARY	MAGNITUDE	PHASE	FLUID POINT	MODE NUMBER	STRUCTURE NATURAL FREQ. (HZ)
5.7371E-01	-4.7195E-01	7.4288E-01	320.56	1041	1	3.96034E+01
MODAL PARTICIPATION FACTORS FOR SUBCASE				1 (DLOAD ID =	10)	
DESCENDING ORDER SORT ON MAGN VALUE AT EXCITATION FREQUENCY OF 3.90000E+01 HZ FOR MPF TYPE = PANEL :						
LEFTSIDE AT GRID 106						
*** MODE PARTICIPATION FACTOR (MPF) ***						
REAL	IMAGINARY	MAGNITUDE	PHASE	FLUID POINT	MODE NUMBER	STRUCTURE NATURAL FREQ. (HZ)
1.9775E-01	-1.6267E-01	2.5606E-01	320.56	1041	1	3.96034E+01

When the real and imaginary terms are summed for each panel grid factor, and the magnitude calculated, it will equal the panel factor of 1.511 psi.

## 7.4 Acoustic XY Plots, Random, and Restarts

### XY Plots

The following new "yvtype"s have been added for the XYPLOT Case Control commands.

- FMPF (mode\_id/frequency\_id)
  - Fluid mode participation factors
- GMPF (mode\_id/frequency\_id, panel\_name, panel\_grid\_id)
  - Panel grid Mode participation factors
- LMPF
  - Load mode participation factors
- PMPF (mode\_id/frequency\_id, panel\_name)
  - Panel mode participation factors
- SMPF (mode\_id/frequency\_id)
  - Structural mode participation factors

Also, the following new "ptype"s have been added:

- MODE
  - for given fluid mode plot mode
  - participation vs excitation frequency
  - O2E="NO" on FLSPOUT Case Control command (default)
- FREQ
  - for given excitation frequency
  - plot mode participation vs natural frequency
  - O2E="YES" on FLSPOUT Case Control command
  - point plot only

See the following example:

## Random Analysis

A "AUTO" and "PSDF" XYPLOT random "ptype"s support the new acoustic modal participation "yvtype"s:

- FMPF
- SMPF
- PMPF
- LMPF
- GMPF

Root mean square random results may be a convenient way to calculate the effect of a mode on a fluid response across a frequency range. Weigh the effect of the frequencies on the RANDPS Bulk Data entry and use one of the above "ptype" in an PSDF XYPLOT command and a root mean square result will be calculated.

## Random, XY Plot, Restart Example

The base line and restart input files below represents a simple fluid-structure problem for the purpose of showing salient input.

### Model Description

The model consists of a CQUAD4 and four CELAS2 to represent structure, a CHEXA to represent the fluid, and a CHACAB structural element to represent an acoustic absorber. In the baseline Case Control, note the use of the FLSFSEL to select lower limits on the fluid and structure frequency range modes. Also note the use of FLSTCNT to set the reference pressure. Some fictitious random entries are input to demonstrate the XYPLOT features for structural and fluid grids. On large models of automobiles and aircraft, the modal solutions, especially on the structure, can take significant time and disk space. Also modal participation can, on large models, generate gigabytes of data. Thus no participation information is requested for the base line run, as participation results will be requested on the restart run. Since a data base is required, the MSC.Nastran base line submittal contains SCR=NO.

In the restart file, note the use of the ASSIGN and RESTART commands. Notice, also, that Case Control commands such as FLSFSEL, FLSTCNT, FREQ, SPC, METHOD(STRUCT) and METHOD(FLUID) are *copied* over from the base line run. To

change or omit these would cause the restart logic to attempt to do the modal analysis over again. To calculate mode participation information, the FLSPOUT command is added to case control along with three set definitions directly above.

To define random data appropriate to modal participation, a different RANDOM command is provided. Also, a different set of XYPLOT commands are provided that are to be used for plotting of participation results.

For the restart run, the original bulk data entries are removed and some PANEL and SET1 entries are added to define panels for the participation run. Also, new RANDPS and RANDT1 entries are added for the participation analysis. Note that no TABRND1 is included because the original one from the base line run is used. Since none of the bulk data from the base line run is changed or deleted, the restart file does not contain any (/) bulk data entries.

---

**Note:** DISP(PLOT)=ALL is currently required in the restart run.

---

## Input File For Base Line Run

```
SOL 111 $ MODAL FREQUENCY RESPONSE
CEND
TITLE= FLUID-STRUCTURE INTERACTION
$
FLSFSEL LFREQFL=1.0,LFREQ=2.0
FLSTCNT PREFDB=20.-6
$
OLOAD=ALL
DISP ( SORT2 , PHASE ) =ALL
STRESS ( SORT2 , PHASE ) =ALL
RANDOM 10
$
SPC =1313
FREQ=200
METHOD ( STRUCT ) =30
METHOD ( FLUID ) =20
$
SUBCASE 100
DLOAD=100
$
SUBCASE 200
DLOAD=200
$
OUTPUT ( XYPLOT )
XGRID=YES
YGRID=YES
XTITLE=frequency #14
YTITLE=psd grid 61
XYPLOT,XYPEAK DISP PSDF /61(T3)
```



```

XTITLE=frequency #15
YTITLE=auto grid 61
XYPLOT,XYPEAK DISP AUTO /61(T3)
XTITLE=frequency #16
YTITLE=psd grid 224
XYPLOT,XYPEAK DISP PSDF /224(T1)
XTITLE=frequency #17
YTITLE=auto grid 224
XYPLOT,XYPEAK DISP AUTO /224(T1)
$
BEGIN BULK
$ PRODUCE OUTPUT FORTRAN FILES
PARAM,POST,-1
$ DYNAMIC LOADING
$ ON STRUCTURE
DLOAD 100 1. 1. 1002 1. 1001
DLOAD 200 1. 1. 2002 1. 2001
RLOAD1 1001 19 1001
RLOAD1 2001 29 1001
DAREA 19 41 3 125.
DAREA 19 45 3 125.
DAREA 19 61 3 125.
DAREA 19 65 3 125.
DAREA 29 41 3 125.
DAREA 29 45 3 125.
DAREA 29 61 3 125.
DAREA 29 65 3 125.
$ ON FLUID
ACSRCE 1002 101 1001 1. 1.
ACSRCE 2002 102 1001 1. 1.
SLOAD 101 120 125.
SLOAD 101 124 125.
SLOAD 101 220 125.
SLOAD 101 224 125.
SLOAD 102 120 250.
SLOAD 102 124 250.
SLOAD 102 220 250.
SLOAD 102 224 250.
TABLED4 1001 0.0 1.0 0.0 1.E6
0.0 0.0 39.478420.0 ENDT
$ FREQUENCY RANGE
FREQ1 200 2.5 2.5 100
$ DAMPING
PARAM G 0.05
PARAM GFL 0.05
$ METHOD CARD FOR EIGEN VALUE
EIGRL 20 6
EIGRL 30 4
$THE STRUCTURAL POINTS
GRID 41 0.0 0.0 0.0
GRID 45 1.1 0.0 0.1
GRID 61 0.0 0.20 0.0
GRID 65 1.0 0.20 0.0
$THE STRUCTURAL POINTS FOR ABSORBER

```

```

GRID      261          0.0      0.2      0.001          12456
GRID      265          1.0      0.2      .001          12456
GRID      141          0.0      0.0      0.001          12456
GRID      145          1.1      0.0      .1001         12456
$THE STRUCTURAL ELEMENTS
CQUAD4  100      4444      41      45      65      61
CELAS2  1007      .0625      61      3
CELAS2  1008      .0625      65      3
CELAS2  1009      .0625      41      3
CELAS2  1010      .0625      45      3
$ STRUCTURAL PROPERTIES
PSHELL  4444      77      .05      77      1.0
MAT1     77      100.      .333      1.000
$
$ FLUID POINTS
GRID      200          0.0      0.2      0.001      -1
GRID      204          1.0      0.2      .001      -1
GRID      220          0.0      0.2      1.0      -1
GRID      224          1.0      0.2      1.0      -1
GRID      100          0.0      0.0      0.001      -1
GRID      104          1.1      0.0      .1001      -1
GRID      120          0.0      0.0      1.0      -1
GRID      124          1.0      0.0      1.0      -1
$ FLUID ELEMENTS
CHEXA    1000      15      100      104      124      120      200      204
          224      220
$ FLUID PROPERTIES
PSOLID   15      25
MAT10    25      1.      0.1
$
$ SPC-STRUCTURAL AND FLUID
SPC1     1313      1246      41      THRU      61
SPC1     1313      5      41      61      65      45
SPC1     1313      146      65
SPC1           1313      1      200
$ DEFINE ACOUSTIC ABSORBER
CHACAB   9      9      41      45      65      61      141      145
+         265      261
PACABS   9      YES      11      22      33      1.      1000.
TABLED1  11
+         2.      .15      50.      .15      100.      .15      150.      .15
+         200.      .15      250.      .15      300.      .15      ENDT
TABLED1  22
+         2.      -40.      50.      14.      100.      30.      150.      46.
+         200.      62.      250.      78.      300.      94.      ENDT
TABLED1  33
+         2.      1.      50.      1.      100.      1.      150.      1.
+         200.      1.      250.      1.      300.      1.      ENDT
$
$ RANDOM INPUT
RANDPS   10      100      100      4.      1
RANDPS   10      200      200      8.      1
RANDPS   10      100      200      2.      1
TABRND1  1

```

```

          0.          100.          300.          100.          ENDT
RANDT1  10          4          0.          .004
$
ENDDATA

```

```

ASSIGN MASTER='1_a.MASTER'
RESTART VERSION=1 KEEP
SOL 111 $ MODAL FREQUENCY RESPONSE
CEND
SUBTITLE= FLUID-STRUCTURE INTERACTION
$
SET 29 = BNDY
SET 224 = 224
FLSPOUT FLUIDMP=ALL,GRIDFMP=224,SEPS=0.,ARS=0.,
        STRUCTMP=ALL,PANELMP=29,GRIDMP=ALL,
        OUTFMP=ALL,OUTSMP=ALL,PSORT=(ABSOLUTE,DESCENDING)
FLSFSEL LFREQFL=1.0,LFREQ=2.0
FLSTCNT PREFDB=20.-6, SKINOUT=ALL
$
RANDOM=50
$
SPC =1313
FREQ=200
METHOD(STRUCT)=30
METHOD(FLUID)=20
DISP(PLOT)=ALL
$
SUBCASE 100
DLOAD=100
$
SUBCASE 200
DLOAD=200
$
OUTPUT(XYPLOT)
XGRID=YES
YGRID=YES
$
$-1 plt 2nd sc
XTITLE=EXCITATION FREQUENCY FROM 2.5 to 250 hertz sc 200 #1
YTITLE=fluid mode pf at fluid point 224 for mode 2
XYPLOT,XYPEAK FMPF(2) MODE 200 /224
$-2,3 plt 1st & 2nd sc
XTITLE=EXCITATION FREQUENCY FROM 2.5 to 250 hertz default sc 100 200 #2 and
#3
YTITLE= load pf at fluid point 224
XYPLOT,XYPEAK LMPF MODE /224
$-4 1st sc
XTITLE= natural modes from .4 to 2 hertz #4
YTITLE= panel mode pf at fluid point 224 for mode 9 sc 100
XYPLOT,XYPEAK PMPF(9,BNDY) MODE 100 /224
$-5 1st sc
XTITLE= natural modes from .4 to 2 hertz sc 100 #5
YTITLE=structure mode pf at fluid point 224 for mode 3

```

```

XYPLOT,XYPEAK SMPF(3) MODE 100 /224
$-6,7 1st & 2nd  sc
XTITLE= natural modes from .4 to 2 hertz default sc 100 200 #6 and #7
YTITLE=grid panel mode pf at fluid point 224 for mode 9 grid 61
XYPLOT,XYPEAK GMPF(9,BNDY,61) MODE  /224
$-8,9 1st & 2nd  sc
XTITLE= natural modes from .4 to 2 hertz try 100,200 #8 and #9
YTITLE=grid panel mode pf at fluid point 224 for mode 9 grid 61
XYPLOT,XYPEAK GMPF(9,BNDY,61) MODE  100,200 /224
$
$-10
XTITLE=frequency #10
YTITLE=psd mode participation for grid 224
XYPLOT,XYPEAK FMPF(3) PSDF  /224
$-11
XTITLE=frequency #11
YTITLE=auto mode participation for grid 224
XYPLOT,XYPEAK FMPF(3) AUTO  /224
$-12
XTITLE=frequency #12
YTITLE=psd grid mode participation for grid 224 firstone at grid 61 first
decend mode
XYPLOT,XYPEAK GMPF(9,BNDY,61) PSDF  /224
$-13
XTITLE=frequency #13
YTITLE=auto grid mode participation for grid 224 firstone at grid 61 first
decend mode
XYPLOT,XYPEAK GMPF(9,BNDY,61) AUTO  /224
$ ----- expect 2 plots(4th group)
$
BEGIN BULK
$ DEFINE STRUCTURAL PANELS
PANEL          BNDY          777
SET1    777      SKIN
$ NEW RANDOM INPUT FOR PARTICIPATION
RANDPS  50      100      100      2.              1
RANDPS  50      200      200      5.              1
RANDPS  50      100      200      1.              1
RANDT1  50      6        0.        .009
$
ENDDATA

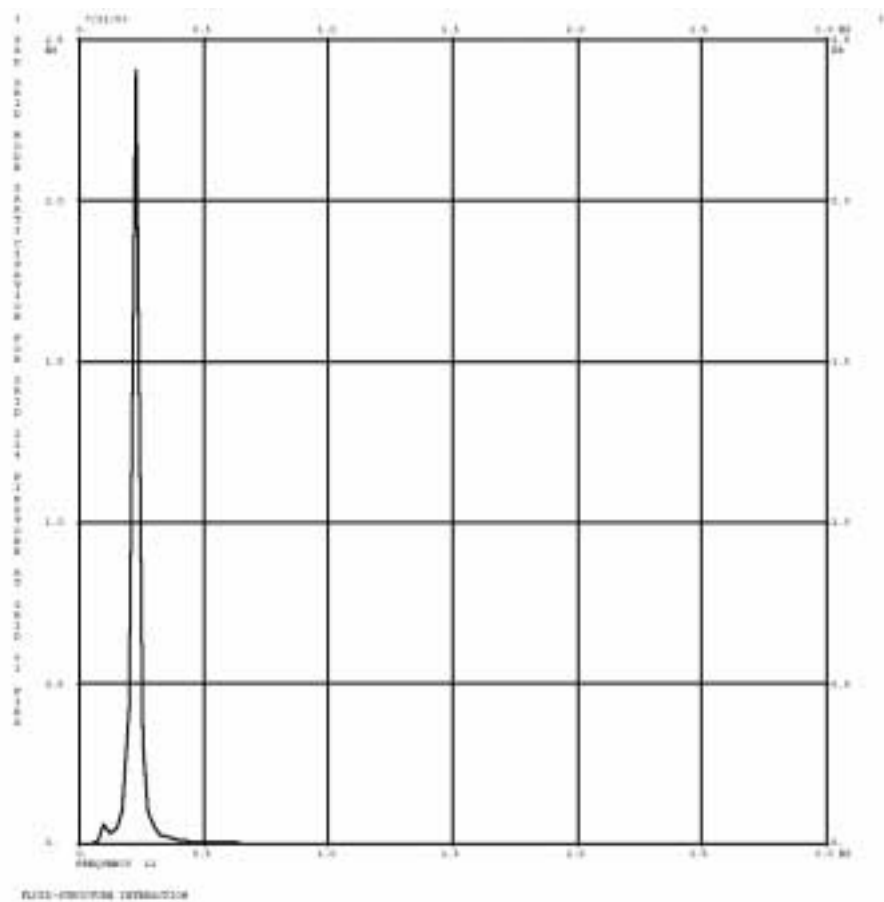
```

The following XYPLOT commands create the plot below:

```

XTITLE=FREQUENCY #12
YTITLE=PSD GRID MODE PARTICIPATION FOR GRID 224 FIRSTONE AT GRID 61 FIRST DECEND
XYPLOT,XYPEAK GMPF(9,BNDY,61) PSDF  /224

```



This is the PSD of structural modal participation of grid point 61, mode 9, on fluid grid 224. In order to get structural grid participation factors, the structural grid must be defined on a panel and called with GMPF in the XYPLOT command. Panels are defined with PANEL and SET1 Bulk Data entries. In this case, the "SKIN" option on the SET1 bulk data entry selects all of the structural interface grids to be in panel "BNDY":

```
PANEL    BNDY    777
SET1     777     SKIN
```

For PSD curves, the root-mean-square value across frequencies is calculated and printed in the .f06 file:

XY - OUTPUT SUMMARY ( AUTO OR PSD F )											
PLOT	CURVE	FRAME	CURVE ID./		RMS	NO. POSITIVE	XMIN FOR	XMAX FOR	YMIN FOR	X FOR	YMAX FOR
TYPE	TYPE	NO.	PANEL	: GRID ID	VALUE	CROSSINGS	ALL DATA	ALL DATA	ALL DATA	YMIN	ALL DATA
FREQ	NO.	0	BNDY	: 61							
PSDF	GRID-P	3	224( 3)		3.048659E+04	2.654195E+01	2.500E+00	2.525E+02	2.185E+03	5.000E+00	2.411E+08
											2.250E+01

This RMS value, 3.048659E+04, can be used as a measure of the combined relative effect of mode 9 across all the frequencies.

## 7.5 Acoustic Source Change

### Simple Acoustic Source (ACSRCE) Equation

The complex source strength is defined in Reference 2. in Appendix B as

$$Qe^{i\omega t} = \int_s \dot{u} \cdot \dot{n} dS \quad \text{Eq. 7-17}$$

Also, similar to the Maxwell-Betti reciprocity law in structures, they define a principle of acoustic reciprocity which states that in an unchanging environment if the locations of a small receiver and small source are interchanged, the received signal will remain the same.

With these definitions, they arrive at the expression for power  $P(f)$  in terms of complex source strength, which for a pulsating sphere is a real value  $Q$ . The resulting expression as used in MSC.Nastran is:

$$Q = \frac{1}{2\pi f} \sqrt{\frac{8\pi c P(f)}{\rho}} \quad \text{Eq. 7-18}$$

Acoustic power is applied by referencing the ACSRCE Bulk Data entry from a DLOAD Case Control command or Bulk Data entry. The ACSRCE entry specifies the power from a TABLEDi Bulk Data entry and also points to DAREA Bulk Data entries to specify the degree of freedom to be loaded as well as a scale factor. In Eq. 7-18 it should be noted that source strength is proportional to the square root of power. This nonlinear relation is a source of inconsistencies in MSC.Nastran over different versions.

In MSC.Nastran versions prior to 2001, the power was translated to source strength from the values on the TABLEDi entries. This source strength was then distributed linearly to the degrees of freedom using the factors on the DAREA entries. This appeared to cause an inconsistency in that if the power was scaled using the DAREA factor, a different power was obtained than if the power was scaled in the TABLEDi because of the nonlinear relationship between source strength and power.

In MSC.Nastran 2001, this was changed so that if the power was scaled on the DAREA factor or on the TABLEDi value the same energy was applied.

Unfortunately, this created an inconsistency with superelements and ACMS where the ability to know if the source strength came from an ACSRCE (power) or a RLOADi or TLOADi (source strength) is lost, and thus incorrect loading occurred with the ACSRCE. The result was that answers from an ACMS or superelement run did not match a residual-only run when an ACSRCE was present.

In MSC.Nastran 2004, the ACSRCE logic has been returned to pre MSC.Nastran 2001, which works with superelements in ACMS.

Acoustic engineers need to know that when using the ACSRCE Bulk Data entry that if power is to be scaled, do it within the TABLEDi. If source strength is to be scaled, do it on DAREA. This may be confusing to the beginner, but actually follows the definition of power point sources given previously.

A way to avoid this confusion is to change your power to source strength using [Eq. 7-18](#), and apply it as needed on the RLOADi and TLOADi Bulk Data entries.

## 7.6 Efficiency Enhancement in Virtual Mass

### Introduction

Efficiency features formerly provided with the SSSALTER of Reference 22 are now built into MSC.Nastran 2004. Upward Compatibility Issues describes the differences in the new method and older methods that may be built into old input files. These techniques apply only to modal analysis, not direct analysis. No changes have been made to improve the efficiency of the basic Virtual Mass (VM) modules.

### Input

Remove any param, VMOPT entreis that may be present in existing input files, and add PARAM,VMOPT,2 for the new efficient method. A sample test file vm01.dat is included on the delivery CD.

### Theory and Output

The modes of the structure without the fluid are computed first ("dry" modes). The fluid effects are added in the modal basis during the residual flexibility computation to produce the "wet" modes. Both eigenvalue tables are printed, allowing comparison of the dry and wet modes. The wet modes are used in modal dynamic analysis. The cost savings result from the dense Virtual Mass matrix being kept out when computing modes in the physical basis. Its presence can increase memory and computation times by an order of magnitude. The VM is added only in the smaller generalized basis used in Residual Flexibility Computations. The approximations introduced by this approach are generally small due to the homogeneous nature of the fluid.

### Upward Compatibility Issues

Prior to MSC.Nastran 2001 the vm\* series of alters provided much of the efficiency described here. They were dropped for MSC.Nastran 2001, and a different method suggested that provided some of the efficiency by use of built-in features, without the need for alters. These features include q-set DOFs on special variables (QSETi, SPOINT entries). These special features still work the same, so old input files may be used unchanged in MSC.Nastran 2004. The alternative is to remove the special features, and use param,vmopt,2 instead. This newer method should be slightly faster for fluids with a free surface, and much faster for a structure enclosed in an infinite fluid.



## 7.7 Modifications to the Computation of Fluid and Structural Modes in SOL 103

In SOL 103, with fluid-structure models, the fluid modes and structural modes are now computed separately as is currently done in SOLs 110, 111, 112 and 200. This was done in order to enable more efficiently restarts from SOL 103 to SOLs 110, 111, 112 and 200. This also means that SOL 103 can now distinguish between METHOD(FLUID) and METHOD(STRUCTURE) Case Control command specifications.

When both METHOD(STRUCTURE) and METHOD(FLUID) are used in the same subcase, eigenvalues are listed from 1 to m structure modes in one table and 1 to n fluid modes in a separate table. However, the eigenvector output and all other data recovery is in one table with the structure modes first from 1 to m, and the fluid modes next from m+1 to n+1.

Modal participation factor output is now computed for the fluid superelement (PARAM,FLUIDSE).



CHAPTER

8

# Nonlinear Analysis

- MSC.Nastran Implicit Nonlinear -- SOL 600
- Prerelease of the General Nonlinear Solution Sequence SOL 400
- Performance Improvements in SOL 129
- Buckling Analysis

## 8.1 MSC.Nastran Implicit Nonlinear -- SOL 600

### Introduction

MSC.Nastran Implicit Nonlinear Analysis, also known as SOL 600, is a new capability within MSC.Nastran. Its primary purpose is to allow MSC.Nastran users to execute complex nonlinear problems with minimum changes to MSC.Nastran input used for other linear and nonlinear analyses. In simple nonlinear large deformation or plasticity analyses, only the SOL command needs to be modified. For more complex nonlinear analyses, only a few new bulk data entries and/or parameters are needed to describe the new features. The new entries have been formulated in such a way that in most cases they may be added to an existing MSC.Nastran file without the use of a GUI (exceptions are rigid contact described by nurbs which normally require a GUI). Of course, use of a GUI will usually make data input much easier and the user is encouraged to examine the capabilities of MSC.Patran to incorporate SOL 600 features into his model.

Through SOL 600, the MSC.Nastran user for the first time has access to the following features:

- 2D & 3D Contact
  - Flexible-Flexible surfaces
  - Flexible-Rigid surfaces
  - Enforced displacement or velocity of rigid surfaces
- Large strain for all element types
- The nonlinear options may be used in either static or dynamic analyses
  - Natural frequency calculation after any time or load step
  - Buckling calculations after any time or load step
- Advanced nonlinear material models some of which are listed below:
  - Ogden rubber formulation
  - Arruda-Boyce rubber formulation
  - Gent rubber formulation
  - Formulations to describe foam materials
  - Additional plasticity options
  - Plasticity of orthotropic and anisotropic materials
  - Strain-rate dependent materials

- Crack growth simulation
  - Analysis of porous materials
  - Additional soil models
  - Temperature-dependent plasticity, rubber, etc. descriptions
  - Visco-elastic materials
  - Thermo Visco-elastic materials
  - Viscoplasticity for creep analysis
  - Gasket materials
- Failure of portions of the structure
  - Composite materials modeled using solid elements
  - Parallel execution using domain decomposition

## What is SOL 600 and How Does it Work

For many years, MSC.Nastran has incorporated the most important nonlinear features of structural and heat transfer analyses through solutions 106, 129, 153 and 159. The basic features include large deformation, plasticity and a few additional nonlinear material descriptions such as soils. Relative to some other specialized nonlinear programs there are still many missing features. A few years back, MSC Software purchased a world-class nonlinear program named Marc which contains nearly all the advanced features of nonlinear analysis. There have been many requests from MSC.Nastran customers to incorporate many of these nonlinear features into MSC.Nastran. The most common request has been 3D contact of deformable bodies and large strain.

Two development projects were initiated to address these user desires. One project, which is not completely ready for the initial MSC.Nastran 2004 release, adds MSC.Marc's 3D contact routines directly inside MSC.Nastran. In addition, large rotation of rigid bodies and additional types of rigid mechanisms are being added. This project is known as SOL 400 and is described in "[Prerelease of the General Nonlinear Solution Sequence SOL 400](#)" on page 362 of this guide.

The other project allows the MSC.Nastran user access to nearly all of MSC.Marc's advanced nonlinear technology for structural analysis while still maintaining the MSC.Nastran environment (or look and feel) both from an input and output perspective. This project is known as SOL 600. As described in the introduction, many additional nonlinear features become available through this project that would not be

possible to incorporate directly inside MSC.Nastran in a reasonable length of time. Most of the features not incorporated in the current release are planned for a future release.

MSC.Nastran Implicit Nonlinear works as follows.

1. Inside MSC.Nastran, a translator between MSC.Nastran and MSC.Marc is automatically activated when the SOL 600 Executive Control statement is encountered. MSC.Nastran Executive Control, Case Control and Bulk Data are all examined and translated to the proper MSC.Marc formats whenever possible. When such translation is not possible, error messages or Translator Warning messages are issued. The translation is performed in the IFP portion of MSC.Nastran and an actual MSC.Marc input file is produced and saved.
2. The MSC.Marc input file may then be executed directly inside MSC.Nastran if requested by the user (it can also be saved for editing and execution of MSC.Marc in a separate run if so desired). The default is to execute MSC.Marc.
3. MSC.Marc's "unformatted" or "binary" results file known as the t16 file may optionally be converted to MSC.Nastran op2, xdb, and/or punch files. The results can even be placed in the f06 file in the same "format" as other MSC.Nastran "printed output" if so desired.

### **Description of the new SOL 600 Executive Control Command:**

The most important options for the new SOL 600 Executive Control statement are shown below:

SOL 600, ID PATH= COPYR= OUTF=op2,xdb,pch,f06 STOP=

Example: SOL 600,106 PATH=/progs/marc2003/tools OUTF=op2,f06,xdb

**SOL 600,ID** is a new Executive Control statement like SOL. The difference between other solution sequences and SOL 600 is that the computations (element matrix formulations, matrix decomposition, etc.) will be performed by a companion program (in this case MSC.Marc) rather than directly within MSC.Nastran. Inputs and outputs as much as possible will be the same as (or similar to) the familiar MSC.Nastran inputs and outputs. The SOL 600 statement should normally be used only for nonlinear analysis, but it may also be used for certain classes of linear static or dynamic analyses. The recommended form of this statement is shown with the options provided above. If entered with "SOL 600,ID" only, it acts just like SOL except a MSC.Marc input data

file “jid.marc.dat” will be generated. (“jid” is the name of the MSC.Nastran input file without the extension). For example, if the MSC.Nastran input file is named abcd.dat, (or abcd.bdf) then jid is abcd).

**ID** (required) designates the familiar MSC.Nastran solution type as designated by integers or names in the same manner as used with a standard SOL statement. Examples are 106, 129, NLSTATIC, NLTRAN, etc. The following solutions are not available: 107, 110, 114, 115, 116, 118, 144, 145, 146, 153, 159, 190, 200, and 400 (and their equivalent names). Solutions specified in table 3-3 of the standard SOL statement may be used except for 7, 10, 14-16 and their equivalent names. In future releases, it is anticipated that the heat transfer solutions will be added.

**PATH** is an optional item which determines the location of the version of MSC.Marc to be executed. If PATH is omitted, a special version of MSC.Marc included with MSC.Nastran will be used if it can be located. In this case, the run script for MSC.Marc (run\_marc or run\_marc.bat) will be executed to be in the directory /MSC\_BASE/marc/MSC\_ARCH/marc2003/tools. MSC\_BASE and MSC\_ARCH are environmental variable set when MSC.Nastran first starts execution that define the base installation directory for MSC.Nastran and the machine type such as aix, linux, hpux, alpha, irix, solaris, i386, etc. If for some reason MSC\_BASE and/or MSC\_ARCH cannot be determined, the commands to spawn MSC.Marc will fail and the user must re-run MSC.Nastran with one of the PATH options set or the NASM\_PATH environmental option set to the desired location of MSC.Marc’s tools directory.

If **PATH=1** MSC.Nastran will determine the proper command to execute the companion program. To aid MSC.Nastran in determining the program’s location, a file named **marcrun.pth** must be available in the same directory where the MSC.Nastran input file resides. The marcrun.pth file must contain one line providing the location (complete path) of the run\_marc script. Typical examples of the line in the file marcrun.pth would be

```
/mycomputer/marc200x/tools      (UNIX)
c:\marc2003\tools                (Windows)
```

To this path MSC.Nastran appends the string “/run\_marc -jid name.marc -v no -iam nanl” and possible other items to form the complete string used to execute MSC.Marc. This complete string looks like the string shown in the PATH=3 example below.

If **PATH=2** is specified, it is expected that the directory with the run\_marc script is on the PATH. If PATH=2 is specified, MSC.Marc will be executed from inside MSC.Nastran using the command:

```
run_marc -jid jid.marc.dat -v no -iam nanl
```

Note that the last item (-iam nanl) requests a special type of licensing where MSC.Marc is bundled with MSC.Nastran. If you already have a full version of MSC.Marc, this command line option is not needed and can be turned off for those options which automatically generate the command line using the bulk data parameter PARAM,MARCIAMN,0

When **PATH=3** is specified, the complete command to execute MSC.Marc must be contained in a file named marc.pth (lowercase). This file should typically contain one line of the form:

```
/mycomputer/marc200x/tools/run_marc -jid name.marc -v no -iam nanl
```

**COPYR** is an optional item. If **COPYR** is specified, MSC.Marc output files will be copied to MSC.Nastran output files and/or deleted according to the option (=0, =1, or =2) shown.

COPYR Option	Copy MSC.Marc Output Files to MSC.Nastran Output Files	Delete MSC.Marc Input & Output Files
0 (default)	No	No
1 or -1 (see below)	Yes	Yes
2 or -2 (see below)	Yes	No
3	No	Yes

If COPYR is 1 or 2, Marc’s out and log files will be copied exactly as produced by MSC.Marc.

If COPYR is -1 or -2 the actions as shown above for +1 or +2 will occur, and MSC.Marc-type text will be converted to MSC.Nastran-type text (or any other desired text) using an ASCII file named marcfilt.txt. This file must be located in the same directory where the MSC.Nastran input resides or in the same directory where the MSC.Marc executable resides. The marcfilt.txt file can contain as many lines as desired like that shown below:

“Marc string 1” “Replacement String 1”  
“Marc string 2” “Replacement String 2”

That is, each line contains two strings. Each string starts and ends with a double quote sign (“). The MSC.Marc string must match the exact content and case as found in the MSC.Marc .out or .log files. The replacement string may be any string desired and can be the same length, shorter or longer than the MSC.Marc string. The two strings must



be separated by at least one space, but more spaces are acceptable. Line lengths for marcfile.txt, as well as MSC.Marc's .out and .log files are limited to 200 characters for the text replacement option.

The following MSC.Marc files are potentially affected by the COPYR option:

MSC.Marc Output File COPYR	MSC.Nastran Output Copied to	
name.marc.out	name.f06	1,2,-1,-2
name.marc.log	name.log	1,2,-1,-2
name.marc.t16	not copied, will remain if produced	
name.op2, fort.11 or ftn 11	not copied, will remain if produced	

**OUTR** is an optional item. If **OUTR** is specified, MSC.Marc output results will be converted to various type of Nastran formats. The type of output to be produced depends on The OUTR options entered as well as any DMAP entered in the executive control. If OUTR is omitted, no MSC.Marc output will be brought back into MSC.Nastran, but standard MSC.Marc .out, .t16 and/or .t19 will be available. This is the recommended option for MSC.Patran users because the MSC.Marc t16 results file can be read by the MSC.Nastran preference in MSC.Patran. The following options are available:

OUTR=OP2,F11,F06,PCH,XDB,T16,T19

Use any or all of the above to request the following options:

- OP2      Create output2 file named jid.op2 consisting of input model and output results datablocks
- F11      Create output file fort.11 or ftn11 (depending on the computer system) consisting of output results datablocks only.
- F06      Put MSC.Marc output results (displacements, stresses, strains) in MSC.Nastran's jid.f06 file using OFP. The resulting output will look just like any standard MSC.Nastran run.
- PCH      Create punch file named jid.pch with MSC.Marc's output in standard MSC.Nastran punch format.
- XDB      Create xdb database file named jid.xdb with input model and output results. This Option required PARAM,POST,0 in the Bulk Data Section.

- T16 MSC.Marc results will be saved during the MSC.Marc execution on a binary (or unformatted) file named `jid.marc.t16`
- T19 MSC.Marc results will be saved during the MSC.Marc execution on an ASCII file named `jid.marc.t19`

Note that geometry data blocks may be placed in the `op2` or `xdb` files but not both (unless you add your own DMAP statements). This is accomplished using the bulk data `PARAM,POST,-1` (for `op2`) or `PARAM,POST,0` (for `xdb`) parameters. If `PARAM,POST` is omitted, the geometry datablocks will not be present. Therefore you should normally only specify either `outr=op2` or `outr=xdb` but not both.

**STOP** is an optional item. **STOP** is used to prevent execution of MSC.Marc or halt MSC.Nastran after IFP module if so desired. DO NOT ENTER any of the STOP options if any of the OUTF options are entered as the DMAP generated automatically by MSC.Nastran will put an EXIT in the proper place. The various options are as follows:

If **STOP=1** MSC.Nastran will be gracefully stopped after IFP. This option is used to prevent MSC.Nastran from performing its own solution (normally used when the solution is performed by the MSC.Marc and required for 3D contact unless the OUTF option is used).

For **STOP=2** MSC.Marc will not be executed. This options is frequently used if the user wishes to examine the MSC.Marc input file and make changes prior to running MSC.Marc. However, if **STOP=2** is entered, the OUTF options will not be available.

**STOP=3** is a combination of **STOP=1** and **STOP=2**.. MSC.Nastran is stopped after IFP and MSC.Marc is not executed. This would be the normal STOP option if the user wants to examine a MSC.Marc input file, then execute MSC.Marc manually. The STOP option should not be entered if the user wants to obtain comparative results between standard MSC.Nastran solutions and MSC.Marc solutions (in which case all input options must be fully supported by both programs).

## Summary of Case Control and Bulk Data Entries relating to SOL 600

The following Case Control and Bulk Data entries are available in SOL 600 only and should not be used for other solution sequences. Please refer to the [MSC.Nastran Quick Reference Guide](#) for a detailed description of each entry.

## Case Control Enter

**BCONTACT**      Selects a set of contact bodies for a particular subcase. Options are available to select the entire model or all contact body definitions.

---

**Note:** Line contact options are not available in SOL 600.

---

## New Bulk Data Entries

<b>BCBODY</b>	Defines flexible or rigid contact bodies
<b>BCBOX</b>	Defines a 3D flexible contact region within a box (8 grid points or 8 xyz coordinates)
<b>BCHANGE</b>	Changes definition of contact bodies
<b>BCMATL</b>	Defines a 3D flexible contact region described by a material id
<b>BCMOVE</b>	Defines movement of bodies in contact
<b>BCPARA</b>	Contact parameters
<b>BCPROP</b>	Defines a 3D flexible contact region described by a property id
<b>BCTABLE</b>	Defines a contact table (which bodies can make contact)
<b>BSURF</b>	Defines a flexible contact surface using element IDs
<b>GMNURB</b>	Alternate way to describe a rigid contact surface made of nurbs
<b>IPSTRAIN</b>	Initial element plastic strains
<b>ISTRESS</b>	Initial element stresses
<b>MARCIN</b>	A way to directly input data to various portions of a MSC.Marc input file
<b>MARCOUT</b>	Selection of which outputs will be placed on MSC.Marc's t16/t19 results files
<b>MATEP</b>	Elasto-Plastic material property description
<b>MATHE</b>	Hyperelastic material property description (Mooney, Ogden, Foam, etc.)
<b>MATHED</b>	Damage model descriptions for hyperelastic materials
<b>MATT</b>	Failure model descriptions
<b>MATG</b>	Gasket material property description
<b>MATORT</b>	Orthotropic material property description

MATV	Visco-elastic material property description
MATVP	Viscoplastic or Creep material property description
MATTEP	Temperature dependent Elasto-Plastic material property description
MATTHE	Temperature dependent Hyperelastic material property description
MATTG	Temperature dependent Gasket material property description
MATTORT	Temperature dependent Orthotropic material property description
MATTVE	Temperature dependent Visco-elastic material property description
NLAUTO	Controls for automatic load or time stepping (MSC.Marc's AUTO STEP method)
NLDAMP	Damping constants for SOL 600
NLSTRAT	Solution strategy parameters for SOL 600
NTHICK	Define plate/shell thickness at nodes rather than from property entries
PARAMARC	Defines number of domains and how to define them for parallel processing in SOL 600
RESTART	Specifies restart options for SOL 600

---

**Note:** Line contact options are not available in SOL 600.

---

### **Bulk Data Parameters**

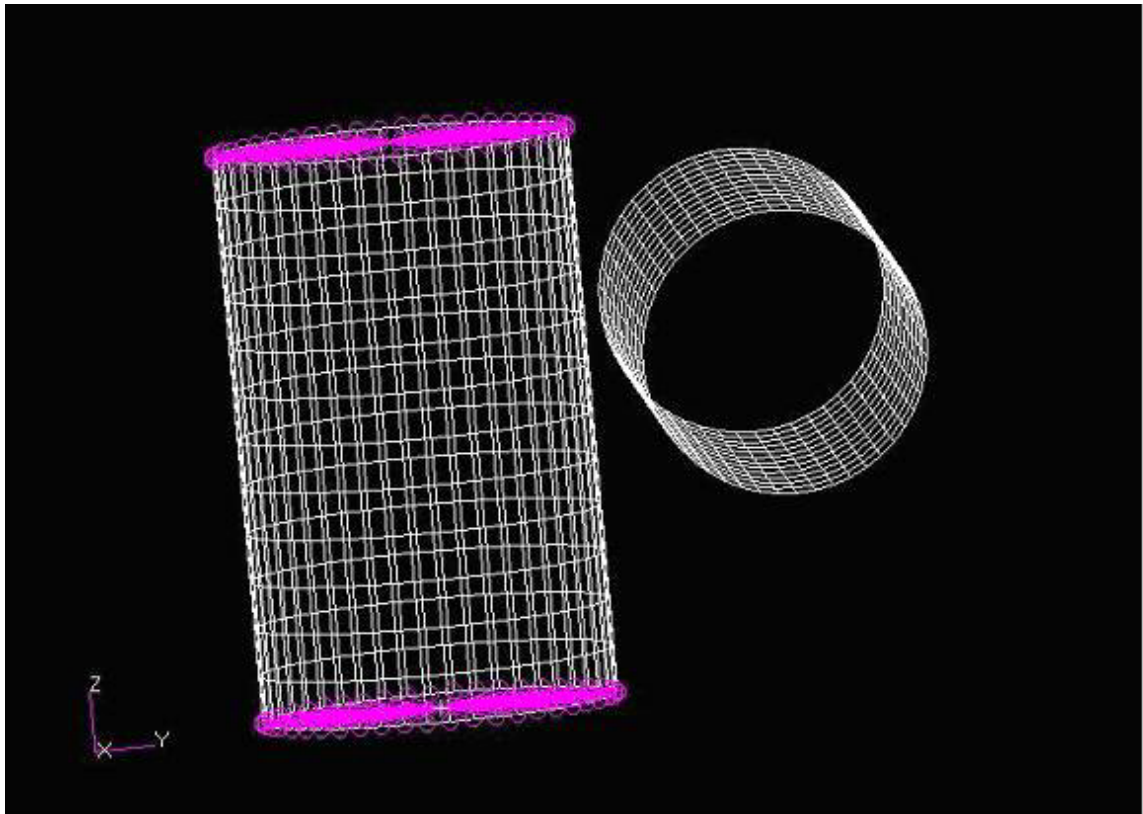
LGSTRN	Selects small or large strain analysis and strain measures to use for SOL 600
STRESS	Selects stress measure to use for SOL 600 (*)
MARCPOS	Selects whether a non-positive definite matrix may or may not be solved for SOL 600 (*)
MARCOPT	Selects bandwidth optimization option for SOL 600 (*)
MARCMEM	Selects amount of memory to be used by MSC.Marc (*)
MARCSIZi	Allows you to specify all values on MSC.Marc's SIZING entry (*)

MARCDISi	Allows you to specify all values on MSC.Marc's DIST LOADS memory specification entry (*)
MARCCONi	Allows you to specify all values memory-related values on MSC.Marc's contact entries (*)
MARCVERS	Specifies which version of MSC.Marc will be used (the current released version is the default) (*)
MARCRBE2	Specifies the MSC.Marc formulation of RBE2s (*)
MARCRBE3	Specifies the MSC.Marc formulation of RBE3s (*)
MARCRBAR	Specifies the MSC.Marc formulation of RBARs (*)
MARCSINC	Controls how often spline output files are written (*)
MARCT19	Allows you to generate an ASCII MSC.Marc results file
MARCT16	Allows you to generate an unformatted (or binary) MSC.Marc results file, done by default (*)
MARCEKND	Specifies which type of strains (total, plastic, elastic) are placed in op2,xdb files (*)
MARCSHLT	Specifies how many shell integration layers are used in plasticity analyses (*)
MARCPST	Specifies the MSC.Marc version designation of MSC.Marc's t16/t19 files (default=10) (*)
MARUPDAT	Specifies whether to use Total Lagrange or Updated Lagrange method (*)
MARSUMY	Requests a summary of output at each time or load step in the .out file (*)
MRSRING	Specifies a stiffness value to be added to the main diagonal of the stiffness matrix
MARCIAMN	Specifies whether a special or full version of MSC.Marc is to be used
MARCDILT	Specifies whether constant dilation formulation or not
MARCASUM	Specifies whether assumed strain formulation is used or not (*)
MARCGAUS	Specifies whether output is at Gauss points or center (*)
MRCONRES	Specifies whether continuous or separate output files are produced for restart runs

MARCRBAL	Specifies whether left hand and right hand sides of equation should be re-balanced before computing eigenvalues.
MRALIAS	Specifies an alias element type (similar element with a different formulation) (*)
MALIASij	Specifies an alias element type (similar element with a different formulation) (*)

### Example: Two Pipes Crushing (Contact & Large Strain):

Let's examine a problem to illustrate how easy it is to change a standard MSC.Nastran input file to describe two of the more advanced nonlinear capabilities. Consider the case of two pipes initially separated by a small distance as shown below.



There is a force at the top and bottom of the left cylinder forcing it toward the right cylinder. The left cylinder has rigid (RBE2) connections at the top and bottom and is connected to ground with soft springs. The right cylinder is pinned at one end and restrained in the radial direction at the other end. The soft springs are necessary for a nonlinear static analysis as the left cylinder would otherwise be unconstrained.

Partial input for SOL 600 is shown below (note that ID=NLSTATIC which is the same as 106, path=1 means the path to run MSC.Marc is specified in a file named marcrun.pth in the same directory as the MSC.Nastran input data and outr=op2,f06 means that the results will be placed in a standard MSC.Nastran OP2 file as well as printed in the f06 file). New or modified input data is shown in **bold**.

**SOL 600,NLSTATIC path=1 outr=op2,f06**

TIME 10000

CEND

ECHO = NONE

DISPLACEMENT(print,PLOT) = ALL

\$ NLSTRESS(PLOT) = ALL

STRESS(PLOT,CORNER) = ALL

strain(plot)=all

SPC = 1

LOAD = 1

NLPARM = 1

**bcontact=121**

BEGIN BULK

NLPARM	1	100		AUTO	1	25		YES+
+								+

+

PARAM,LGDISP,1

\$PARAM,POST,-1

PARAM,OGEOM,NO

PARAM,AUTOSPC,YES

PARAM,GRDPNT,0

FORCE	1	1301	0	1.	0.	5000.	0.
-------	---	------	---	----	----	-------	----

FORCE	1	1302	0	1.	0.	5000.	0.
-------	---	------	---	----	----	-------	----

SPC	1	663	23	0.			
-----	---	-----	----	----	--	--	--

SPC	1	664	23	0.			
-----	---	-----	----	----	--	--	--

CQUAD4	1149	2	1297	1298	1015	1016	
--------	------	---	------	------	------	------	--

CQUAD4	1150	2	1298	1299	1014	1015	
--------	------	---	------	------	------	------	--

CQUAD4	1151	2	1299	1300	1013	1014	
--------	------	---	------	------	------	------	--

CQUAD4	1152	2	1300	1011	1012	1013	
--------	------	---	------	------	------	------	--

RBE2	1153	1301	123456	327	328	329	330	331+
------	------	------	--------	-----	-----	-----	-----	------

+	332	333	334	335	336	337	338	339+
---	-----	-----	-----	-----	-----	-----	-----	------

+	340	341	342	343	344	345	346	347+
---	-----	-----	-----	-----	-----	-----	-----	------

+	348	349	14	15	16	17	18	19+
---	-----	-----	----	----	----	----	----	-----

+	20	21	22	23	24	25	26	27+
---	----	----	----	----	----	----	----	-----

+	28	29	30	31	32	33	34	35+
---	----	----	----	----	----	----	----	-----

+	36	326	13					
---	----	-----	----	--	--	--	--	--

RBE2	1154	1302	123456	363	364	365	366	367+
------	------	------	--------	-----	-----	-----	-----	------

+	368	369	370	371	372	373	374	375+
---	-----	-----	-----	-----	-----	-----	-----	------

+	376	377	378	379	380	381	382	383+
---	-----	-----	-----	-----	-----	-----	-----	------

+	384	385	50	51	52	53	54	55+
---	-----	-----	----	----	----	----	----	-----

+	56	57	58	59	60	61	62	63+
---	----	----	----	----	----	----	----	-----

+	64	65	66	67	68	69	70	71+
---	----	----	----	----	----	----	----	-----

```

+           72           49           362
MAT1           21000000.           0.25
PROD           4           2           1.E-5
CROD          1155           4           1301           1303
PROD           5           2           1.E-5
CROD          1156           5           1302           1304
$
BSURF, 101, 1, THRU, 576
BSURF, 102, 577, THRU, 1152
BCBODY, 111, , DEFORM, 101, 0, .25
BCBODY, 112, , DEFORM, 102, 0, .25
BCTABLE, 121, , , 1, , , , , +
+ , SLAVE, 111, , , .25,
+ , MASTER, 112
$
ENDDATA

```

Note that only the SOL 600,NLSTATIC Executive Control statement, the BCONTACT=121 Case Control command, BSURF, BCBODY and BCTABLE Bulk Data entries are changed or new compared to a standard MSC.Nastran input file. Of these, all except the SOL 600,NLSTATIC statement describe contact between the cylinders. Here is a detailed description of these commands:

SOL 600,NLSTATIC path=1 outr=op2,f06

The SOL 600,NLSTATIC command is just like a standard MSC.Nastran Executive Control SOL statement except for the “600,” before NLSTATIC and the extra information after NLSTATIC. This information is as follows:

- path=1 – This means that a file named marcrun.pth is in the same directory as the MSC.Nastran input data and will specify the full path to MSC.Marc’s “tools” directory.
- outr=op2,f06 – This means that after the MSC.Marc execution has completed, displacements, stresses tensors and plastic strains tensors will be placed in standard op2 and f06 files. These results although calculated by MSC.Marc will look identical in format to those produced by a standard MSC.Nastran analysis which does not spawn MSC.Marc. This should not be confused with the copyr option which would copy the MSC.Marc results to MSC.Nastran’s f06 file (in which case the output would still look like MSC.Marc output). The op2 and f06 results are produced by new code in MSC.Nastran which reads MSC.Marc’s binary output t16 file and generates a results-only op2 file which we call an f11 file. DMAP is generated “on the fly” to read the f11 file into the Nastran database (DBALL and MASTER)



using INPUTT2. Additional DMAP is generated to produce the desired OP2 file which now contains geometry datablocks as well as results datablocks. Finally DMAP is generated on the fly to place the results in the f06 file using the OFP module.

BCONTACT=121

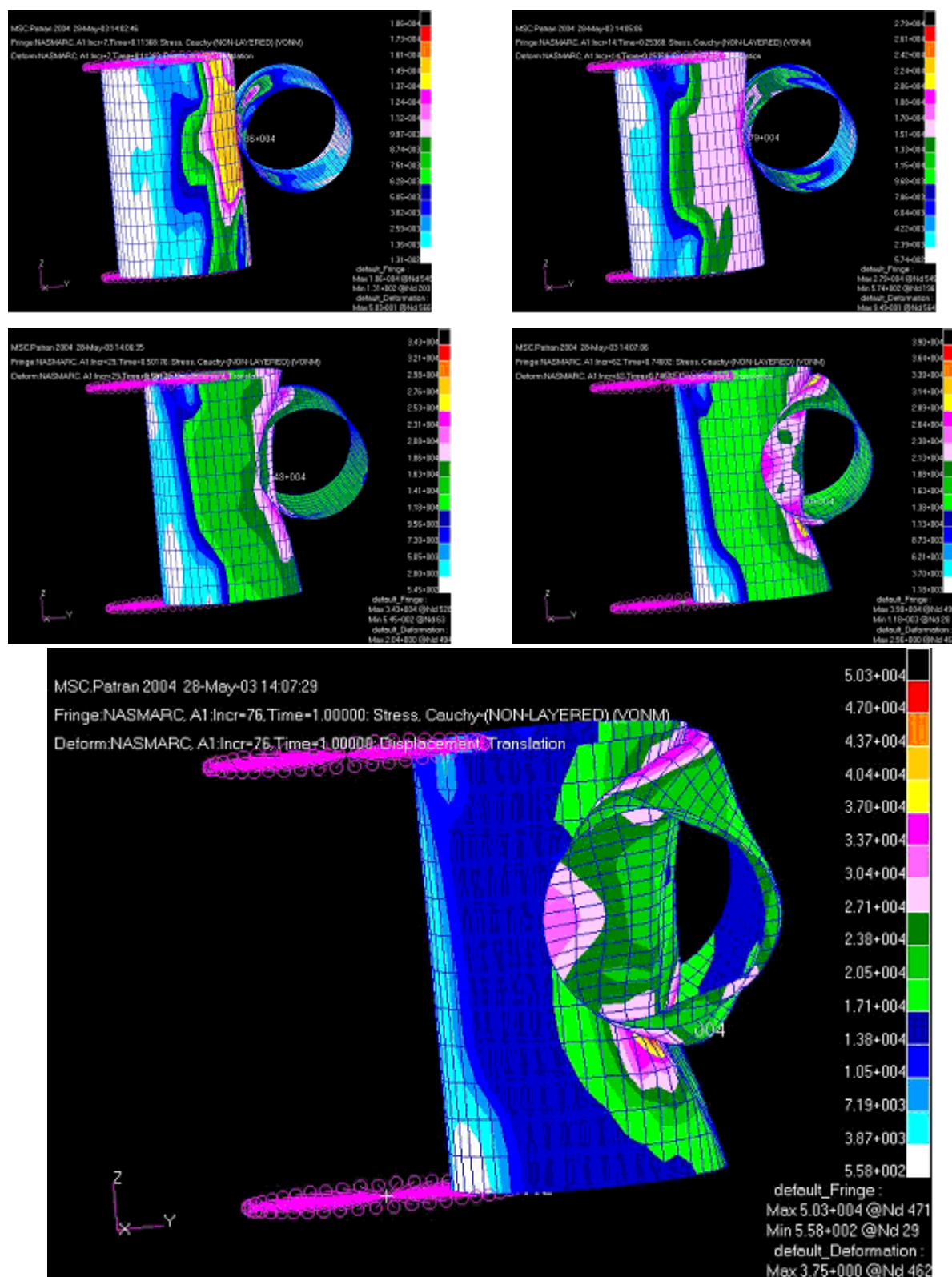
This Case Control command means that contact will be described using a BCTABLE Bulk Data entry with ID 121.

### Contact Bulk Data Entries:

```
BSURF, 101, 1, THRU, 576
BSURF, 102, 577, THRU, 1152
BCBODY, 111, , DEFORM, 101, 0, .25
BCBODY, 112, , DEFORM, 102, 0, .25
BCTABLE, 121, , , 1, , , , , +
+ , SLAVE, 111, , , .25,
+ , MASTER, 112
```

These Bulk Data entries describe the contact surfaces. Because large deformation is expected, all elements of the left cylinder may potentially contact those of the right cylinder. The two BSURF entries describe the elements of each cylinder. The two BCBODY entries describe the friction coefficients (0.25) for each of the cylindrical bodies. Finally, the BCTABLE hooks up the two cylindrical surfaces (as described by the two BCBODY entries) as a contact condition for the subcase. If there were more bodies and more subcases, different contact bodies could be described for each subcase if so desired. For small models, it is not necessary to describe the bodies. The Case Control BCONTACT command could be changed to BCONTACT=ALL and then each element in the model would check for contact with every other element in the model at each time (or load) step. Obviously, this would require more computer resources than if those elements that might possible contact are specified by the user.

The MSC.Nastran run is submitted the same as any other MSC.Nastran run. In this model, there is no stop option on the SOL 600,ID statement, so MSC.Marc will automatically be executed. Since OUTR=op2,f06 is specified, the results will be brought back into the “MSC.Nastran Database” to produce op2 and f06 (printed) output as explained above. The MSC.Marc input and output files will remain (in the same directory as the MSC.Nastran input data) for the user to inspect after the run. Results may then be examined or plotted using any standard MSC.Nastran postprocessor. For example, the following results were produced using MSC.Patran from the op2 file.



We elected to place the results in the f06 file as well as the op2 file. A sample of some of the results from the f06 file follow.

The following are samples “total displacements i.e. relative to the initial position, Cauchy stresses and Plastic strains at the final position (“time”=1.0)

TIME = 1.000000E+00

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
2	G	-8.006727E-03	3.679180E+00	1.043056E-03	2.845422E-03	2.369322E-02	1.204350E-02
3	G	-2.011038E-02	3.677816E+00	1.519643E-03	2.597190E-03	1.290626E-02	3.169994E-02
5	G	-3.224353E-02	3.676953E+00	1.765113E-03	2.757947E-04	7.053518E-03	6.177027E-02
6	G	-3.788499E-02	3.676907E+00	1.501512E-03	4.993313E-03	9.955634E-03	8.023518E-02
10	G	-1.402493E-02	3.683437E+00	1.045132E-04	1.284459E-03	-2.054870E-02	2.056682E-02

ELEMENT ID.		FIBER DISTANCE	STRESSES IN QUADRILATERAL ELEMENT COORD SYSTEM			ELEMENTS (QUAD4) PRINCIPAL STRESSES (ZERO SHEAR)			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
0	1	-1.000000E+00	2.460815E+03	1.935240E+03	2.565853E+03	42.0762	4.777302E+03	-3.812473E+02	4.978885E+03
		1.000000E+00	-9.141631E+03	-8.298519E+03	5.878244E+03	47.0510	-2.826734E+03	-1.461342E+04	1.342513E+04
0	2	-1.000000E+00	-1.057425E+04	5.934804E+03	-1.950141E+03	-83.3538	6.162038E+03	-1.080149E+04	1.487286E+04
		1.000000E+00	7.265749E+03	-5.363500E+03	6.036017E+03	21.8539	9.686568E+03	-7.784319E+03	1.516010E+04
0	3	-1.000000E+00	2.479291E+03	1.367514E+04	4.870156E+03	69.4885	1.549713E+04	6.572963E+02	1.517916E+04
		1.000000E+00	-3.051442E+03	-9.212279E+03	2.676518E+03	20.4934	-2.051085E+03	-1.021264E+04	9.357237E+03

TIME = 1.000000E+00

TIME = 1.000000E+00

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

ELEMENT ID.		FIBER DISTANCE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
0	1	-1.000000E+00	-1.928319E-04	1.118255E-04	3.583091E-04	56.5159	3.488417E-04	-4.298481E-04	6.755803E-04
		1.000000E+00	-2.790075E-03	-1.048802E-04	2.158601E-03	60.9403	1.094594E-03	-3.989548E-03	4.634821E-03
0	2	-1.000000E+00	-1.151780E-03	8.069643E-04	-1.218730E-03	-64.3927	1.391073E-03	-1.735889E-03	2.713512E-03
		1.000000E+00	5.727272E-05	-6.128556E-04	2.944954E-03	41.7545	2.686162E-03	-3.241745E-03	5.141228E-03
0	3	-1.000000E+00	-5.297469E-04	1.939139E-03	-9.462342E-04	-71.2644	2.260077E-03	-8.506845E-04	2.784641E-03
		1.000000E+00	-1.830743E-05	-1.381564E-03	1.953345E-03	35.3817	1.368922E-03	-2.768794E-03	3.651086E-03

## Where Can I Find More Information

MC.Nastran Implicit Nonlinear Analysis - SOL 600 is documented in the following Manuals and guides:

- MSC.Nastran Quick Reference Guide
- MSC.Nastran Implicit Nonlinear User's Guide
- MSC.Patran User's Guide
- MSC.Patran – MSC.Nastran Preference Guide volumes 1 and 2
- MSC.Marc Volume A – Theory and User Information
- MSC.Marc Volume B – Element Library
- MSC.Marc Volume C – Program Input

## 8.2 Prerelease of the General Nonlinear Solution Sequence SOL 400

### Introduction

In order to improve the nonlinear solution procedure in MSC.Nastran, a general nonlinear solution sequence SOL 400 is introduced. This solution sequence will include all nonlinear analyses types, such as nonlinear static analysis, nonlinear transient analysis, nonlinear buckling analysis, and nonlinear normal modes analysis. SOL 400 also supports linear transient analysis. Eventually, SOL 400 is intended to replace the current nonlinear static solution sequence, SOL 106, and the nonlinear transient solution sequence SOL 129. In future releases, the nonlinear heat transfer solution sequences, SOL 153 and SOL 159 will be included in SOL 400. In addition, it is planned to include 3D surface contact with the same text interface as SOL 600. The prerelease is intended to get user feedback. In the current stage, SOL 400 does not replace SOL 106 and 129.

### Benefits

- A new SUBCASE/STEP combination allows the user to perform nonlinear analysis similar to the existing methods plus the option to include independent loadings in a single run.
- When fully implemented, all nonlinear solution procedures, such as linear static analysis, nonlinear static analysis, transient nonlinear analysis, etc., can be performed in a single run. For example, you can run a transient analysis using the nonlinear static analysis results of a previous step of the initial condition.
- An improved nonlinear iteration procedure, which speeds up convergence.
- Allows the simulation of existing nonlinear solution sequences such as SOL 106 or SOL 129.
- The new Lagrange rigid elements allow correct solution of the rigid elements such as RBAR in a geometric nonlinear analysis.
- A new large rotation representation method, the right rotation method, to be used in a geometric nonlinear analysis.
- A comprehensive Nonlinear Iteration Summary Table allowed user to track the solution status easily.
- A new user friendly restart procedure.

- A more flexible output method allowing intermediate results output during the analysis.

## Limitations for the Current Release

Because SOL 400 is a pre-release, the following capabilities are not supported:

- Nonlinear transient analysis.
- Nonlinear normal modes and nonlinear buckling analysis.
- RFORCE and Creep.
- Arc-length method (input by NLPCI Bulk Data entry).
- The GPFORCE and ESE output requests.
- Slide line contact.

The first 5 items will be supported in future release. The last item ‘line contact’ will be replaced by a more general contact capability with the same text input as SOL 600.

Features supported by SOL 400 in the current release are discussed in the following section.

## Analysis Steps

The sequence of load application and analysis method is important in determining the final response of a nonlinear structure. On the other hand, it is convenient for the user to have the ability to perform analyses of multiple independent cases in a single run. For this reason, a new Case Control command, STEP, is created exclusively for the new nonlinear solution sequence SOL 400. This command provides a mechanism for defining the multiple load steps and other characteristics for a SUBCASE in a sequential manner, and, at same time, allow the user to run multiple independent cases in a single run. The format of this Case Control command is:

STEP n

where n is the step identification number. The usage of the SUBCASE and STEP Case Control commands are shown below.

- The simplest input contains one SUBCASE and one STEP. Either or both the SUBCASE and STEP commands may be omitted from the input. The default value for both SUBCASE and STEP is 1.

- With one SUBCASE and multiple steps, the procedure is only slightly more complicated than previous example. Each step defines the total new external load and other characteristics for the step, which will be applied by the completion of the step. The solution of any STEP is a continuation of the solution of its previous STEP. STEP identification numbers must be in ascending order. The following is a typical example:

```
SUBCASE 1
NLPARM = 200
  STEP 10
  LOAD = 10
  STEP 20
  LOAD = 20
  STEP 30
  LOAD = 30
```

- Multiple SUBCASEs may be executed in a single analysis where loads and boundary conditions may change. All SUBCASEs are independent of each other, i.e., no load history information is transmitted from one SUBCASE to the next. At the start of each SUBCASE, the deflections, stresses and strains throughout the model are zero. In each SUBCASE, there may be a number of STEPs. For example

```
NLPARM = 200
SUBCASE 1
  STEP 10
  LOAD = 10
  STEP 20
  LOAD = 20
SUBCASE 2
  STEP 100
  LOAD = 100
```

In the previous example, the solutions for SUBCASE 1 and SUBCASE 2 are independent of each other. If the solution diverges in a particular step, MSC.Nastran will terminate the solution of the current subcase and proceed to the next subcase if it exits.

- A case control command placed below the step level allows that command to vary from on step to another. If it is placed above the step level, the command remains constant for all steps in the subcase. Most of the case control commands, which can be placed below the subcase level, can also placed below the step level. For example, all steps in above examples use the same Case Control command, NLPARM = 200.
- The meaning of multiple SUBCASEs **without** STEP is dependent on the system cell NASTRAN SYSTEM (366) as follows:

0 - The solutions of all SUBCASEs are independent of each other. This is consistent with the new SOL 400 procedure. MSC.Nastran will keep all SUBCASE commands in the Case Control file and insert internally a "STEP 1" for each SUBCASE. This is a default value.

1 - The solution of each SUBCASE is a continuation of the previous SUBCASE. This is similar to the solution sequence SOL 106 procedure. MSC.Nastran internally converts all the SUBCASE identification number to STEP identification numbers and insert a "SUBCASE 1" before the first STEP.

The default is 0.

## Linear Static Analysis and Nonlinear Static Analysis

The SOL 400 uses an enhanced dynamic solution algorithm, which makes the linear static solution and the nonlinear static solution become special cases of the general nonlinear solution procedure. For this release, only the linear static analysis and the nonlinear static analysis are supported.

The linear static analysis and the nonlinear static analysis are requested by the Case Control command, ANALYSIS. For example:

```
SUBCASE 10
  STEP 1
    ANALYSIS = LNSTATIC
    LOAD = 10
  STEP 2
    ANALYSIS = NLSTATIC
    LOAD = 20
    NLPARM = 20
```

In the previous example, SUBCASE 10 has two steps: the first step requests a linear static analysis and the second step requests a nonlinear static analysis. The default ANALYSIS method, i.e., ANALYSIS command in the Case Control file, is NLSTATIC.

## Vector Operations and Convergence Criteria

The convergence criteria are specified by using the Bulk Data entry, NLPARM. In performing the convergence tests, we compute three error factors: the displacement, the load, and the work (energy) error factors, which are printed in the “**Nonlinear Iteration Summary Table**” on page 367. These three error factors must satisfy the error tolerance rules specified by CONV, EPSU, EPSP, and EPSW on the NLPARM Bulk Data entry.

In computing the error factors, SOL 106 used the l-set (the leftover set) vectors for displacements and forces. By using this method, the effect of SPC loads and MPC constraints are accounted for only indirectly. Also, there are difficulties accounting for



for the effect of Lagrange multipliers for the Lagrange rigid elements, which are discussed below. For these reasons, in SOL 400, whenever possible, the matrix and vector operations, which include the computations of error factors, are performed in p-set (the physical set). For MSC.Nastran set definition, please refer to the “[Degree-of-Freedom Set Definitions](#)” on page 778 of the *MSC.Nastran Quick Reference Guide*.

Another major modification is the computation of the work error. In SOL 106, the work error is based on the multiplication of the residual force and the displacement change. During iteration, both the residual force and the displacement change become smaller; therefore, the convergence rate of this value is proportional to the square of the convergence rate of the solution. Thus it can become very small near convergence. Also, it does not have a counter part in the physical world. In SOL 400, the total work done to structure model is computed during iterations and the work error is estimated based on the total work. Using this method, the work error gives an estimation of error in actual work done to the structural model. The total work for each iteration is printed on the “[Nonlinear Iteration Summary Table](#)” on page 367. Note, however, that this total work is only an approximation.

## Solution Algorithm and Simulation of SOL 106

In SOL 400, the solution algorithm is modified substantially in the following areas:

- The nonlinear iterations and stiffness update algorithm for the AUTO and SEMI of the KMETHOD on the NLPARM Bulk Data entry. The action of system 401 is as follows:
- The algorithm for load bisections.
- The solution divergence processing.
- For most of the problems tested, SOL 400 yields equal or better performance than that of SOL 106. However, in order to provide an option similar to that of SOL 106, system cell ITRFMT (SYSTEM (401)) can be used.
- 0 – Uses the SOL 400 algorithm. This is the default value.
- -2 – Uses algorithm similar to the SOL 106 algorithm.

The default is 0. Note that even with ITRFMT=-2, the iteration solution sequence of SOL 400 will not be the same as that of SOL 106. One reason is that the vector operations in SOL 400 use the p-set, while SOL 106 use the l-set. See the section on “[Vector Operations and Convergence Criteria](#)” on page 365 for further details.



Lagrange Rigid Elements and the Right Rotation Method

The linear rigid elements such as RBAR, RBE1, etc., have been available in MSC.Nastran since the inception of MSC.Nastran. However, these rigid elements use a small rotation theory, which may give erroneous solution in a geometrical nonlinear analysis.

In the current release, a new type of rigid elements, the Lagrange rigid elements, is implemented. For the nonlinear analysis, the Lagrange rigid element is available for SOL 400 only. The Lagrange rigid elements use the large rotation theory in a geometric nonlinear analysis (PARAM, LGDISP, 1) so that the nonlinear solution is correctly computed. Also, for the large rotation theory, a new rotation representation method, the right rotation method, is implemented. For the Lagrange rigid elements and the right rotation method, please refer to “[Rigid Element Enhancements](#)” on page 243.

Nonlinear Iteration Summary Table

In order to allow the user to track the solution sequence during the nonlinear iteration, a detailed Nonlinear Iteration Summary Table is generated. A line for each iteration is generated in the f06 file during the nonlinear iteration. Due to printing of the average and the maximum displacements, you can see the solution status before the end of the solution. This can be useful for large nonlinear problems. However, even for small problems, you can examine the table to observe the progress of the analysis. An example of this table is given below and the descriptions of information given in this table are shown in [Table 8-1](#).

NON - LINEAR ITERATION MODULE OUTPUT																					
STIFFNESS UPDATE TIME 0.01 SECONDS															SUBCASE						
1	STEP 1																				
ITERATION TIME 0.02 SECONDS																					
LOAD NO.		- - ERROR FACTORS - -				CONV		ITR		MAT		NO.		AVG		TOTL		- - - - - DISP - - - - -			
LINE_S	NO.	TOT	TOT																		
STEP	INC	ITR	DISP	LOAD	WORK	RATE	DIV	DIV	BIS	R_FORCE	WORK	AVG	MAX AT GRID C FACT NO QNV KUD ITR								
0.1000	1	1	1.00E+00	1.62E+02	1.62E+02	1.000	0	1	0	1.56E+02	9.852E-01	1.32E-02	9.800E-02	8							

Table 8-1 Description of the Nonlinear Iteration Summary Table

LOAD STEP	The percent of the current step + (step no. - 1). For example, 0.1 means 10 % of the first step and 1.337 means 33.7 % of the second step.
NO. INC	Number of increment. The total number of load increment for a step is requested by NINC on the NLPARM Bulk Data entry.

**Table 8-1 Description of the Nonlinear Iteration Summary Table (continued)**

ERROR FACTORS: DISP LOAD WORK	There are three error factors: displacement, load, and works. In order for an increment to converge, these factors must satisfy the error tolerance rules specified by CONV, EPSU, EPSP, and EPSW on the NLPARM Bulk Data entry.
CONV RATE	Converge rate, which denote how fast the solution converges for the current increment. A value of 0.0 means fast converges and a value > 1.0 means that the solution will never converge.
ITR DIV	Number of iteration divergence. Action to correct solution divergence will be taken if ITRDIV > MAXDIV.
MAT DIV	Number of material divergence + 1, i.e., it will be 1 if there is no material divergence. The material divergence is due to bad creep strain or excessive sub-increments in plasticity.
NO. BIS	Number of bisection performed.
AVG R_FORCE	Average residual force. In order for an increment to converge, this value must become very small.
TOTAL WORK	Accumulated total work done to the structure model. This value is only an approximation.
DISP: AVG MAX AT GRID C	The average displacement, the maximum displacement and its grid point identification number and component number.
LINE_S: FACT NO	Line search factor and number of line searches performed.
NO. QNV	Number of quasi-Newton vectors computed.
TOT KUD	Total number of stiffness updates performed
TOT ITER	Total number iterations performed, including the number of stiffness updates.

## Restart

The purpose of a nonlinear restart is to allow the user to use the material or the geometrical properties of a previously converged solution as a new starting point to continue the analysis. This is useful when the user want to change the loading sequence, the solution criteria, or to extend the analysis.

For SOL 400, a user-friendly restart procedure has been implemented. For this restart procedure, the following principles are noted:

- The restart must be continued at a previously converged solution point by specifying a SUBCASE, STEP, and/or load factor. This is accomplished by using the Case Control command, NLRESTART.
- The geometry and the initial material properties of the structural model cannot be modified. This is obvious because any modification to the geometry or the initial material properties would invalidate the previous analysis and require the nonlinear solution to start from the very beginning. In such cases, it is simpler to initiate another cold start run.

Procedures for performing the restart are described in the following subsections.

## File Management Commands

For a restart, the data of the cold start must be made available by using the File Management commands. For nonlinear restart, two commands are needed: ASSIGN and RESTART. These two commands are existing restart commands and no special requirements are needed for SOL 400.

There are many methods to fetch data for a restart. One method is given in the example below. For other methods, please refer to the “[The File Management Section \(FMS\)](#)” on page 35 of the *MSC.Nastran Quick Reference Guide* or “[Interface With Other Programs](#)” on page 499 of the *MSC.Nastran Reference Manual*.

## Case Control Modifications

The presence of a Case Control command NLRESTART indicates that the current run is a restart run. The Case Control file contains **both** subcases and steps that were included in the cold start run, **and** those that are to be added in the restart run. The first subcase, step and/or load factor to be executed in the restart is indicated by the options on the NLRESTART command. The is shown by the following example:

```
NLRESTART SUBCASE 1, STEP 2, LOADFAC 0.3
SUBCASE 1
    STEP 1
        LOAD = 10
```

```

STEP 2
      LOAD = 20
STEP 3
      LOAD = 30

```

In the above example, the first step through load factor 0.3 of the second step has been previously executed. The restart execution begins with load factor 0.3 of the second step, and continues through the end of the third step. For restart, the case control file structure for SUBCASE and STEP commands must be the same as the cold start run up to the restart point. After the restart point, the user may modify or add Case Control SUBCASE and STEP commands. For example, in above example, steps 1 and 2 must exist in the cold start. However, step 3 may or may not exist in the cold start run.

The following Case Control commands may be modified in a nonlinear restart:

- Boundary conditions such as MPC and SPC.
- Nonlinear solution control, NLPARM.
- LOAD requests.
- Output request such as DISP and NLSTRESS.
- The analysis type (ANALYSIS).

Depending on the option selected with the NLRESTART command, the nonlinear restart may be logically divided into three types: a case restart, a step restart, or a load factor restart:

- The case restart begins the execution with a SUBCASE command. All five types of modification described above are acceptable for a case restart.
- The step restart begins the execution with a STEP command, which may be a new step or a previously executed step. Although boundary condition and analysis type modifications are allowed, the user has the responsibility to determine whether they are meaningful. Special attention should be given to the **analysis type** modification, it may not be meaningful in many situations and, thus, may lead to erroneous results.
- The load factor restart begins execution with a LOADFAC command. For a load factor restart, user may not modify analysis type, boundary conditions, and load requests. User needs to exercise discretion when attempting other types of modification at this level. Also, in order to perform this type of restart, the INOUT=YES must be set on the NLPARM Bulk Data entry on a cold start run so that the required data are available for restart.

## Bulk Data Modifications

The Bulk Data file for a nonlinear restart contains only those entries which are to be added to the cold start. The Bulk Data deletion entry ‘/’ should not be used. This is to serve as a reminder that the geometry and the initial properties cannot be modified. User may make modifications to the Bulk Data file by introducing new entries which may be copies of the original entries with appropriate changes and new identification numbers. The following is list of entries that can be added in a restart run:

- Load entries such as LOAD, FORCE, PLOAD4, and SPCD.
- NLPARM entries.
- Boundary condition entries such as SPC, SPC1, and MPC.

## Output

The output is requested by using the appropriate Case Control commands--e.g., DISP, STRESS, etc.

Two output are special for SOL 400. One is the Nonlinear Iteration Summary Table, which will allow the user to monitor the status of the nonlinear solution sequence. This table is discussed in Description of the “[Nonlinear Iteration Summary Table](#)” on page 367.

Another one is the PARAM, PH2OUT. In MSC.Nastran, the nonlinear analysis is divided into three phases. In phase I, the upstream superelements and geometrical data of the residue structure are processed. In phase II, the solution of the residue structure is obtained. And, in phase III, data are recovered for both the residual structure and upstream superelements. The output requests are processed in phase III. In many situations, the solution terminates prematurely in phase II and the user is unable to obtain any output for debugging the model. In order to remedy this deficiency, the parameter PH2OUT is for requesting phase II output. The phase II output consist of all output requested by Case Control commands for the residual structure only and are printed in sort 1 format. The parameter PH2OUT has following options:

- 0 -- Output phase III outputs only. In this case, both outputs for residual structure and upstream superelements are processed. This is the default value.
- 1 -- Output phase II outputs only. In this case, no output is processed for upstream superelements.
- 2 -- Output both phase II and phase III output. In this case, the output for the residual structure may be redundant.

If there is an abnormal termination in phase II, user can rerun the job with phase II output request for debugging.

## User's Interfaces

The following section summarizes the User's Interface for SOL 400., which are important to the nonlinear analysis and can be used in SOL 400 Refer to the [MSC.Nastran Quick Reference Guide](#) for further details.

## Nastran System Cells

- STPFLG (SYSTEM (366)) – Selects the SUBCASE or STEP methodology when there are a number of SUBCASE commands and no STEP command in the Case Control file, see “[Analysis Steps](#)” on page 363.
- ITRFMT (SYSTEM (401)) – Selects the convergence parameter computation method and the divergence solution checking method, see section “[Solution Algorithm and Simulation of SOL 106](#)” on page 366 of SOL 106.

## File Management Commands

The following File Management commands are required for restarts. Please refer to the “[The File Management Section \(FMS\)](#)” on page 35 of the *MSC.Nastran Quick Reference Guide* or Chapter 12 of the “[Interface With Other Programs](#)” on page 499 of the *MSC.Nastran Reference Manual*.

- ASSIGN -- Assigns database physical file names used by MSC.Nastran.
- RESTART -- Requests that data stored in a previous run be used in the current run.

## Executive Control Command

- SOL 400 or SOL NONLIN -- Requests the SOL 400 general nonlinear solution sequence

## Parameters

- PARAM, LANGLE -- Selects the method to represent large rotations in a geometric nonlinear analysis -- 1 for the Gimbals angle method, 2 for the left rotation method, and 3 for the right rotation method. For detailed descriptions, refer to User's Interface section in Rigid Elements of the current Release Guide. The default value is 2 for the current release and will become 3 for a future release.

- PARAM, LGDISP -- Requests a geometric nonlinear analysis.
- PARAM, FOLLOWK -- Requests whether the follower force stiffness will be used in a geometric nonlinear analysis.
- PARAM, FKSVMFAC -- Controls whether the symmetrical follower force stiffness will be used in a geometric nonlinear analysis.
- PARAM, MAXLP -- Specifies maximum number of iterations for element relaxation and material point sub-increment process.
- PARAM, NLAYERS -- Specifies the number of layer for through thickness integration in the material nonlinear analysis.
- PARAM, NLTOL -- Selects defaults for CONV, EPSU, EPSP, and EPSW for the NLPARM Bulk Data entry.
- PARAM, PH2OUT -- Requests phase II or phase III output for a nonlinear analysis, see “[Output](#)” on page 371.

## Case Control Commands

- ANALYSIS -- Selects solution method for an analysis step, see “[Linear Static Analysis and Nonlinear Static Analysis](#)” on page 365.
- NLRESTART -- Requests a restart execution at a specific solution point for SOL 400, see “[Restart](#)” on page 369.
- NLSTRESS -- Requests the form and type of the nonlinear element stress output.
- STEP -- Delimits and identifies an analysis step, see “[Analysis Steps](#)” on page 363.

## Bulk Data Entries

- MATHP- Specifies the hyperelastic material properties for an element.
- MATS1 -- Specifies the stress-dependent material properties for an element.
- NLPARM -- Defines a set of parameters for nonlinear static analysis iteration strategy.

## Examples

The following two examples show the input for a cold start run and a restart run of the nonlinear static analysis. The intention of these examples is to show the input structure for SOL 400. The model itself and parameters specified on the NLPARM Bulk Data entry are not important.

Example one, NLSTS5, is a cold start run. This model only contains a single HEXA element and one QUAD4 element. It has both material nonlinearity (MATS1) and geometrical nonlinearity (PARAM, LGDISP, 1). All the **bold**-font statements are entries pertaining to the nonlinear analysis.

When the job is completed, in addition to the regular .f04, .f06 and .log files, two database files will also be created. They are 'inlstrs5.DBALL' and 'inlstrs5.MASTER', which are needed for the restart run.

```
ID MSC, NLSTRS5
TIME 5
SOL 400
CEND
TITLE      = THE COLD START RUN OF THE NONLINEAR STATIC ANALYSIS
SUBTITLE   = ELASTIC-PLASTIC ANISOTROPIC MATERIAL
LABEL      = UNIAXIAL STRESS STATE
$ this model has 1 HEXA AND 1 QUAD4.
ECHO       = SORT
ANALYSIS = NLSTAT
DISP       = ALL
SPCFORCES  = ALL
SUBCASE 101
    NLPARM = 100
    STEP 10
        LOAD = 101
    STEP 20
        LOAD = 102
    STEP 30
        LOAD = 103
SUBCASE 102
    NLPARM = 200
    STEP 10
        LOAD = 101
    STEP 20
        LOAD = 102
    STEP 30
        LOAD = 103
BEGIN BULK
$ DEFINE GRID POINT COORDINATES AND BOUNDARY CONDITIONS.
$      GRID-ID COORD-SYS      X      Y      Z      DOFS FIXED
GRID    1      0      0.0    0.0    0.0      123456
GRID    2      0      1.0    0.0    0.0      3456
GRID    3      0      1.0    1.0    0.0      3456
GRID    4      0      0.0    1.0    0.0      13456
GRID   101      0      0.0    0.0    1.0      12456
GRID   102      0      1.0    0.0    1.0      456
GRID   103      0      1.0    1.0    1.0      456
GRID   104      0      0.0    1.0    1.0      1456
$
GRID   11      0      0.    0.    0.      123456
GRID   12      0      1.    0.    0.      3456
GRID   13      0      1.    1.    0.      3456
```



```

GRID      14      0      0.      1.      0.      13456
$
$  DEFINE ELEMENT - 8 NODE ISOPARAMETRIC SOLID ELEMENT.
$      EL-ID  PROPERTY      C O N N E C T I V I T Y
CHEXA      1      100      1      2      3      4      101      102      +1
+1      103      104
$
CQUAD4     11      101      11      12      13      14
$
$  DEFINE ELEMENT GEOMETRIC PROPERTIES.
$
PSOLID     100      120
PSHELL     101      121      1.0
$
$  DEFINE MATERIAL PROPERTIES.
$
MAT9       120      7.9812E6-6.149E50.      1.5136E60.      0.      .9539E6+MT1
+MT1       0.      1.5136E60.      0.      1.E6      0.      0.      0.      +MT2
+MT2       3.0824E60.      0.      3.E5      0.      3.E5
MATS1     120      PLASTIC 5.E5      5.E3
$
MAT2       121      7.9812E6-6.149E51.5136E64.9539E61.5136E63.0824E6
MATS1     121      PLASTIC 5.E5      5.E3
$
NLPARM     100      10      ITER      1      YES
NLPARM     200      20      AUTO
PARAM      LGDISP  1
$
$  DEFINE GRID POINT STATIC LOADS BY SPECIFYING VECTORS.
$      LOAD SET GRID  COORD.      SCALE  N1      N2      N3
$      ID      ID      SYS. ID      FACTOR
FORCE     101      2      0      5.0E3  0.5      0.0      0.0
FORCE     101      3      0      5.0E3  0.5      0.0      0.0
FORCE     101     102      0      5.0E3  0.5      0.0      0.0
FORCE     101     103      0      5.0E3  0.5      0.0      0.0
$
FORCE     101     12      0      5.0E3  1.0      0.0      0.0
FORCE     101     13      0      5.0E3  1.0      0.0      0.0
$
LOAD      102      2.0      1.0      101
LOAD      103      3.0      1.0      101
$
ENDDATA

```

Example two, NLSTRS5R, is a restart run. This example continues the job from the end of the last run with one extra step (STEP 40) in SUBCASE 102. Before the new STEP 40, the Case Control file structure for SUBCASE and STEP must be the same as that of the cold start. On the other hand, the output requests such as STRESS, etc can be modified. In this example, the **bold**-font statements are pertaining to the restart run.

Two file management commands -- ASSIGN and RESTART -- are required to fetch stored data for restart. One way is shown here. There are many other ways to obtain data for restart. Please consult the “[The File Management Section \(FMS\)](#)” on page 35 of the *MSC.Nastran Quick Reference Guide* or “[Interface With Other Programs](#)” on page 499 of the *MSC.Nastran Reference Manual*.

```

ASSIGN MASTER='nlstrs5.MASTER' $ File management command to assign
$                               master Dbset for the current run.
RESTART $ File management command, request data stored in
$       'nlstr5.MASTER' are used for restart.
ID MSC, NLSTRS5R
TIME 5
SOL 400
CEND
TITLE      = THE RESTART START RUN OF THE NONLINEAR STATIC ANALYSIS
SUBTITLE   = ELASTIC-PLASTIC ANISOTROPIC MATERIAL
LABEL      = UNIAXIAL STRESS STATE
ECHO       = SORT
ANALYSIS   = NLSTAT
DISP       = ALL
SPCFORCES  = ALL
STRESS    = ALL
ELFORCE   = ALL
OLOAD     = ALL
NLRESTART $   Since there is nothing specified in this command,
$             job will restart from the end of the previous run.
SUBCASE 101
    NLPARM = 100
    STEP 10
        LOAD = 101
    STEP 20
        LOAD = 102
    STEP 30
        LOAD = 103
SUBCASE 102
    NLPARM = 200
    STEP 10
        LOAD = 101
    STEP 20
        LOAD = 102
    STEP 30
        LOAD = 103
    STEP 40          $ | New added STEP
    NLPARM = 300
    LOAD = 104

$
BEGIN BULK
PARAM,PH2OUT,1 $ Request the intermediate output in PHASE II - the
$             residual run
NLPARM 300      30          AUTO
LOAD 104      3.0      1.0      101

```

```
$  
$ no other bulk data is required for restart.  
$  
ENDDATA
```

When checking the .f06 output file after the Restart run, you will find that the printout only includes the load increments that have been requested in the restart run. There is no printout for the load increments before the restart point.

## 8.3 Performance Improvements in SOL 129

### Introduction

Nonlinear transient response analysis in SOL 129 has become more robust and reliable. It can be used for linear analysis to take advantage of the automatic time stepping. If the problem is truly linear, i.e. without NOLIN, SOL 129 should converge in one iteration at every time step. The computing speed is comparable to SOL 109 in such cases if the same number of time steps are used. MSC.Nastran 2004 implements an updated automatic time step adjustment algorithm. The automatic time stepping algorithm implemented over ten years ago was hinged on the dominant response frequency which is based on the system characteristics. This strategy is not effective if the loading changes faster than the system response. In case of an impact loading, an indiscriminate use of automatic time stepping would cause the impulse represented inadequately or unrecognized by the algorithm. This could be avoided by turning off the automatic time stepping. But it relies on the user's intervention and therefore is not user-friendly.

### Benefits

SOL 129 will activate time step adjustment unless it is suppressed. As a result, the solution will be more accurate and the solution time is reduced, especially the engineering time will be significantly saved because accurate solution may be attained in the first trial. That is the goal of the automatic and adaptive time stepping method. However, if the model includes NOLIN entries, SOL 129 is likely to activate iteration for equilibrium, while SOL 109 takes one-step solution for each time step. That is because SOL 109 uses explicit approach for the NOLIN process, taking NOLIN forces at the previous time step for the equilibrium of the current time step. The iteration makes SOL 129 slower than SOL 109 when NOLIN entries are present, but the solution is more accurate. The iteration could be avoided by using large tolerances for convergence criteria.

Saving engineering time is by far more efficient even if the computation takes more time. The automatic time stepping strives for a one-shot solution for this reason, and the algorithm takes a rather conservative approach, i.e., taking a smaller stride to avoid divergence or convergence difficulties. Particularly, bisection algorithms are devised for accuracy and effectiveness, not for efficiency. If the accuracy can be compromised for efficiency, the adaptive bisections should be turned off. Also larger strides can be achieved by specifying a smaller value for MSTEP field (e.g., 10) in the TSTEPNL entry.

The following performance comparison of the analyses of a model with a lot of NOLIN entries shows the contrast in efficiency vs. strategy:

**Table 8-2**

Version	method	tsteps	usertime	systime	elapsed	Remarks
2001	fixed	3200	1787.69	1208.59	50:52	no bisections
2001	auto		1662.74	1187.90	49:20	no bisections
2004	fixed	3200	1520.04	1147.16	45:20	nolin=0, tzeromax=0
2004	fixed	37039	17567.10	10535.30	9:48:34	bisections allowed
2004	auto	4540	1434.06	1228.18	47:42	nolin=0, tzeromax=0
2004	auto	14652	11854.37	5724.78	5:02:58	nolin=1., tzeromax=1
2004	auto	1320	829.84	570.50	23:38	nolin=0., tzeromax=1, mstep=10

## Theory

### Follower Force Generation

First, the load generation should be performed in the iteration module for efficiency. Direct time integration (in NLTRD2) is to solve the equation

$$M\ddot{u} + C\dot{u} + F(u) = P(t) \quad \text{Eq. 8-1}$$

at every incremental time step. This process involves an integration in time domain combined with an iterative process at each time step to satisfy equilibrium. The iteration is based on an incremental process using

$$\left[ \frac{4}{\Delta t^2} M + \frac{2(1-\eta)}{\Delta t} C + \tilde{K} \right] \{ \Delta U^{i+1} \} = \left\{ R_{n+1}^i \right\} \quad \text{Eq. 8-2}$$

with

$$\left\{ R_{n+1}^i \right\} = \left\{ P_{n+1}^i - F_{n+1}^i \right\} + \frac{4(1-\eta)}{\Delta t} M \{ \dot{U}_n \} + (1-2\eta) \{ P_n - F_n \} \quad \text{Eq. 8-3}$$

$$- \left[ \frac{4}{\Delta t^2} M + \frac{2(1-\eta)}{\Delta t} C \right] \left\{ U_{n+1}^i - U_n \right\}$$

where  $\eta$  denotes numerical damping (PARAM, NDAMP).

Currently, the load vectors ( $P_n$ ) are generated for a constant time interval by the module TRLG, and they are interpolated or extrapolated in the NLTRD2 module adaptive to the time step size. This is not only inefficient, but also makes it difficult to adjust the time step size based on the loading history. Particularly, the follower forces are recomputed in the NLTRD2 at every iteration (i) at time  $t$  as follows:

$$P_t^i(u) = \frac{\Delta t_r}{\Delta t} \{ P_n \} + \left( 1 - \frac{\Delta t_r}{\Delta t} \right) \{ P_{n+1} \} + f(t) \cdot \left\{ P_f(u^i) - P_f(0) \right\} + N_t^i(u, u) \quad \text{Eq. 8-4}$$

where the first two terms represent a linear interpolation between two load vectors at steps  $n$  and  $n+1$  (from TRLG), the third term a correction for follower forces, and the fourth term NOLIN forces.

The new design implemented in MSC.Nastran 2004 is to generate load vectors in NLTRD2 one column at each time step, so that no update or interpolation is necessary; i.e.,

$$P_t^i(u) = f_{nf}(t) \cdot \{ P_{nf} \} + f_f(t) \cdot \left\{ P_f^i \right\} + N_t^i \quad \text{Eq. 8-5}$$

where the first term represents non-follower forces, which is to be generated in NLTRD2 (migrated from TRLG). Notice that the non-follower force vectors are formed once per time step, but the follower force vector (the second and the third terms) must be computed at every iteration. It improves not only load-dependent adaptive time stepping but also the memory usage and reduces the I/O operations.

## Load-Dependent Time Step Adjustment

The second step is to make the time step adjustment algorithm adaptive to the loading history based on the TLOADi data. The adaptive time step algorithm for TLOAD1 is implemented by considering time steps specified in the TABLEDi entries. A new time step size is compared against time interval in the TABLED(1-4) at every time step. When adjusted step exceeds the user-specified load increment interval or causes an

abrupt changes in load magnitude, the adjusted value is bisected to be consistent with the binary-based time adjustment algorithm. This process will continue sweeping through the time duration for these subcase.

The transient loads specified by TLOAD2 are analytical functions, continuous and smooth, but the excitation frequency should be considered for adaptive time stepping so that the period (field F on TLOAD2) is less than 8 times the incremental time step, i.e., bisect if the time step size is larger than 1/8 of a period.

This algorithm activates an adaptive bisection upon load spikes to account for impulsive loads with adequate number of time steps. This modifications work for both structural analysis and heat transfer analysis. Linear forces (i.e., any loads other than follower forces) have not been implemented in MSC.Nastran 2004, and yet to be included in the time step adjustment.

## Adaptive Time Stepping for NOLIN and NLRGAP

The third phase makes the time step adjustment algorithm sensitive to the NOLIN data. The NOLIN forces, denoted by  $N_i(u, \dot{u})$  in Eq. 8-4, are implicit functions of time (except for NLRGAP that could be explicit function of time), which may vary abruptly as a function of time. Then the solution may not render a correct solution with a large time step. The NOLIN entries consists of

- NOLIN1 with TABLEDi:  $N1_i = s \cdot T_j(u, \dot{u})$
- NOLIN2:  $N2_i = s \cdot X_j \cdot X_k$
- NOLIN3:  $N3_i = s \cdot X_j^A$
- NOLIN4:  $N4_i = -s \cdot [-X_j]^A$
- NLRGAP: simulates a radial forces on a journal bearing joint

The criteria for the abruptness can be as follows:

$$\Delta t_{n+1} = 0.5 \cdot \Delta t \text{ if } |N_{t+\Delta t} - N_t| > F_n \cdot \text{Max}(|N_t|, 10^{-4})$$

where the factor  $F_n$  is determined based on the numerical experiments (defaulted to 1000). This factor  $F_n$  is controlled by system cell 386 (=nolin), i.e.,

$$F_n = 1000. / nolin$$

where nolin is the alias name of the real-valued sys(386) as follows:

nolin = 0          bisection is suppressed (same as MSC.Nastran 2001)  
 nolin = 0.001      increase accuracy slightly  
 nolin = 0.01       increase accuracy a little more  
 nolin = 1.0        allow full adaptive bisection (default)

Consecutive bisections are allowed (KSTEP+1) times, in which KSTEP is defaulted to 2 in the TSTENL entry.

Until MSC.Nastran 2001, NOLIN forces were added to the right-hand side and treated like an external loads as shown in [Eq. 8-4](#). In order to make a full implementation of NOLIN forces as internal forces, they have to be subtracted from element forces, and thus NOLINs are elements having stiffness and damping. To remedy the automatic time stepping without full implementation as elements, NOLINs need to be considered like an internal force in computing a dominant frequency for a time stepping criterion. The dominant frequency should then be computed by

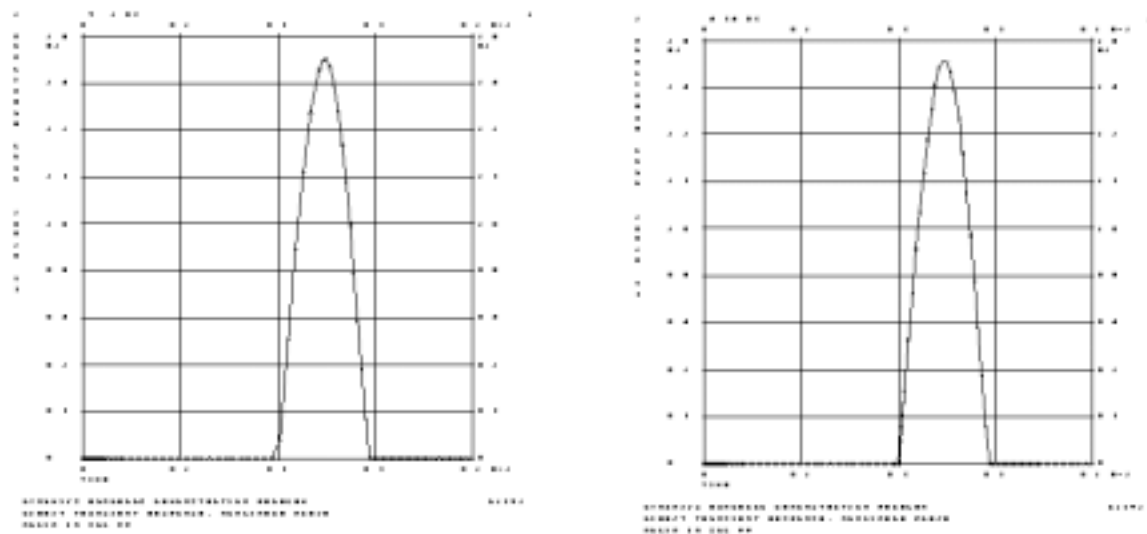
$$\omega_n^2 = \frac{\{\Delta u_n\}^T \{\Delta F_n - \Delta N_n\}}{\{\Delta u_n\}^T [M] \{\Delta u_n\}} \quad \text{Eq. 8-6}$$

where:

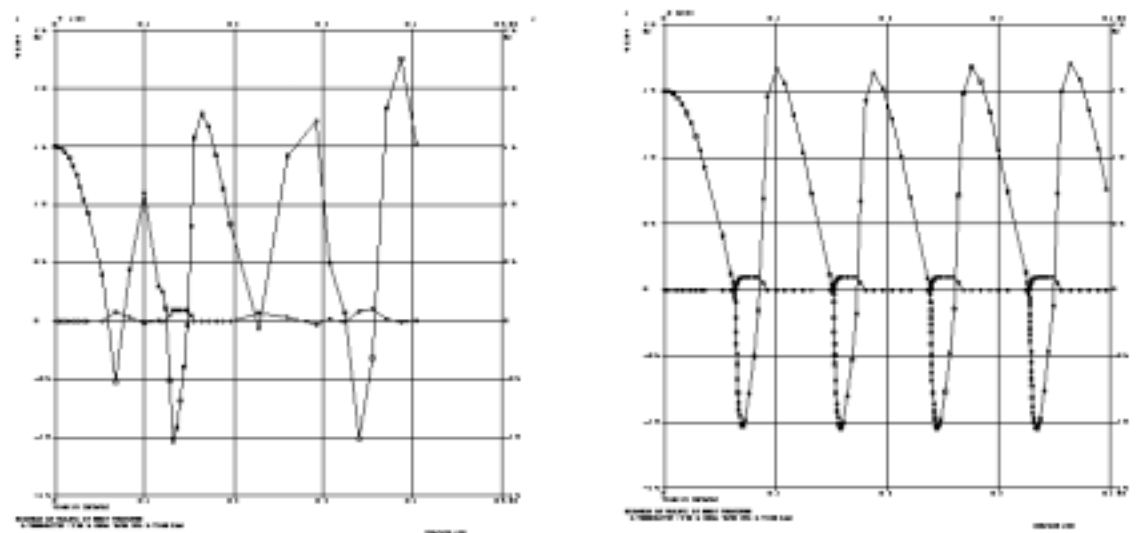
$$\Delta N_n = \{N_n\} - \{N_{n-1}\}$$

The effectiveness for NOLIN-dependent time step adjustment is demonstrated in the two examples below: a vibrating beam with a stopper simulated by NOLIN1 and a thermostat model with a NOLIN1. The result is the accuracy improvement, not efficiency, as shown in the figures. The vibrating beam experiences abrupt changes in the NOLIN1 force (contact) when the GAP opens or closes. The adaptive bisection algorithm detects the transition from open to closed or vice versa. The thermostat model has similar characteristics as a vibrating beam.





**Figure 8-1 Vibrating with a stopper (a) MSC.Nastran 2001 vs. (b) MSC.Nastran 2004**



**Figure 8-2 Thermal system with a thermostat (a) MSC.Nastran 2001 versus (b) MSC.Nastran 2004**

## Initial Time Step Adjustment

The adaptive time stepping algorithm in the NLTRD2 module starts with a user-specified value (DT field in the TSTEPNL entry). The algorithm estimates a dominant frequency in the system, from which an appropriate time step size is computed.

However, this process requires two converged time steps. If the process continues without reverting back to time zero, the solution could be erroneous for the first two steps with a large DT. This is particularly true if you specify excessively (two orders of magnitude) large DT as demonstrated in some heat transfer problems.

An initial quadri-section algorithm has been implemented in MSC.Nastran 2004 to remedy this deficiency. This initial quadri-section is repeated until a right time step size is attained, which is controlled by an integer system cell 373 called TZEROMAX, i.e.,

TZEROMAX > 0 : maximum number of times to return to time zero

TZEROMAX = 0 : no initial time step adjustment (identical to V2001)

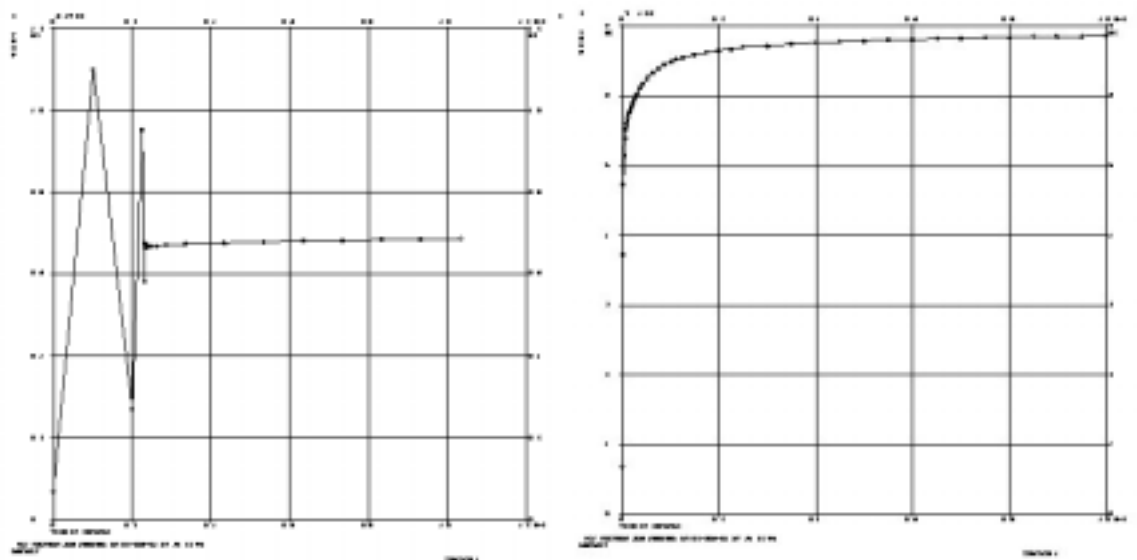
TZEROMAX < 0 : no limit on DT adjustment

with a default value of TZEROMAX=4. When the initial time step adjustment is activated, a UWM (7600) is issued as:

INITIAL TIME STEP SIZE IS REDUCED TO X.XXXE-X.

This new algorithm is applicable to both structural and heat transfer analyses, but especially effective for heat transfer analysis. It allows the adjustment of user-specified value of DT by almost three orders of magnitude by default. This initial time step adjustment algorithm accommodates the existing automatic time stepping capability as it currently works. However, the parameter MAXR (in TSTEPNL entry) has been modified to accept any real number without a bound, with a default value of 32.

An example of a thermal conduction process is analyzed with/without initial time step adjustment starting with DT=1.0e-4. The solution without initial time step adjustment shows gross error at the beginning as shown in [Figure 8-3](#). The initial time step adjustment algorithm adjust DT to 1.56e-6 to render the second solution.



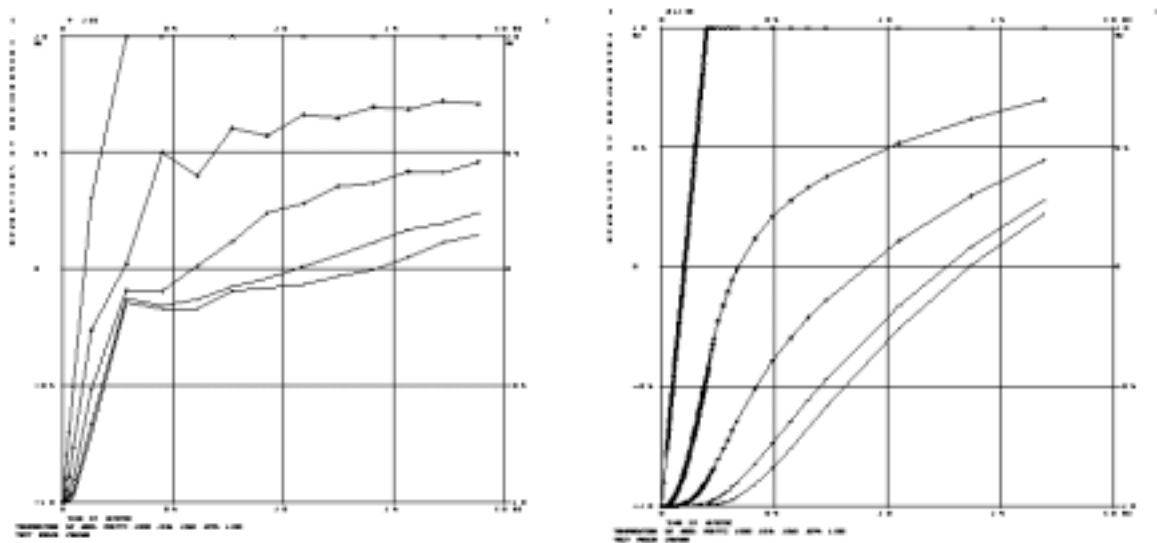
**Figure 8-3** temperature as function of (a) MSC.Nastran 2001 versus (b) MSC.Nastran 2004

## Enhancements For Automatic Time Stepping Due to Error Corrections

Since the automatic time stepping and adaptive bisection/doubling were introduced in Version 67, some errors were identified and/or introduced. Extensive cleaning up on the time stepping routines improved automatic time stepping, adaptive bisection, and doubling scheme. These enhancements resulted in improved accuracy and efficiency. Some analyses show more accurate solution, while some converges now, which diverged before. In order to measure the efficiency, the total number of time steps taken for each Subcase is printed at the end of the Subcase along with LOOPID.

During the enhancement project, an error in PLOADX1 follower force computation has been corrected. The most prominent error discovered was in the double-precision table lookup. The double-precision table lookup is used in NOLIN processes, some of which exhibited erratic numerical behavior that could not be traced because the anomaly shows up after thousands of time steps. The anomaly is manifested by consecutive bisections that make the time step tiny in the middle of the run. The new double-precision table lookup routine is deemed to have corrected this erratic behavior. Also an error in the temperature-dependent thermal conductivity (MATT4) for transient heat transfer analysis has been corrected. In addition, a new PARAM, SEKD has been introduced in the DMAP to control the differential stiffness generation for superelements.

In [Figure 8-4](#), the example shows dramatic improvement in the solution for a phase transformation process, melting a solid.



**Figure 8-4 Phase Transformation (a) MSC.Nastran 2001 versus (b) MSC.Nastran 2004**

## 8.4 Buckling Analysis

### Introduction

An accurate solution to a buckling problem requires more meticulous efforts than just following a numerical procedure. There are a number of factors to consider before a buckling solution can be accepted with confidence. Such points along with a few examples are discussed by comparing solutions obtained from linear buckling analysis with preloads and nonlinear buckling with/without unsymmetric stiffness matrix.

### Linear Buckling Analysis with Preloads

A starting step for a nonlinear buckling analysis should be a linear buckling analysis. The linear buckling analysis capability in SOL 105 used two subcases until Version 70.5: the first subcase for a linear static analysis with a buckling load, followed by the second subcase for an eigenvalue analysis. However, there may be other static loads sustained by the structure, in addition to the buckling load in question. In order to provide more versatile capability for general buckling cases, a new capability was introduced in MSC.Nastran Version 70.7, allowing constant preloads in the linear buckling analysis.

### Theory

Let us start with a linear equation of motion for a preloaded structure, i.e.,

$$M\ddot{u} + C\dot{u} + Ku + K^d u = P(t) \quad \text{Eq. 8-7}$$

where  $M$ ,  $C$ ,  $K$  and  $K^d$  are mass, viscous damping, material and differential stiffness matrices, and  $P(t)$  is a forcing function in time domain. The differential stiffness is created by the initial stress due to preloads, and it may include the follower stiffness if applicable. Ignoring the damping term to avoid complex arithmetic, an eigenvalue problem may be formulated as

$$[(K + K^d) - \omega^2 M]\{\phi\} = \{0\} \quad \text{Eq. 8-8}$$

which is a governing equation for normal mode analysis with a preload, but it could be recast for a dynamic buckling analysis at a constant frequency. For a static buckling, that constant frequency of vibration is zero and the inertia term drops out, i.e.,

$$[K + K^d]\{\phi\} = \{0\} \quad \text{Eq. 8-9}$$

in which  $\{\phi\}$  represents virtual displacements, or buckling modes shapes. A non-trivial solution exists for an eigenvalue that makes the determinant of  $[K + K^d]$  vanish, which leads to an eigenvalue problem

$$[K + \lambda K^d]\{\phi\} = \{0\} \quad \text{Eq. 8-10}$$

where  $\lambda$  is an eigenvalue which is a multiplier to the applied load to attain a critical buckling load. If there exists constant preloads other than the buckling load in question, the above equation should include additional differential stiffness, i.e.,

$$[K + K_{preload}^d + \lambda K_{buckle}]\{\phi\} = \{0\} \quad \text{Eq. 8-11}$$

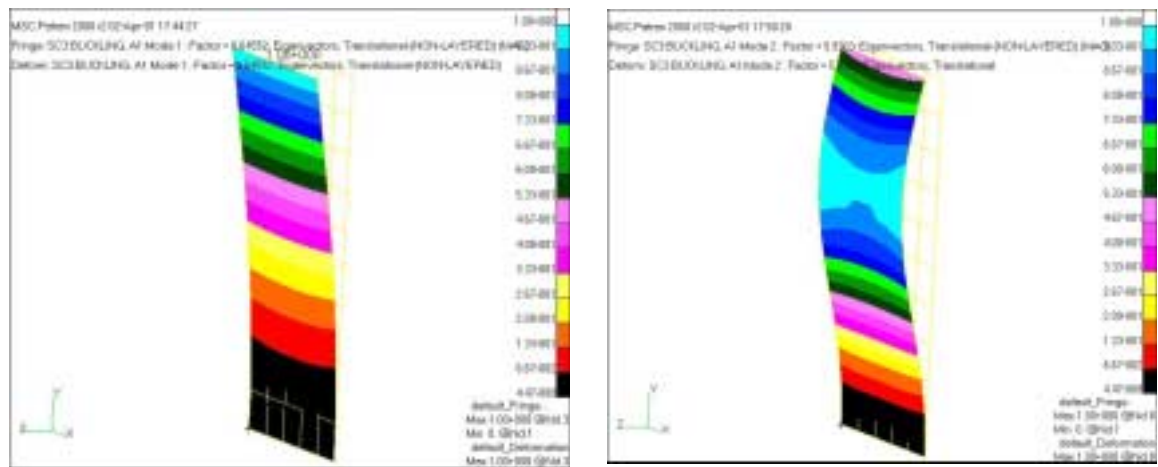
in which differential stiffness is distinguished for constant preload and variable buckling load. Notice that no eigenvalue solutions are meaningful if the preload makes the structure buckle, i.e.,  $[K + K_{preload}^d]$  should be positive definite. The buckling analysis with an excessive preload can render wrong solutions, unless the positive definiteness is ensured.

## Example - prld\_buck.dat

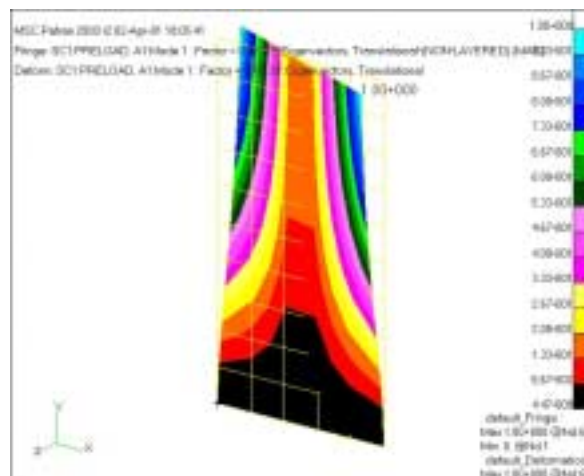
An example of input data for a linear buckling analysis with a preload is shown below:

```
ID plate105, v2001 $   SHL 2/23/01
DIAG 8, 15 $
SOL 105
TIME 60
CEND
TITLE = MSC.Nastran job created on 20-Feb-01
      SPC = 2   $ clamp the bottom edge
      DISP=ALL
SUBCASE 10
      SUBTITLE= preload tensile edge force 300 lb/in
      LOAD = 2
SUBCASE 20
      Subtitle= top pressure 100 lb/in.
      LOAD = 7
SUBCASE 30
      SUBTITLE=buckling analysis
      STATSUB(preload)=10
      STATSUB(buckle)=20
      METHOD=1
BEGIN BULK
PARAM      POST      0
EIGRL      1         0.         6.         3         0         1.
.
.
.
ENDDATA
```

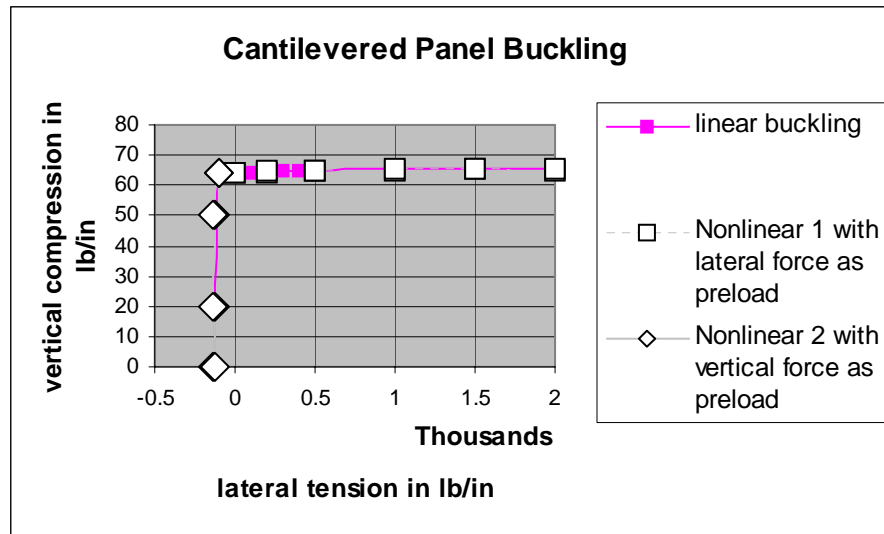
This is an example of a plate model (5 in. by 10 in., 0.1 in thick, steel, clamped on the bottom edge) with lateral tension by an edge load (in x) of 300 lb/in for a preload and vertical edge load (in y) of 100 lb/in for a buckling load. Alternatively, the preload and the buckling load may be switched to perform a similar analysis. The first buckling mode is a bending mode at vertical load of -64.55 lb/in, as shown in **Figure 8-5**. The second buckling mode in bending at -593 lb/in is also shown in **Figure 8-5**, but it is of no physical significance and shown here for verification purposes only. If the lateral load is in compression instead of tension, the plate could buckle in a twist mode at -123.29 lb/in in the absence of a vertical load, as shown in **Figure 8-6**. A parametric study, conducted with various combinations of lateral and vertical loads, resulted in a buckling envelope (or locus) on a loading plane as shown in **Figure 8-7**. The buckling shape is a bending mode above the curve and a twist mode in the left side of the curve. Bifurcation behavior can be observed in the transitional region near the corner point, where either the twist or bending mode could appear as the first and the second mode. Lower-right side of the curve is the safety zone.



**Figure 8-5 Panel Buckling Shapes for the First Two Bending Modes**



**Figure 8-6 Panel Buckling in Twist Mode Under Lateral Compression**



**Figure 8-7 Cantilevered Panel Buckling Envelope**

## Nonlinear Buckling Analysis with Symmetric Stiffness Matrix

Nonlinear buckling analysis capability has been available in SOL 106 by restarts using PARAM, BUCKLE, 1. While serving the purpose of a nonlinear buckling analysis following a static nonlinear analysis, this buckling analysis procedure is cumbersome to the user because it requires a restart. To provide a more convenient and versatile capability for nonlinear buckling analysis, an option for a buckling analysis in a cold-start run with PARAM, BUCKLE, 2 was introduced in MSC.Nastran Version 70.7. This option allows a buckling analysis in any Subcase that has the METHOD command specified for an eigenvalue analysis.

There are two more ways in nonlinear analysis to estimate the critical buckling load. One method is the arc-length method that can provide solutions past the critical buckling load into the post-buckling state. Using the arc-length method, the nodal displacement of a point with the most noticeable motion should be traced to illustrate the peak point. The applied load at this peak resembles the critical buckling load. Another method is to use Newton's method until the solution cannot be obtained due to divergence, in which case the adaptive bisection method is activated in the vicinity of the critical buckling load and stops at the limit load, very close to the critical buckling point. A similar method was applied to the nonlinear dynamic stability analysis using adaptive bisection algorithm in time domain for buckling or bifurcation prediction in the implicit direct-time integration.



The nonlinear buckling analysis procedure in MSC.Nastran allows buckling analysis with preloads. However, the nonlinear procedure follows a different numerical procedure. Therefore, it could provide a means for validation of linear solutions. A parametric study using nonlinear buckling analysis rendered data points for the panel buckling problem, labeled "Nonlinear 1 and 2" in [Figure 8-7](#), using the lateral force and vertical force as preloads, respectively. Although two different methods have been used in the nonlinear buckling prediction: namely, (1) eigenvalue estimation by extrapolation and (2) detection of the critical load by consecutive bisections with no recourse due to divergence, all the nonlinear solution points fell, by and large, on the same curve. This figure also shows that the problem is basically linear, because the "Large Displacement" option introduced no visible effects.

## Nonlinear Buckling Analysis with Unsymmetric Stiffness Matrix

Follower forces usually create unsymmetric follower stiffness unless the follower forces are completely enclosed, exerting zero resultant forces. This asymmetry condition should be addressed with caution. The follower stiffness in SOL 106 was introduced in Version 70.7, where symmetrization was applied conservatively so that it is activated in rare occasions. The logic was modified in MSC.Nastran 2001 to always symmetrize the follower stiffness by default. Some problems exhibited convergence difficulties as a drawback of symmetrizing follower stiffness. The symmetrization does not show adverse effects until the follower stiffness effect becomes dominant under large displacements. In MSC.Nastran 2004, the symmetrization is modified to be activated adaptively, detecting the severity of the asymmetry.

For adaptive symmetrization of the follower stiffness, we need a criterion to determine the extent of asymmetry. This can be achieved by normalizing the norm of unsymmetric part of the follower stiffness by a stiffness matrix that is independent from follower stiffness. The differential stiffness was chosen for this purpose after some numerical experiments. Then, the ratio of asymmetry is compared with a tolerance, which is represented by a user parameter FKSYMFAC so that the user can control the process. The default value for FKSYMFAC is determined to be 0.024 by a parametric study, allowing symmetrization of the follower stiffness during the stiffness update process if the iterative process can render converged solutions. If the asymmetry ratio is greater than FKSYMFAC, symmetrization process will not be activated. The asymmetry ratio increases as the geometric nonlinearity intensifies, which is confirmed by numerous test runs.

When the follower stiffness becomes significantly asymmetric, the real eigenvalue analysis would be invalid or erroneous. The eigenvalue analysis with unsymmetric matrices requires complex arithmetic with left and right-handed eigenvectors. The complex eigenvalue analysis capability is added to SOL 106 in MSC.Nastran 2004 with some modifications to complex Hessenberg and Lanczos methods for unsymmetric buckling analysis. To use the complex eigenvalue analysis, you must specify CMETHOD instead of METHOD in the Subcase and the CMETHOD should reference EIGC Bulk Data entry. The eigenvalues obtained from the complex eigenvalue analysis for buckling with asymmetric stiffness are complex numbers, of which imaginary numbers become zeros. All the eigenvectors are also real numbers. By comparing results from both real and complex eigenvalue analysis, it is confirmed that the real eigenvalue analysis renders wrong solutions to unsymmetric stiffness for nonlinear buckling. Particularly, the mode shapes are erroneous with real eigenvalue analysis from unsymmetric problem, whereas the discrepancy in the critical buckling load is not pronounced.

The buckling shape should be examined graphically to fully understand the physical phenomenon. Fortunately, the complex eigenvalue analysis produces real eigenvectors from real and unsymmetric matrices. However, the buckling shapes displayed with eigenvectors currently are with reference to the undeformed shape, and do not visualize the real buckling shape if the deformed shape at the onset of the buckling has very large displacements.

## Example

An example problem is a cantilevered beam model subject to an end force that is a follower force, i.e., FORCE1. The end points rotates 180 degrees by making a complete U shape at the load factor of 2.8 before coming to the shape of deformation shown in [Figure 8-8](#) at a load factor 4. The eigenvalue analysis is performed at load factor 4, where the asymmetry ratio is 0.069, which produced buckling shapes shown in [Figure 8-9](#) (obtained by complex method) and [Figure 8-10](#) (obtained by real eigenvalue method). They are different and the real eigenvalue method is likely to be in error. The display tool should be improved so that the buckling shapes be plotted with an eigenvector as a perturbation with reference to the deformed shape to fully perceive the physics.

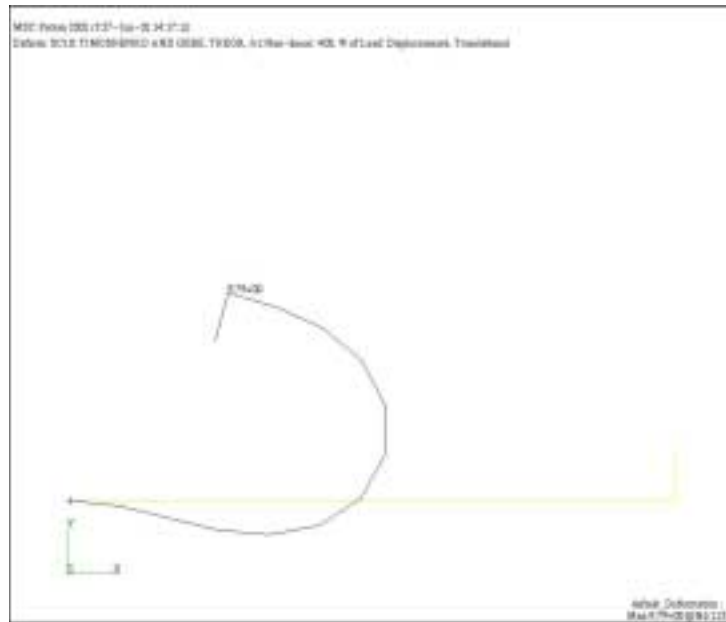


Figure 8-8 Deformed shape at load factor 4 ( $F_1=771.2$ )

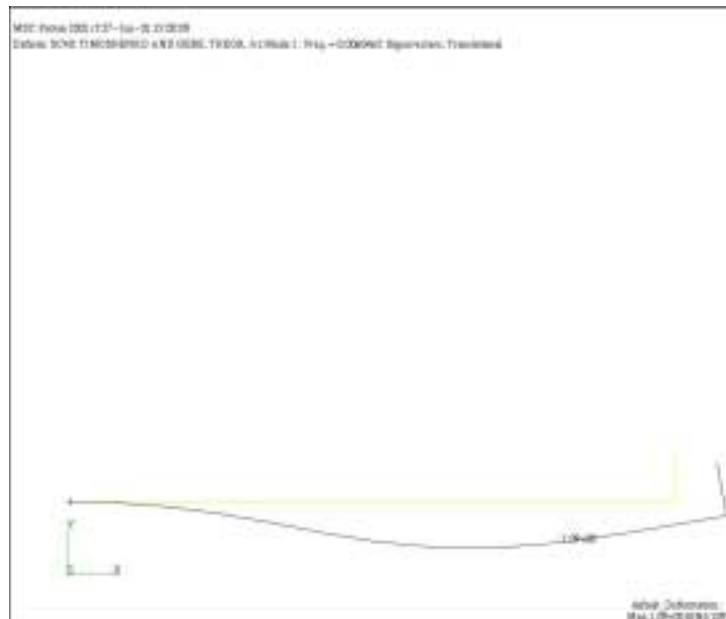
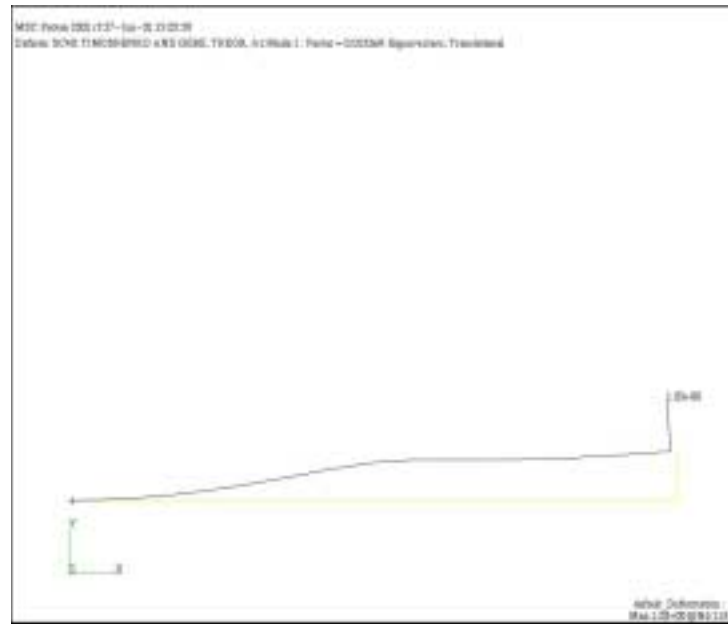


Figure 8-9 Buckling shape by complex method at L.F.=4.



**Figure 8-10 Buckling shape by real eigenvalue analysis at L.F.=4.**

## An Example of Cylinder Buckling

A hollow cylinder is used to demonstrate an interesting behavior in buckling. Its geometric attributes include: 50 in. in diameter, 100 in. in height and 0.125 in. wall thickness with closed ends of 0.5 in. in thickness. The cylinder, made of steel, is subjected to a combined load of external pressure on the cylindrical surface and vertical edge force on the top, while the bottom is fixed in  $z$ .

The cylinder could buckle in two different modes depending on the combination of loads as shown in [Figure 8-11](#) and [Figure 8-12](#), where the first mode with vertical creases appear when the external pressure is predominant (showing a star-shaped cross-section at the mid-span) and the second mode with horizontal creases appear when the vertical edge force is more dominant. A failure curve (or buckling envelope) can be constructed by a parametric study using various magnitudes of the preloads with external pressure or vertical edge load, as shown in [Figure 8-13](#). The first buckling shape appears above the curve and the second buckling in the right side of the curve. A bifurcation occurs at the corner, where either buckling mode may occur. The cylinder is free from buckling in the lower-left region of the curve. It is noted that the buckling resistance against the vertical load does not increase even if the pressure direction is reversed to an internal pressure (creating a tensile hoop stress), although the buckling load increases against the external pressure as the vertical compression decreases.

Nonlinear buckling solution points are added to the curve rendered by linear buckling solutions, where three categories of nonlinear solutions are presented. Solution points in square symbol are from the eigenvalue analysis using an extrapolation scheme, and those in diamond symbol are from the limit process using consecutive bisections upon divergence during the incremental/iterative processes. Although the second buckling loads in the right side of the curve predicted by nonlinear analyses are almost identical to the linear solutions, nonlinear solutions for the first buckling shape above the curve are somewhat dispersed. In the absence of an exact solution for a reference, these nonlinear solutions seem plausible to indicate that the geometric nonlinearity has significant effects on the buckling of the first kind with vertical creases, while the geometric nonlinear effect is negligible in the second kind. These solutions, however, were obtained with symmetrized stiffness matrix. The solution points in triangular symbol show tremendously different trend in the upper side of the curve, where the buckling is caused by the external pressure.

The pressure load creates follower stiffness, which is unsymmetric in this problem. The solutions in triangular symbol were obtained with unsymmetric matrices. By preserving asymmetric stiffness matrices, the solutions converged further and the predicted buckling loads were higher. The buckling analysis with asymmetric matrices (but real-valued) requires complex eigenvalue analysis, although the solution turns out to be real-valued. Furthermore, an ideal cylinder without the end condition does not buckle under pressure because of the follower stiffness effect. That is why the solutions with asymmetric stiffness converge to much higher external pressure, and the complex eigenvalue analyses with extrapolation at the last converged steps render solution points shown by triangles. Solutions may continue for increased pressure and it has been observed that the predicted buckling point increases as the eigenvalue analysis is performed at higher pressure. Two of the buckling modes of these solutions are shown in [Figure 8-14](#) and [Figure 8-15](#), which are rather spurious and fortuitous. However, the deformed shape of the second kind of buckling as shown in [Figure 8-16](#) was obtained by a limit process using adaptive bisections upon divergence, which depicts a genuine buckling shape. It also signifies that the nonlinear eigenvalue analysis based on extrapolation is an approximation and it can vary appreciably depending on where the extrapolation is applied.

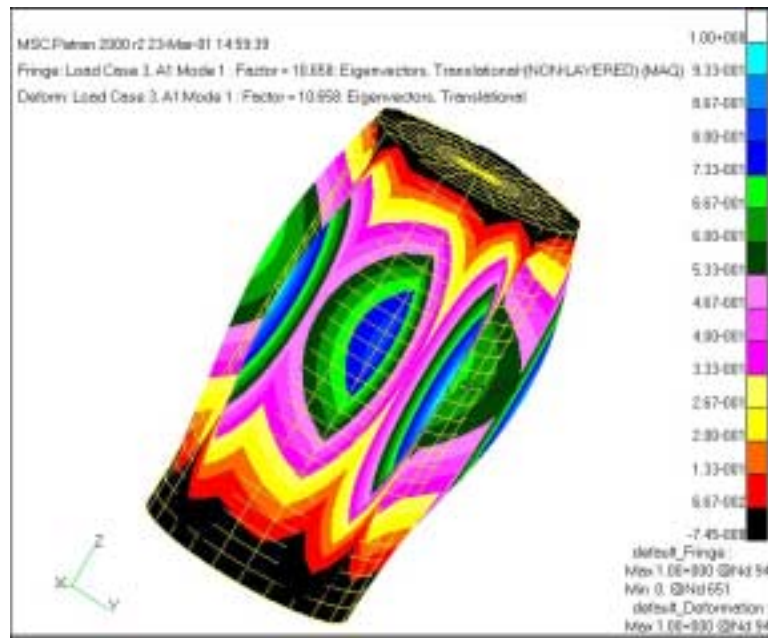


Figure 8-11 Cylinder buckling shape 1 with vertical creases.

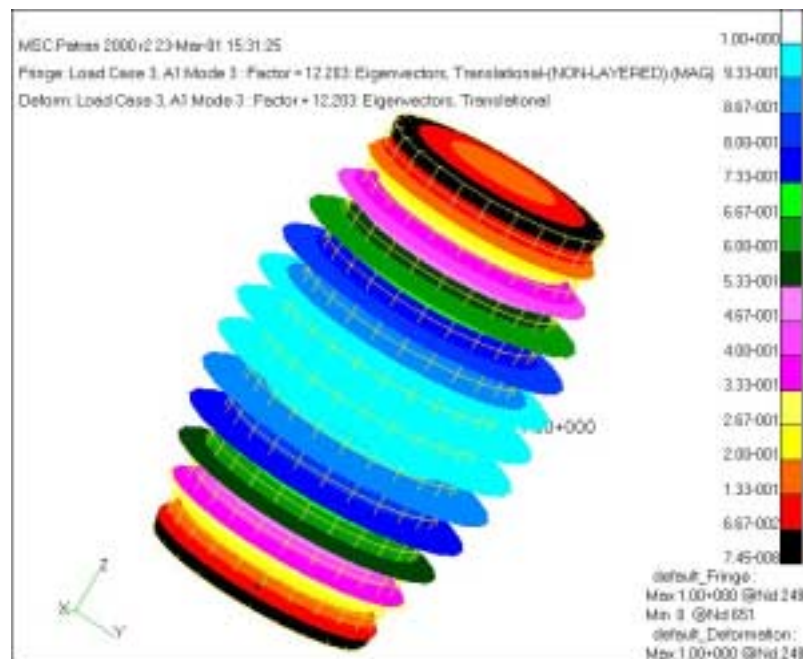
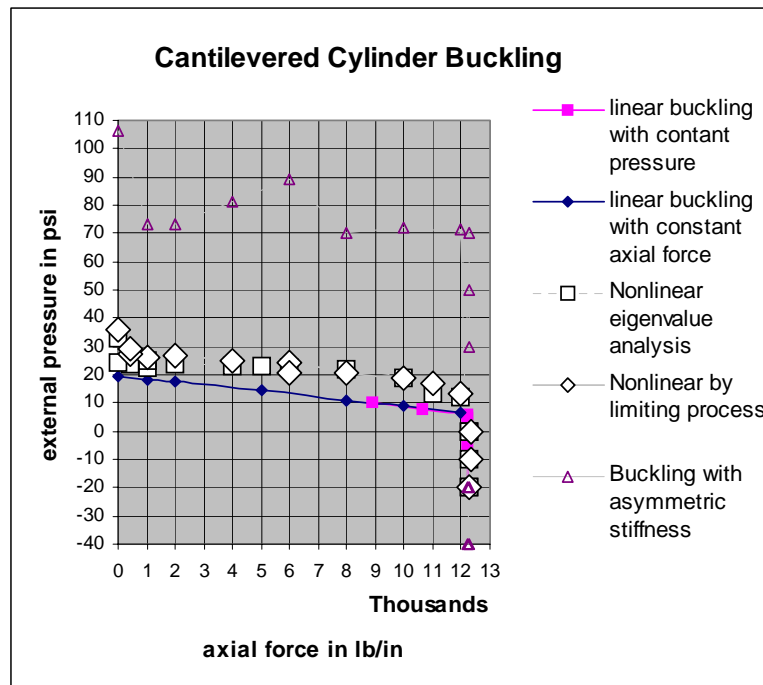
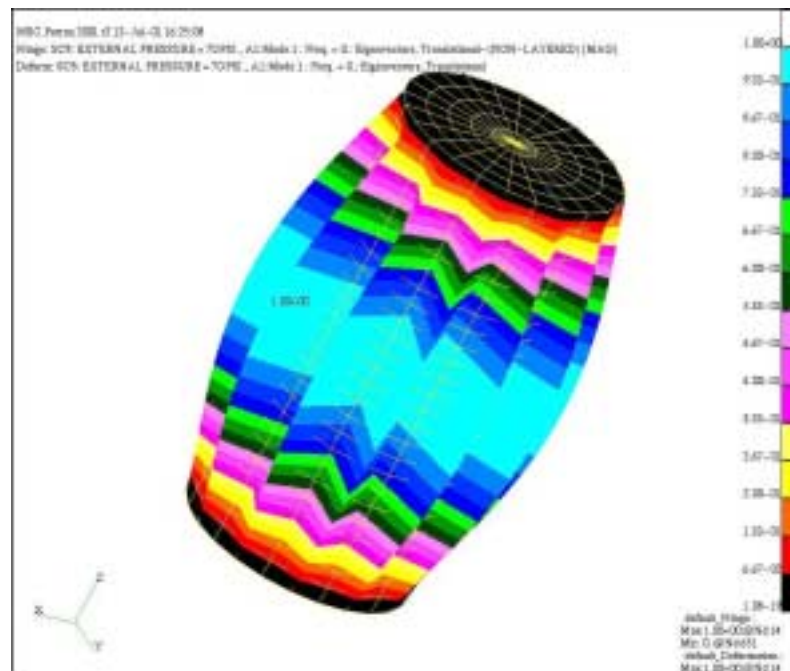


Figure 8-12 Cylinder buckling shape 2 with horizontal creases.

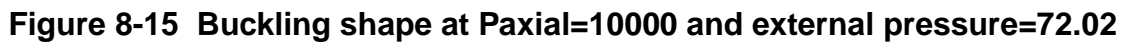


**Figure 8-13 Cylinder buckling envelope (with nonlinear solutions)**



**Figure 8-14 Buckling shape at Paxial=2000 and external pressure=73.44**







CHAPTER

9

# Numerical Enhancements

- Introduction to Parallel Numeric Enhancements
- Domain Decomposition
- New Distributed Memory Parallel Methods
- Inputs
- Method Selection Guidelines
- Limitations
- Examples
- Parallel Performance Data
- Other Numeric Enhancements

## 9.1 Introduction to Parallel Numeric Enhancements

Shared Memory Parallel (SMP) processing was first introduced in MSC.Nastran in 1987. Shared memory architectures feature multiple processors, each processor sharing a common memory and disk I/O. SMP is invoked by adding “PARALLEL=n” to the nastran submittal command, where “n” is the number of desired processors.

With the rise in popularity of distributed network computing, a new parallel paradigm was needed to accommodate the distributed computing environment. In this environment, each processor has its own memory and disk I/O, and communication is done through a network. The MSC.Nastran Distributed Memory Parallel (DMP) method addresses this environment.

MSC.Nastran DMP achieves parallelism by dividing the problem to be solved at a high level. Division of the problem into sub-problems may be performed in different ways. For example, problems posed in the frequency domain may be divided based naturally on frequency. A suitable and effective domain decomposition serves as the cornerstone of MSC.Nastran DMP parallel performance.

Previous versions of MSC.Nastran provided domain decomposition techniques in the frequency and geometric domains. Frequency domain decomposition allows a problem to be divided into frequency sub-ranges and solved independently in each sub-range. Geometric domain decomposition seeks to divide the model geometrically, based on grid point connectivity, into sub-models. Each sub-model is then solved independently, followed by a solution at the geometric boundary defined by the domain decomposition.

MSC.Nastran 2004 provides a third domain decomposition option, degree-of-Freedom (DOF) domain decomposition, which provides for the division of the problem at the degree of freedom level, after single- and multi-point constraints have been eliminated. Decomposition of the problem at the DOF level minimizes some of the potential complications that may arise with domain decomposition at the grid point level, and is a more appropriate decomposition choice in some cases. See “[Domain Decomposition](#)” on page 403 for further discussion on the various domain decomposition techniques, and “[New Distributed Memory Parallel Methods](#)” on page 404 for an introduction to modal analysis techniques based on DOF domain decomposition.

MSC.Nastran 2004 also allows, for the first time, a combination of frequency and geometric domain decomposition techniques in modal analysis. This combination, called the Hierarchic Method, first divides the problem geometrically at the grid point level. Then during the eigenvalue analysis, each sub-model is further divided in the frequency domain, yielding an additional level of parallelism.

Also for the first time, DMP is supported in SOL 200 design optimization. DMP methods are available during the analysis phase of the design cycle. DMP methods for normal modes (ANALYSIS=MODES), modal dynamic analyses (ANALYSIS=MFREQ, MTRAN, MCEIG, and FLUTTER), and direct frequency response (ANALYSIS=DFREQ) are supported. Please see “[New Distributed Memory Parallel Methods](#)” on page 404 for more details.

The DOMAINSOLVER command was revised and extended in order to handle new DMP developments discussed in this chapter. Please see “[Inputs](#)” on page 407 for a complete description.

“[Method Selection Guidelines](#)” on page 411, helps the user select the appropriate strategy for solving large problems with DMP. Limitations that preclude the use of DMP are presented in “[Limitations](#)” on page 413. Finally, “[Examples](#)” on page 415 show how to execute DMP strategies with the appropriate commands and submit procedures.

**Table 9-1** contains a list of computers and corresponding operating systems that support DMP. The system must also have Message Passing Interface (MPI) software installed. MPI software provides the interface for communication between the distributed processors. DMP analyses must be performed on the same type of machines. **Table 9-1** summarizes the DMP system requirement.

**Table 9-1 DMP System Prerequisites**

Platform	System Requirements	
AIX	Processor OS MPI	Any AIX 4.3.2 POE 2.3
Compaq Alpha UNIX	Processor OS MPI	Any UNIX 4.0f, 5.0 Compaq MPI 1.8.1
HP	Processor OS MPI	PA-RISC 2.0 HP-UX 11.x, 64-bit <i>No additional software is required.</i>

**Table 9-1 DMP System Prerequisites**

Platform	System Requirements	
SGI	Processor OS MPI	R8K, R10K, R12K, R14K, R16K IRIX 6.5 and maintenance releases MPT 1.4 and later
Linux	Processor OS MPI	IA32 2.2.17 or later LAM 6.5.6
Linux	Processor OS MPI	IA64 2.4.3 or later LAM 6.5.6
SUN	Processor OS MPI	SPARC64 Solaris 8.0 mpi2
NEC	Processor OS MPI	SX-6 SUPER-UX 11.1 MPI/SX 5.0
Cray	Processor OS MPI	SV1 Unicos 10.0 MPT 1.4.0.0
Fujitsu	Processor OS MPI	Fujitsu vector processor UXP/V, UNIX V5.4 with vector capability enhancement mpi2

## 9.2 Domain Decomposition

### Introduction

Decomposition of a problem in the frequency domain is a natural and intuitive way to divide a problem into smaller pieces. Frequency domain decomposition is available in MSC.Nastran for the following analysis types: normal modes analysis, modal dynamic analysis, and direct frequency response (SOLs 103, 111 and 112, and 108, respectively). In addition, it is available in the analysis portion of SOL 200 (design optimization). For 103, 111, and 112, the Lanczos eigenvalue extraction method will allow for frequency domain decomposition based on the number of available processors. In 108 and 111, the frequency response calculations are also subject to domain decomposition. The frequency domain offers straightforward parallelism since results at every frequency are mathematically independent. Generally, the number of frequency domains will equal the number of processors devoted to the problem. This method is known as Frequency Domain Decomposition.

Another intuitive way to subdivide a problem is to physically divide the problem into discrete parts, or components. In MSC.Nastran, the geometric domain is considered to be the collection of grid and scalar points, and their connectivity via elastic and rigid elements. Geometric domain decomposition is the product of an automatic partitioning algorithm based on the “nested dissection” of the model into smaller and smaller parts. Nested dissection is in turn a product of general graph theory. The product of a geometric domain decomposition is a collection of  $N$  discrete components, plus a common geometric boundary. The  $N$  components are independent and may be processed in parallel. In MSC.Nastran, this is accomplished utilizing superelement technology. For most DMP methods,  $N$  equals the number of available processors. Geometric domain decomposition is available in SOLs 101, 103, 111, and 112.

In MSC.Nastran 2004, a third domain decomposition method (DOF based Domain Decomposition) has been added.

## 9.3 New Distributed Memory Parallel Methods

### Introduction

MSC.Nastran 2004 contains two new DMP features: a Hierarchic distributed parallel Lanczos method, and a DOF based domain decomposition. Starting with MSC.Nastran 70.7, various Distributed Memory Parallel (DMP) solution techniques have been implemented. Direct frequency response (SOL 108) and real eigenvalue analysis (SOL 103) in the frequency domain, and static analysis (SOL 101) in the geometric domain, were implemented in Version 70.7. This was followed in MSC.Nastran 2001 by normal modes analysis (SOL 103) and Automated Component Mode Synthesis (ACMS) in the geometric domain.

### Hierarchic Distributed Parallel Lanczos Method

This new method combines two existing principles of domain decomposition in the distributed parallel normal modes analysis: frequency and geometry domains. The hierarchic approach still solves the global eigenvalue problem exactly, as opposed to the component modal reduction method, which is an approximate approach.

The new hierarchic normal modes analysis method has two layers. First, in the outer layer of the hierarchy, the global finite element matrices are automatically partitioned and each processor sees only a local partition. In the inner layer of the hierarchy, the frequency range is again automatically subdivided and processors that are working on the same geometry partition work on different frequency segments.

More specifically, in the new method a subset of processors solves the same global problem in a given frequency interval. Another subset solves the problem in another interval. Within one subset, however, the processors work on their local geometry and communicate with the others on the boundary. Each processor of a given subset computes a (geometrically) local section of the global eigenvectors of the subset's frequency interval and communicates it with its partners in the subset. The process concludes with the final collection of the global mode shapes of these intervals on a master processor to facilitate follow-on modal analysis techniques or appropriate postprocessing.

### Inputs

The user interface is identical to that of the geometric domain parallel method. However, but to invoke the hierarchic method, the number of domains specified must be the square root of the number of processors requested. For example, to execute a hierarchic Lanczos process with 2 geometric partitions and 2 frequency segments simultaneously, a submittal command similar to the one shown below must be used:

```
nastran jobname dmp=4 ...
```

Furthermore, the following executive control statement must also be used.

```
DOMAINSOLVER MODES (NUMDOM=2, PARTOPT=GRID)
```

The PARTOPT keyword specifies the method of partitioning; “GRID” specifies the grid based geometric domain, which is equivalent to the MSC.Nastran 2001 “GDMODES” method. For complete information on the updated DOMAINSOLVER command, please see “[Inputs](#)” on page 407.

## DOF Based Domain Decomposition

The Degree of Freedom (DOF) domain is defined as the degree of freedom set left over after all constraints have been eliminated from the finite element problem. DMP methods based on DOF domain decomposition, postpone the partitioning of the global problem until after all constraints have been removed. For example, in a conventional real eigenvalue analysis, DOF domain decomposition is performed on the F-set (the KFF and MFF matrices).

A domain decomposition is performed on the degrees of freedom in this set, and the associated connectivity of these degrees of freedom. The partitioning algorithm is based on a multilevel nested dissection, which yields a multilevel elimination tree. The domains created are all naturally independent, and so may be processed by multiple processors in parallel. A common boundary is also generated, that is solved with the aid of message passing interface (MPI) communication.

The purposes for introducing a new parallel domain are twofold. Generally, it is a sound strategy to offer more than one way to accomplish any significant task, such as domain decomposition. Secondly, it is a convenient means of avoiding potential limitations encountered during the grid point based geometric domain decomposition. The MSC.Nastran geometric domain decomposition relies on superelement technology to drive the solution process. In some cases, this technology limits the ways problems can be posed. For example, models that contain duplicate element IDs cannot be processed with MSC.Nastran superelements. This may occur when a rigid element is added to a model whose ID is the same as an existing QUAD4 element, for example. The DOF domain thus offers more flexibility to provide a robust DMP solution.

In MSC.Nastran 2004, DOF domain decomposition is available in SOL 101 and SOL 103 and in the normal modes analysis portion of SOLs 111, 112, and 200.

## User Interface

To run a SOL 103 real eigenvalue analysis with the distributed Lanczos method, using DOF domain decomposition, a submittal command similar to the following can be used:

```
nastran jobname dmp=N ...
```

Furthermore, the following DOMAINSOLVER Executive Command Statement must be added.

```
DOMAINSOLVER MODES (PARTOPT=DOF)
```

where “N” is the number of available processors.



9.4 Inputs

The user interface for Distributed Memory Parallel (DMP) MSC.Nastran is comprised of a combination of the “dmp” keyword on the submittal line and the DOMAINSOLVER Executive statement. The DOMAINSOLVER command has been changed from MSC.Nastran 2001 such that many of the old keywords are no longer supported.

The format of the DOMAINSOLVER Executive Statement is as follows:

DOMAINSOLVER  $\begin{bmatrix} \text{STAT} \\ \text{MODES} \\ \text{FREQ} \\ \text{ACMS} \end{bmatrix} \left[ \left( \text{PARTOPT} = \begin{bmatrix} \text{DOF} \\ \text{GRID} \\ \text{FREQ} \end{bmatrix}, \text{NUMDOM} = \text{int}, \text{UPFACT} = \text{real}, \right. \right. \\ \left. \left. \text{TREE} = \begin{bmatrix} \text{SINGLE} \\ \text{MULTI} \end{bmatrix}, \text{ALLOC} = \begin{bmatrix} \text{STATIC} \\ \text{DYNAMIC} \end{bmatrix}, \text{PRINT} = \begin{bmatrix} \text{YES} \\ \text{NO} \end{bmatrix} \right) \right]$

Examples:

DOMAINSOLVER STAT (PARTOPT=DOF)

DOMAINSOLVER ACMS (UPFACT=1.5,NUMDOM=128)

For DOMAINSOLVER ACMS, there are additional parameters UPFACT, ALLOC, and PRINT. The keywords function as follows:

STAT	Linear Statics
MODES	Normal Modes
FREQ	Frequency Response
ACMS	Automated Component Modal Synthesis

The description of the parameters is as follows:

PARTOPT	Partitioning option. Selects which domain is to be decomposed.
DOF	Degree of Freedom domain
GRID	Grid point (Geometric) domain
FREQ	Frequency domain
The default is dependent upon solution sequence. See Table 9-3 for further descriptions.	

NUMDOM	Selects the number of domains as follows:
ACMS	<p>If NUMDOM = 0 or 1, then the model will not be split. The Default depends on the model size (number of grid points) as follows:</p> <p>NGRID <math>\leq</math> 2,000; NUMDOM = 4</p> <p>2,000 <math>\leq</math> NGRIDS &lt; 10,000; NUMDOM = 16</p> <p>10,000 <math>\leq</math> NGRIDS &lt; 50,000; NUMDOM = 32</p> <p>50,000 <math>\leq</math> NGRIDS &lt; 100,000; NUMDOM = 64</p> <p>100,000 <math>\leq</math> NGRIDS &lt; 300,000; NUMDOM = 128</p> <p>300,000 <math>\geq</math> NGRIDS; NUMDOM = 256</p>
The model will be split into NUMDOM domains.	
STAT	Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (= no. of processors used for the run). The model will be divided into NUMDOM domains in either the geometric (grid based) or DOF domains, depending on the value of PARTOPT.
MODES	Default = <i>dmp</i> ; if NUMDOM has any other value, the Hierarchical method will be used. For the hierarchical method, NUMDOM must be the square root of <i>dmp</i> . Otherwise, it NUMDOM will be reset to <i>dmp</i> (= no. of processors used for the run). The model will be divided into NUMDOM domains in either the geometric (grid based) or DOF domains, depending on the value of PARTOPT.
FREQ	Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (= no. of processors used for the run). The frequency range will be divided into NUMDOM regions that are then solved independently.
UPFACT (ACMS)	By default, the frequency range used for upstream component modes is two times larger than the desired range on the EIGR/L entry. To modify this factor, specify the UPFACT parameter (real; default=2.0).
TREE (ACMS)	<p>Specifies the type of elimination tree to use:</p> <p>TREE = MULTI: binary multilevel tree (default)</p> <p>TREE = SINGLE: single level tree</p> <p>A multilevel tree requires that the number of domains be a power of two and be evenly divisible by <i>dmp</i>, the number of processors specified for the run; otherwise a single level tree will be used automatically.</p>

ALLOC	Determines how superelements are assigned to processors:
(ACMS)	<p>ALLOC = STATIC: components are divided evenly at job start (default)</p> <p>ALLOC = DYNAMIC: components are assigned to processors on a first come, first served basis. This option is effective when one or more processors is heavily loaded and/or has a slower computational rate.</p>
PRINT	Controls intermediate print of upstream and data recovery processing in F06 and F04 files. Default='NO'. If PRINT=NO and an error occurs upstream, the intermediate output is placed in a separate output file named "jid.acms_out" for examination.
(ACMS)	

**Table 9-2** shows the availability of partitioning methods with each analysis type. An asterisk (“\*”) indicates a supported implementation.

**Table 9-2 Analysis Types and Partitioning Methods**

Solution Sequence	DMP Method	Partitioning Methods Available				
		GEOM	DOF	FREQ	G+F	D+F
101	STAT	*	*			
103	MODES	*	*	*	*	
	ACMS	*				
108	FREQ			*		
111	MODES		*	*		*
	ACMS	*		*	*	
	FREQ			*		
112	MODES		*	*		
	ACMS	*				
200	MODES		*	*		*
	FREQ			*		

The DOMAINSOLVER command is optional. If “dmp=” is specified on the command without a DOMAINSOLVER command in the Executive Section, the following actions will result based on solution sequence.

Table 9-3 DOMAINSOLVER Defaults

Solution Number	Default DOMAINSOLVER Options	
	DMP Method	Partitioning Option
101	STAT	GRID
103	MODES	GRID
108	FREQ	FREQ
111	MODES	FREQ
	FREQ	FREQ
112	MODES	FREQ
200	MODES	DOF
	FREQ	FREQ

## 9.5 Method Selection Guidelines

The modeling and analysis of real world phenomena is a complicated process. No single numerical approach satisfies every situation. Multiple solutions, each more or less tailored for a particular modeling scenario, computer architecture, or analysis type, are available in MSC.Nastran. Only the largest and most challenging problems are suitable for Distributed Memory Parallel (DMP) methods. This section provides guidelines for the most productive use of DMP. It is organized by finite element analysis type.

### Static Analysis

- Models larger than 20,000 grid points may show parallel speedup using DMP, with greater benefit for larger problems.
- Nontrivial models with dozens or hundreds of load cases would exhibit speedup.
- Large solid models would also benefit from the MSC.Nastran iterative solution, particularly if there are only a few load cases.

### Normal Modes Analysis

- For large problems in SOL 103 with only a few modes desired, the geometric domain parallel Lanczos may be preferred.
- For problems in SOL 103 with a large number of modes desired:
  - Frequency domain decomposition will be effective if the modes are distributed fairly evenly throughout the frequency range. This is especially true if the computer architecture is truly distributed, i.e. each processor has its own memory and scratch disk space.
  - For very large problems ACMS is effective.
- If disk space is limited, any of the geometric domain decomposition methods (parallel Lanczos, Hierarchical, ACMS) will help minimize scratch disk space requirement.

### Modal Dynamic Analysis (SOLs 111 and 112)

In modal dynamic analysis, two levels of parallelism are possible, one in the eigensolution, and the other in the dynamic analysis. Parallelization of the frequency response module is automatically activated in SOL 111. Parallelization of the transient response module in SOL 112 is not supported at this time.

- For large cases over wide frequency ranges, with multiple dynamic loads, ACMS is the preferred DMP method.
- The DOF domain decomposition method is generally effective.
- Frequency domain decomposition in the real eigensolution will be effective if the modes are fairly evenly distributed throughout the frequency range. This is especially true if the computer architecture is truly distributed, i.e. each processor has its own memory and scratch disk space.

## **Direct Frequency Response (SOL 108)**

- If sufficient resources are available to solve the physical problem on each processor, DMP is a very effective way to provide parallel speedup in direct frequency response (SOL 108).

## 9.6 Limitations

This section lists limitations and unsupported features in DMP.

### **The following limitations apply to all DMP methods:**

1. ASSIGN statements are supported only in a truly distributed environment, and if separate scratch disks are assigned to each machine for the DMP run. The limitation does not apply to USER FORTRAN files (e.g., output2, output4, xdb, etc.)
2. P-elements are not supported.

### **Following is a list of additional limitations that are present only for the ACMS method:**

1. In acoustic analysis, MPCs or rigid elements cannot be connected between the fluid and the structure.
2. ACMS is available for SOL 200 in serial only. In addition, ACMS efficiency in SOL 200 may be inhibited by large numbers of design variables.

### **Following is a list of additional limitations that apply only to the MODES method:**

1. Acoustic analysis (fluid-structure interaction) is not supported for the geometric domain. It is supported in the DOF domain (PARTOPT=DOF).
2. Disjoint models are not allowed.
3. Scalar Points are not supported.
4. Virtual mass (MFLUID) is supported. However, if the partitioner assigns the virtual mass to the boundary, System Fatal Message 7795 will be issued. In this case, switch to the ACMS method, or use frequency domain decomposition.
5. Residual vectors are not available with geometric domain decomposition (PARTOPT=GRID).

### **Following is a list of additional limitations that apply only to static analysis (SOL 101):**

1. Disjoint models are not allowed.
2. Scalar points are not supported.
3. Virtual mass (MFLUID) is not supported.

**The following limitations apply to the Geometric Domain option:**

1. Duplicate element IDs are not supported.
2. Restarts are not supported.
3. Dummy elements (CDUM8, CDUM9) are not supported.
4. User-defined superelements are not supported.
5. Large numbers of dependent degrees of freedom on MPCs or rigid elements may result in uneven partitions that may increase runtime.
6. Acoustic analysis in SOL 111 (fluid-structure interaction) is supported only in the ACMS method.
7. Results from the grid point weight generator are not correct.

**There are no known limitations for the Frequency Domain option.**



## 9.7 Examples

MSC.Nastran DMP requires the use of the `dmp` keyword on the `nastran` submittal line. In addition, the `DOMAINSOLVER` executive statement is required in most cases, unless the default options are desired. Below is a series of examples with the `nastran` submittal line in italics.

### Example 1 – SOL 101

The following SOL 101 example uses geometric domain decomposition with four processors. The domain decomposition algorithm divides the model into four domains that will be solved independently, followed by the distributed boundary solution.

```
nastran jobname dmp=4 ...  
  
SOL 101  
DOMAINSOLVER STAT (PARTOPT=GRID)  
CEND  
...  
ENDDATA
```

This is equivalent to:

```
SOL 101  
DOMAINSOLVER STAT  
CEND  
...  
ENDDATA
```

and is equivalent to:

```
SOL 101  
CEND  
...  
ENDDATA
```

This problem will be divided into four components, plus a common boundary, based on grid point (geometric) domain decomposition. Each processor will operate on its local domain independently in parallel. Solution at the boundary proceeds via inter-process communication. `GRID` is the default `PARTOPT` for the `STAT` method, so it is not necessary to specify it explicitly. The `DOMAINSOLVER` statement is not needed for SOL 101, because it is the default.

## Example 2 – SOL 103 with Frequency Domain Decomposition

Run a normal modes analysis using frequency domain decomposition on two processors.

```
nastran jobname dmp=2 ...

SOL 103
DOMAINSOLVER MODES (PARTOPT=FREQ)
CEND
...
ENDDATA
```

In this example, the frequency range specified on the EIGR/L Bulk Data entry will be divided into two sub-regions. Each processor will solve the full eigenvalue problem for its own local frequency domain. Results are merged on the master processor and the analysis continues.

## Example 3 – SOL 103 with Geometric Domain Decomposition

Run a normal modes analysis using geometric domain decomposition on four processors.

```
nastran jobname dmp=4 ...

SOL 103
DOMAINSOLVER MODES (PARTOPT=GRID)
CEND
...
ENDDATA
```

This is equivalent to:

```
SOL 103
DOMAINSOLVER MODES
CEND
...
ENDDATA
```

and is equivalent to:

```
SOL 103
CEND
...
ENDDATA
```

This problem will be divided into four components, plus a common boundary, based on grid point (geometric) domain decomposition. Each processor will perform a complete eigenvalue analysis on the full frequency range for its local domain.

Solution at the boundary follows. Then, the results are merged on the master processor and the analysis continues. Because GRID is the default PARTOPT for the MODES method, it is not necessary to specify it explicitly. The DOMAINSOLVER statement is not needed for SOL 103 because MODES is the default describer.

## Example 4 – SOL 103 with the Hierarchical Method

This example executes a normal modes analysis using the Hierarchical combination DMP method of geometric and frequency domain decomposition with 16 processors.

```
nastran jobname dmp=16 ...
```

```
SOL 103
DOMAINSOLVER MODES (PARTOPT=GRID, NUMDOM=4)
CEND
...
ENDDATA
```

This is equivalent to:

```
SOL 103
DOMAINSOLVER MODES (NUMDOM=4)
CEND
...
ENDDATA
```

In this example, the model is divided into four components using geometric domain decomposition (the “outer layer” of the domain hierarchy). During the eigenvalue analysis (in the READ module), each of the four components is further divided in the frequency domain into four frequency segments. Each frequency segment is solved independently within its geometric domain.

## Example 5 – SOL 111 with ACMS

Run a modal frequency analysis (SOL 111) using ACMS with four processors.

```
nastran jobname dmp=4 ...
```

```
SOL 111
DOMAINSOLVER ACMS (PARTOPT=GRID)
CEND
...
ENDDATA
```

This is equivalent to:

```
SOL 111
DOMAINSOLVER ACMS
CEND
...
ENDDATA
```

This is equivalent to:

```
SOL 111
DOMAINSOLVER ACMS FREQ(NUMDOM=4)
CEND
...
ENDDATA
```

In this example, the model will be divided in the geometric (grid point) domain into a default number of components, based on the size of the model (see the DOMAINSOLVER command description for details). A binary, multilevel superelement tree will be constructed and a component modal synthesis will be executed in parallel, with the tree logically divided among the four processors. For the frequency response portion of the run, the frequency range will be divided into four segments, with each segment computed independently, in parallel. (This is the default action.) Because geometric (grid point) domain decomposition is the default partitioning option, it need not be specified on the DOMAINSOLVER statement.

## Example 6 – SOL 108

Run a direct frequency response analysis (SOL 108) with eight processors.

```
nastran jobname dmp=8 ...

SOL 108
DOMAINSOLVER FREQ (PARTOPT=FREQ)
CEND
...
ENDDATA
```

This is equivalent to:

```
SOL 108
DOMAINSOLVER FREQ
CEND
...
ENDDATA
```

and is equivalent to:

```
SOL 108
CEND
...
ENDDATA
```

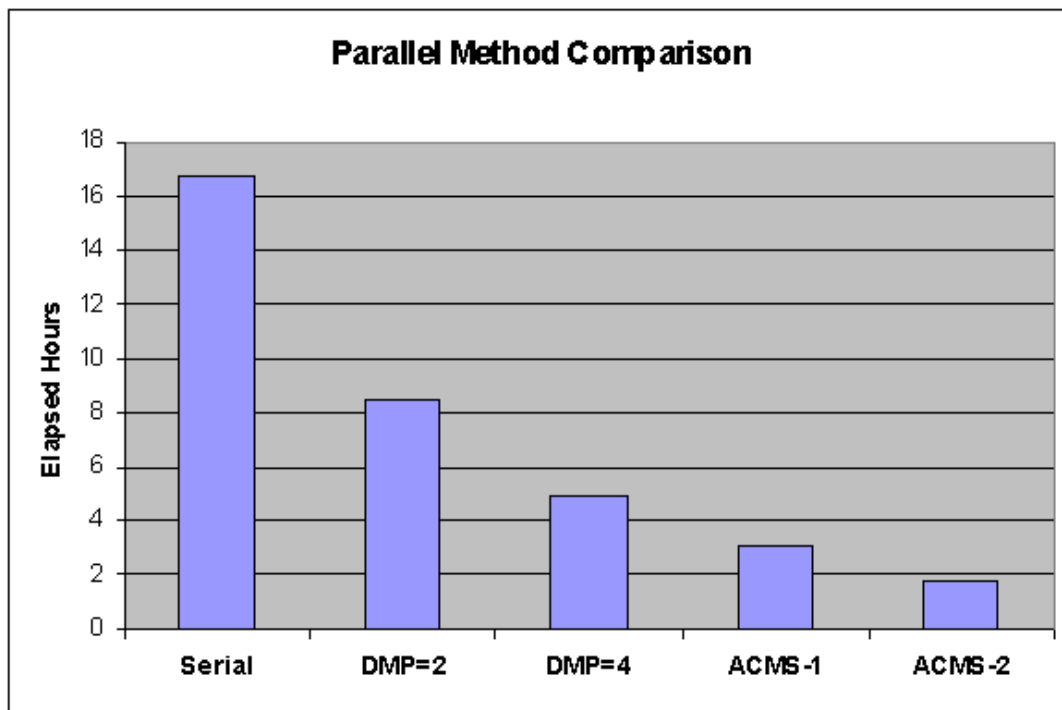
In this example, the frequency list is logically divided among eight processors, and each processor proceeds independently in parallel. Results from each processor are merged onto the master processor, and the analysis continues. Because FREQ is the

default PARTOPT for the FREQ method, it is not necessary to specify it explicitly. The DOMAINSOLVER statement is not needed for SOL 108 since FREQ is the default describer.

## 9.8 Parallel Performance Data

Parallel performance data for an automotive model is covered in this section. This is a SOL 103 run with a model consisting of 1.8 million DOFs and calculating 996 modes. The system configuration is shown below.

- Linux cluster
  - Four nodes, each with two 2.0 GHz P4 CPUs.
  - 250 Gb striped scratch disk per node (85 mb/sec I/O).
  - 3 Gb main memory per node, 100 M/s interconnect.



## 9.9 Other Numeric Enhancements

Two other numeric enhancements have been added to MSC.Nastran 2004. They are Sparse Cholesky Solution and MPYAD Method 1 Storage 3 Option.

### Sparse Cholesky Solution

The DCMP and FBS modules are enhanced to take advantage of the sparse Cholesky factor.

- DCMP modifications were introduced to create a special sparse Cholesky factor. The factor trailer identifier is form=14. The creation of the Cholesky factor may be requested by CHOLSKY input parameter to the DCMP module. This new method is selected by setting system cell 385 = 1 on the command line (sys385=1), in the rcfile (nastran system (385) = 1), or as a nastran statement in the input file. This is advisable for increased performance during large normal modes analyses using the Householder method.
- FBS module was modified to take advantage of the special sparse Cholesky factor. No user intervention is required as the form=14 of the special factor trailer triggers the Cholesky solution.

### New MPYAD Method 1 Storage 3 Option

A new storage option has been added to the matrix multiply and add module MPYAD. Method 1 Storage 3 is selected under the following conditions:

- The non null portion of the A matrix is at least 50% dense
- The non null columns of A fit in available memory
- At least two columns of B and D fit in memory
- All matrices are real

The non null portion of A is multiplied with the non null portion of B using level-3 BLAS (Basic Linear Algebra Subroutine) kernels.

The new option will be automatically selected based on time estimates. The selection or deselection of the new option may be accomplished by nastran system cell system(66). The identifiers associated with the new option are:

Method	System Cell Value
Method 1, Storage 3, NT	System(66) = 67108864
Method 1, Storage 3, T	System(66) = 134217728

---

**Note:** The new storage 3 option only supports the real matrix case.

---



CHAPTER  
**10**

---

## Heat Transfer

- Advanced Sorting Routines for VIEW Factor Shadowing Algorithm
- Axisymmetric View Factors for Gray-Diffuse Surface Character
- Thermostat Control with Deadband Applied to a Heat Source

## 10.1 Advanced Sorting Routines for VIEW Factor Shadowing Algorithm

### Introduction

Advanced 3D interference surface sorting routines have been implemented in MSC.Nastran 2004 for the VIEW Factor Shadowing algorithm. Starting with Version 68, MSC.Nastran introduced an adaptive 3D view factor process for the determination of gray-diffuse view factors between surface elements. With the advent of CAD automatic mesh generation schemes, the number of conduction elements and hence the number of radiation surface elements has, on average, increased dramatically. For complex geometries with extensive third body shadowing effects, this can lead to significant computation times.

### Benefits

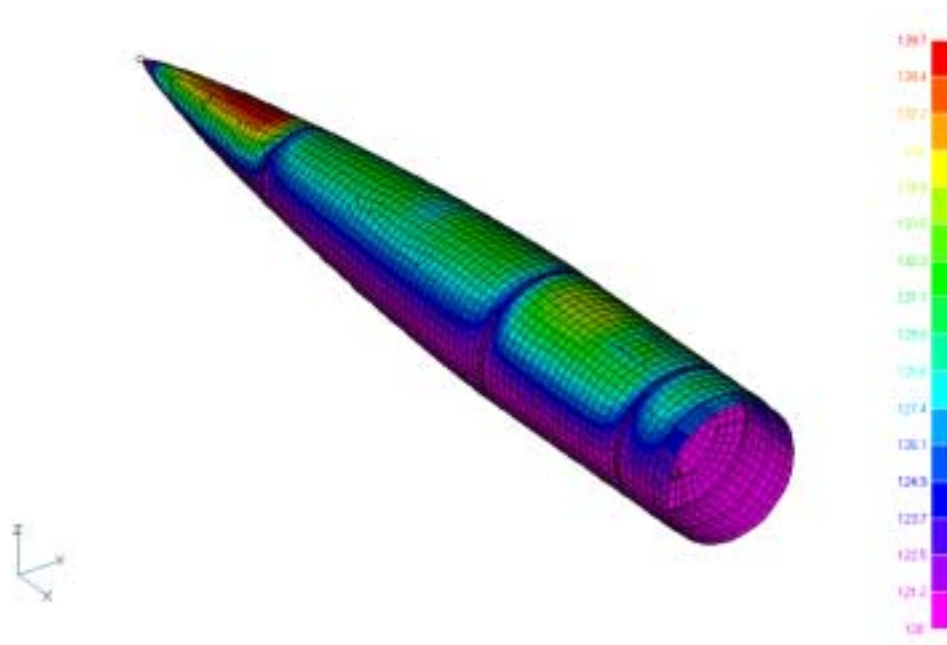
The new advanced sorting routines reduce the total computation time by several hundred percent. An example of the time savings is illustrated in the table below. Four separate models (shown on the following pages) were run with and without the new sorting routines to evaluate time savings.

**Table 10-1 Performance Comparisons**

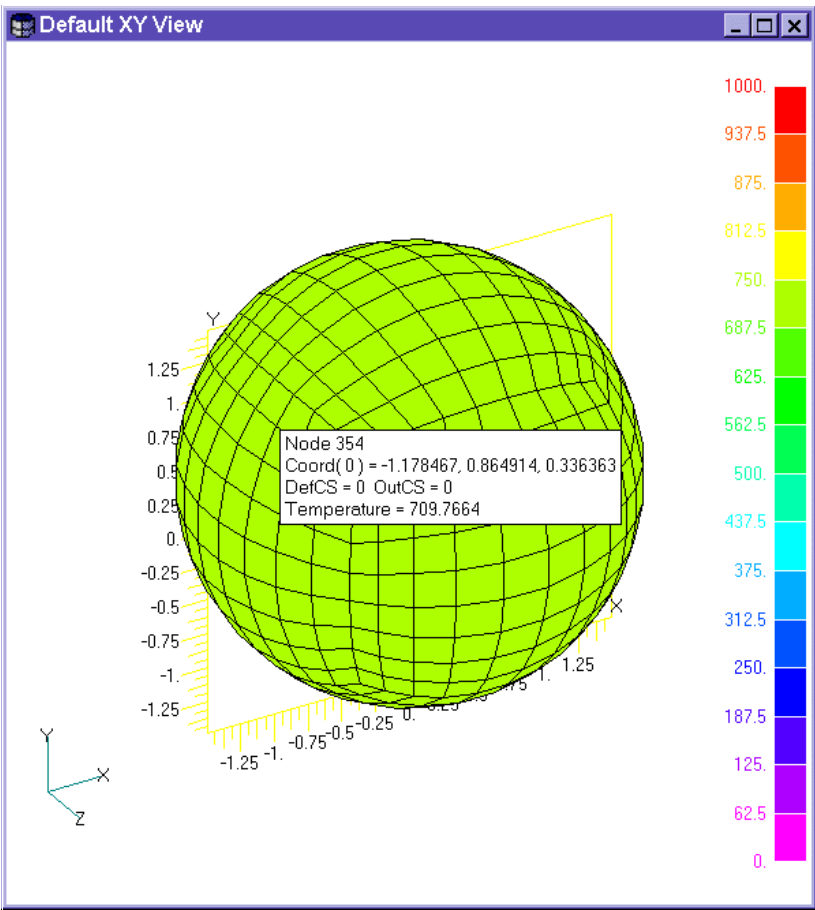
Model Parameters	No. of Rad. Surfs.	Old VIEW3D Time	New VIEW3D Time	Time Spent <u>Old VIEW3D</u> New VIEW3D
Model 1	3953	26734.3	2425.2	11.02
Model 2	702	557.6	239.2	2.33
Model 3	1440	3430.9	601.91	5.7
Model 4 (not shown)	41183	99109.5	30889.8	3.2

These jobs were run on an IBM AIX machine requesting 40 MW of memory.

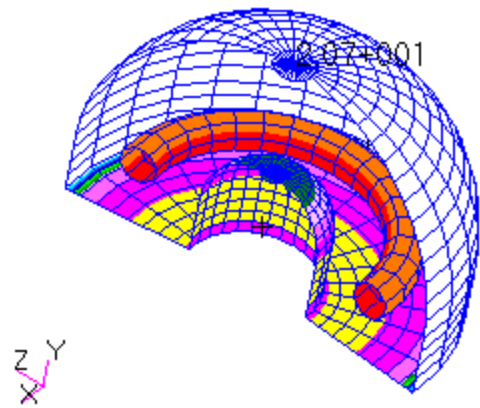
## Model 1: Ballistic Head



Model 2: Concentric Sphere



Model 3: Quartz Lamp



For large models with radiation enclosure exchange, the improvements in view factor calculation performance due to shadowing are typically in the order of a 2 to 3 times reduction in computation time. It is still important to pre-specify whenever possible, any known surface class designations that can shade or be shaded, and to utilize multiple cavity topology to minimize shadowing evaluation.

## 10.2 Axisymmetric View Factors for Gray-Diffuse Surface Character

### Introduction

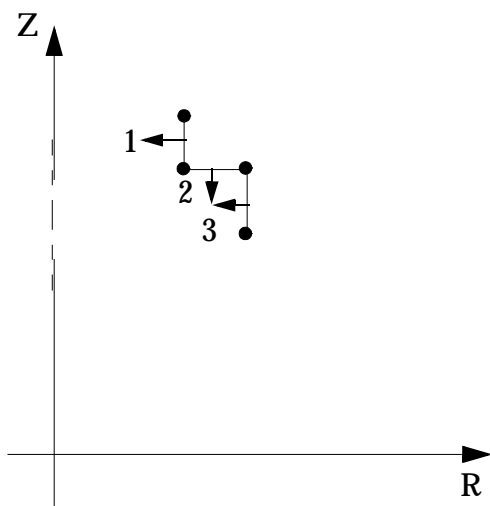
The adaptive view factor module in MSC.Nastran has been enhanced to compute view factors for axisymmetric geometries. The general 3D adaptive methodology forms the basis of this calculation along with various symmetry and shadowing arguments. Control over multiple cavities, self and third-body shadowing, shadowing flags, error estimation, and radiation exchange matrix configuration has been extended to axisymmetric analysis.

### Benefits

Surface to surface radiant exchange can be included using CHBDYi of TYPE = REV within axisymmetric models. Previously, this analysis could only be accomplished with a full three-dimensional model. Using an axisymmetric formulation of the problem will save considerable engineering modeling and calculation time in determining the view factors, running the thermal analysis, and performing a thermal-stress analysis.

### Theory

The axisymmetric geometry with respect to radiation view factors has several clear demarcations relative to the general 3D calculation. In 3D analysis,  $F_{ii} = 0$ , in axisymmetric configurations, every REV element whose normal points toward the z-axis, represents a ring that "sees" itself, and therefore  $F_{ii} \neq 0$ . This leads to the next subtlety concerning third body shadowing. In 3D analysis, independent cavities and shadowing flags are relatively easy to identify if the conditions exists. In axisymmetric analysis, such distinctions are quickly blurred. Consider the following simple cases:



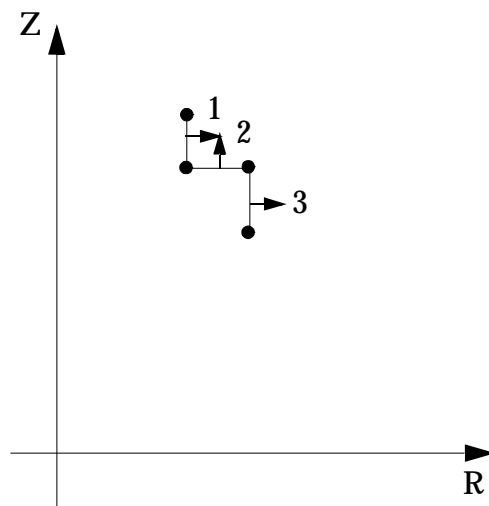
where:

$$F_{1-1} \neq 0$$

$$F_{3-3} \neq 0$$

$$F_{3-1} \neq 0$$

$$F_{1-3} \neq 0$$



where:

$$F_{1-1} = 0$$

$$F_{3-3} = 0$$

$$F_{3-1} = 0$$

$$F_{1-3} = 0$$

In light of this, it is probably safest to set SHADE = BOTH on the VIEW Bulk Data entry. Of course, calculation time can be reduced if demarcation can be accurately made.

## Example

### Model Description

Concentric Spherical shells (ball1\_vftoz\_64.dat)

We have a concentric sphere, the inner radius is at  $R=1.0$ , and the outer radius is at  $R=1.5$ . The thickness of the shell is 0.05.

The thermal boundary conditions are listed below:

1. The inner edge of the inside spherical cap is held constant at 1000 degree K
2. There is view factor exchange between the inside sphere to outside sphere
3. The emissivity for the inner sphere is 0.9
4. The emissivity for the inside of the outer sphere is 0.7
5. Ultimately, the energy from the outer sphere loses energy to cold space at 0 degree K with view factor=1.0, and emissivity = 1.0

6. The sigma-boltzmann constant is  $3.66\text{e-}11 \text{ watt/inch}^2\text{K}^{**4}$
7. The material is made of aluminum, where  $k=4 \text{ watt/inch.K}$

## Results/Description

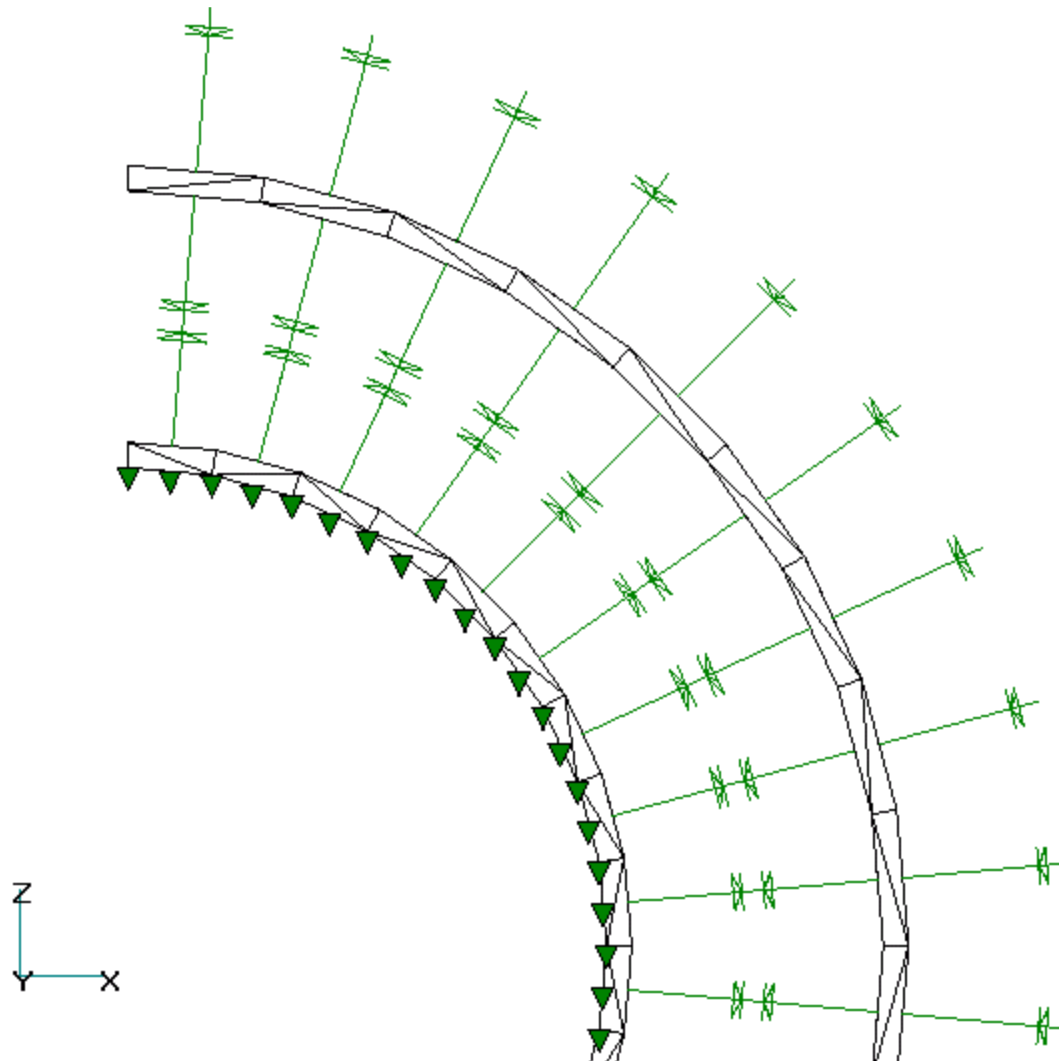
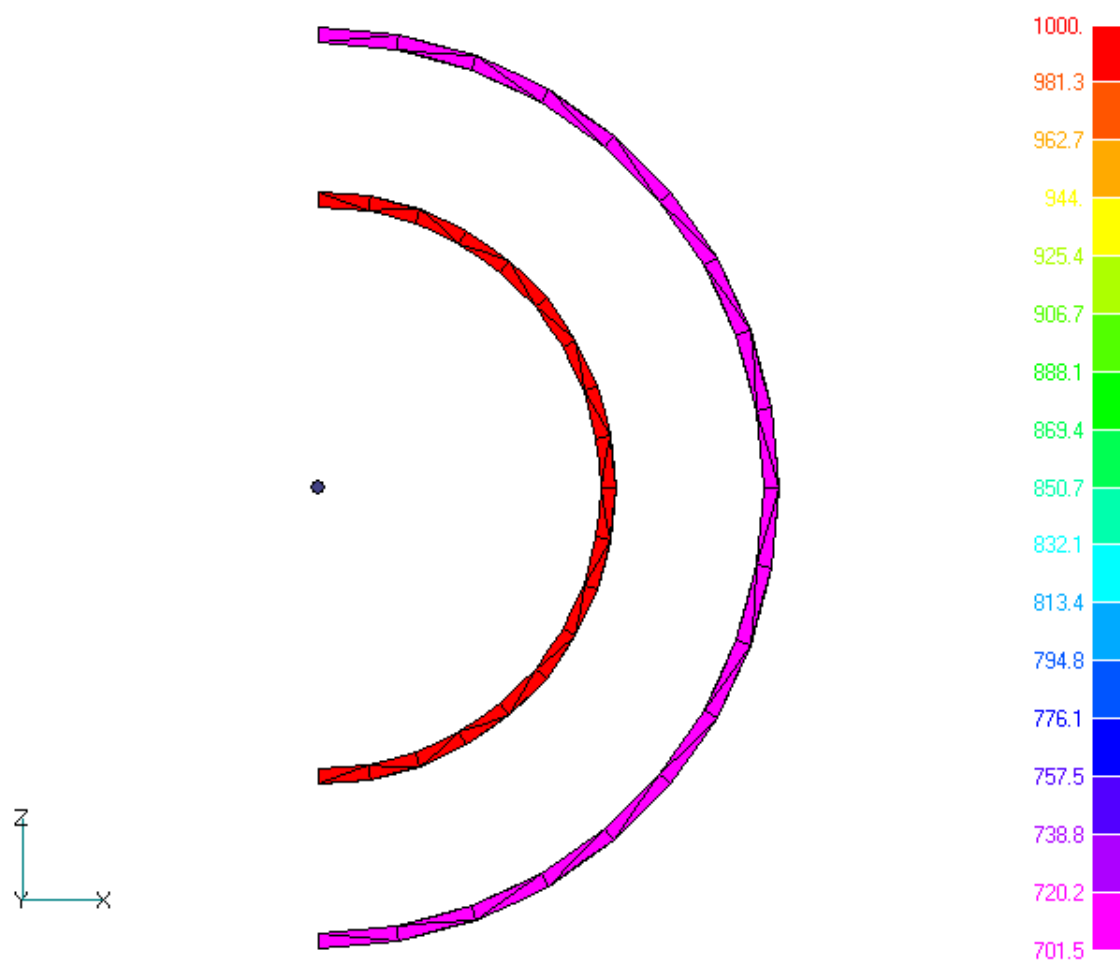


Figure 10-1 Geometry of the concentric sphere





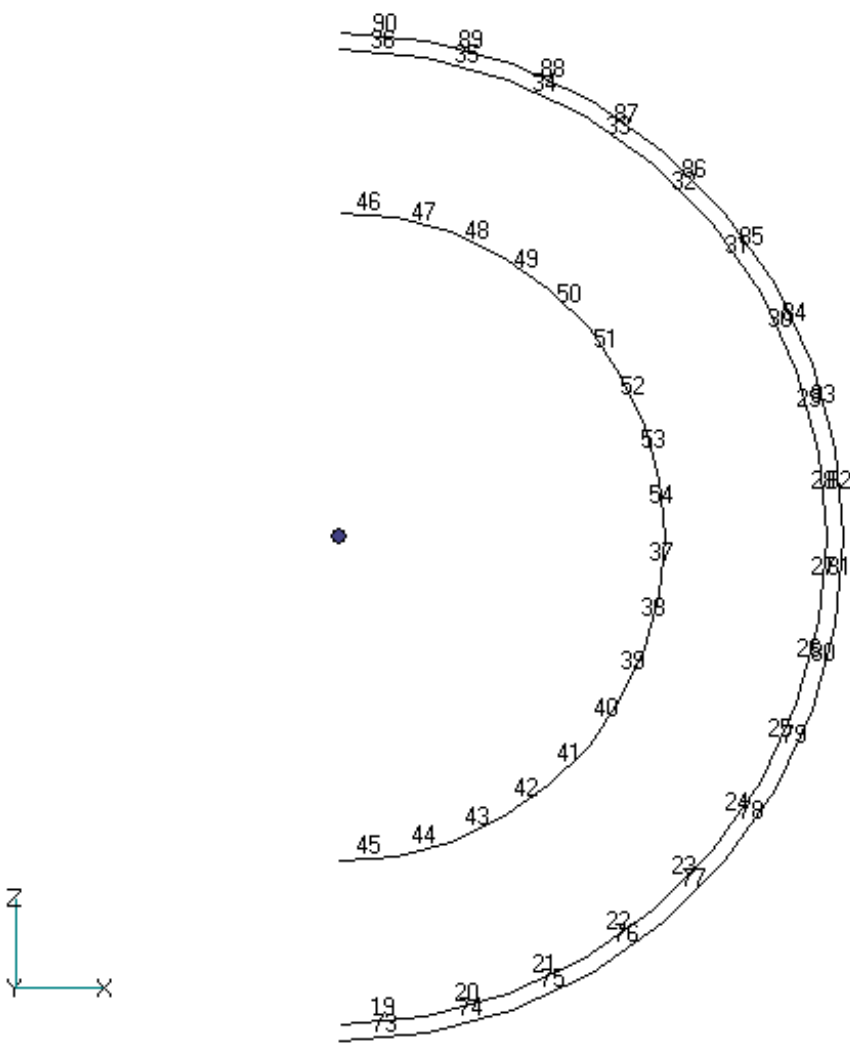


Figure 10-2 CHBDYG element ID

19 -SUM OF	2.16196E-01	9.99992E-01
20 -SUM OF	6.34055E-01	9.99910E-01
21 -SUM OF	1.03089E+00	1.00012E+00
22 -SUM OF	1.40597E+00	1.00014E+00
23 -SUM OF	1.75111E+00	1.00031E+00
24 -SUM OF	2.01178E+00	1.00036E+00
25 -SUM OF	2.21309E+00	1.00038E+00
26 -SUM OF	2.36235E+00	1.00040E+00
27 -SUM OF	2.45948E+00	1.00044E+00
28 -SUM OF	2.45949E+00	1.00044E+00
29 -SUM OF	2.36239E+00	1.00042E+00
30 -SUM OF	2.21312E+00	1.00039E+00
31 -SUM OF	2.01182E+00	1.00038E+00
32 -SUM OF	1.75116E+00	1.00034E+00
33 -SUM OF	1.40599E+00	1.00015E+00
34 -SUM OF	1.03093E+00	1.00015E+00
35 -SUM OF	6.34064E-01	9.99924E-01
36 -SUM OF	2.16198E-01	1.00000E+00
37 -SUM OF	1.09254E+00	9.99900E-01

38 -SUM OF	1.04939E+00	9.99915E-01
39 -SUM OF	9.83127E-01	9.99912E-01
40 -SUM OF	8.93747E-01	9.99907E-01
41 -SUM OF	7.77934E-01	9.99878E-01
42 -SUM OF	6.24725E-01	9.99872E-01
43 -SUM OF	4.58050E-01	9.99860E-01
44 -SUM OF	2.81776E-01	9.99848E-01
45 -SUM OF	9.60719E-02	9.99796E-01
46 -SUM OF	9.60722E-02	9.99800E-01
47 -SUM OF	2.81776E-01	9.99849E-01
48 -SUM OF	4.58050E-01	9.99859E-01
49 -SUM OF	6.24725E-01	9.99873E-01
50 -SUM OF	7.77934E-01	9.99877E-01
51 -SUM OF	8.93747E-01	9.99907E-01
52 -SUM OF	9.83127E-01	9.99912E-01
53 -SUM OF	1.04939E+00	9.99915E-01
54 -SUM OF	1.09254E+00	9.99900E-01

```
view,1,1,both
view,2,1,both
view3d,1,,,0.1,1.0e-8
$VIEW3D      1
set1,111,37,thru,54
set1,222,19,thru,36
radcav,1,,,,,64,
,111,222,222,111
RADSET      1
```

According to the view factor computation:

For a concentric sphere:

$$F1 - 2 = 1.0$$

$$F2 - 1 = (R1 / R2)^{**2}$$

Substituting the radius for this problem, should yield the following analytical solutions.

$$R1 = 1, \text{ and } R2 = 1.5$$

$$F2 - 1 = 0.4444$$

```
*** GLOBAL VIEW FACTORS *** CAVITY ID =      1 ***

MACROI  MACROJ      AI      AI*FIJ      FIJ
111      222      1.25161E+01  1.25147E+01  9.99893E-01
222      111      2.81611E+01  1.25147E+01  4.44398E-01
```

The total view factor from inside to outside is equal to 0.9999, and the view factor from outside surface to inside is equal to 0.4444.

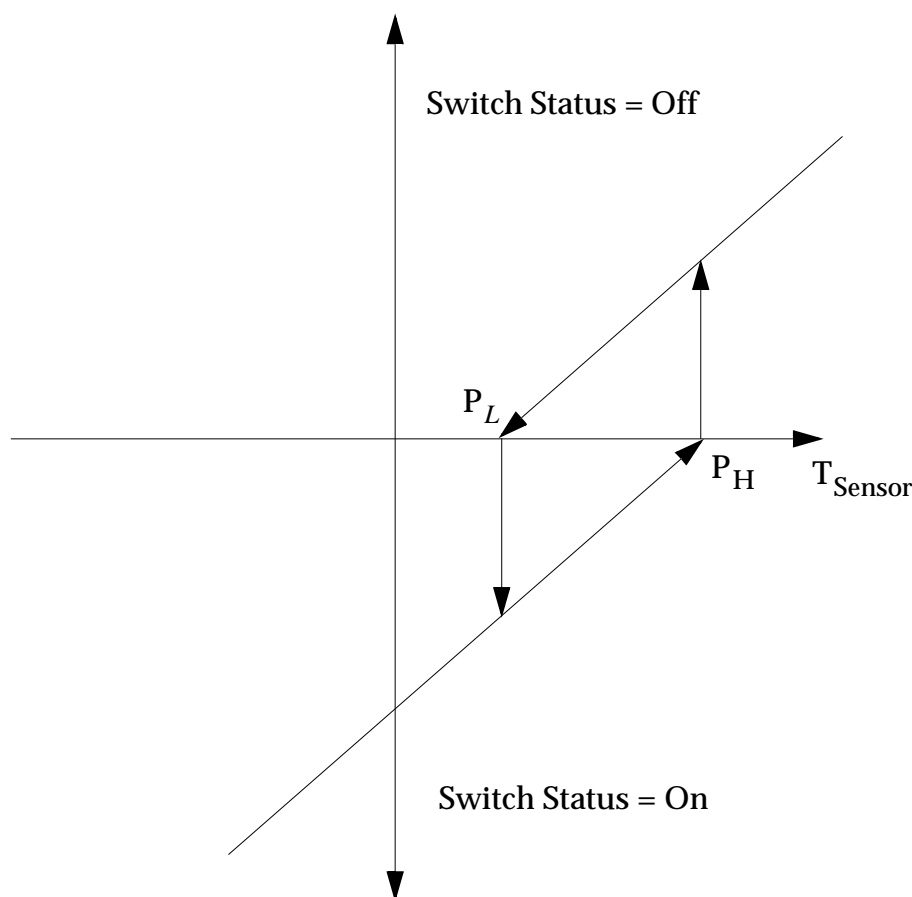
The MSC.Nastran axisymmetric view factor computation correlates well with the analytical calculation.

## 10.3 Thermostat Control with Deadband Applied to a Heat Source

### Introduction

This section demonstrates the principles of thermal control mechanisms that act as on/off switches based on selected temperature band.

Thermal loads (QVOL, QVECT, QBDY3) can be thermostatically controlled by including a CONTRLT Bulk Data entry in the model. A schematic of the process is given in [Figure 10-3](#).



**Figure 10-3 Dead Band Thermostat**

This capability provides thermal control for heating elements with these features:

1. Sensor temperature can be measured from any GRID or scalar point in the model.
2. Numerous switch formats are available with user-defined deadbands.

3. The sensor (therma couple) can be continuously monitored or it can be evaluated at a user-defined sampling rate starting from  $\text{TIME} = 0.0$ . When the sensor is sampled, the control logic is evaluated, and any necessary control action is enforced.
4. The control device can account for the physical actuator time delay and any rise and decay time constants.
5. Automatic time stepping is handled internally giving priority to any controller action in effect. Outside of the control action regions, the time steps size reverts back to the time step size,  $\text{DT}$ , specified by the user on the  $\text{TSTEPNL}$  statement. The standard nonlinear time-step adjustment algorithm is not implemented for control logic analysis.

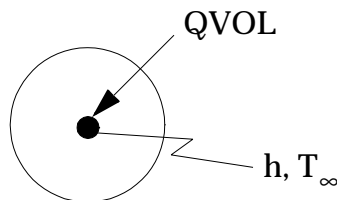
## Examples

### 1 - Lumped Heat Capacitance with Thermostat Control of a Volumetric Heat Source (qvol.dat)

A general assumption for the lumped heat capacitance model is that the Biot number,  $h(V/A)/k$ , is less than 0.1. If this is the case, then a lumped capacity type of analysis assumes a uniform temperature distribution in the solid, and surface convection is large relative to the internal conduction resistance. Furthermore, a simple first order differential equation can be used to arrive at the analytical solution.

### Model Description

In order to model a lumped model, a single CHEXA element that uses the same thermal mass and the convection coefficient as shown in [Figure 10-4](#) is used. Initially, the element is at  $0^\circ\text{C}$ , and is heated by the internal heat generation with 12000 watts. Furthermore, free convection occurs on the six faces on the CHEXA element. The heater is designed to maintain the temperature between 80 and 100 degrees C. When the temperature reached 100 degrees C, the heater shuts off, and when it hits below 80 degrees C, the heater turns back on. The new CONTRLT Bulk Data entry allows control of the temperature from 80 to 100 degrees C, and the volumetric heat generation is turned On or Off based on the SET ID 99.



**Figure 10-4 Lumped Heat Capacitance Model**

$$\begin{aligned} \rho \cdot \text{Vol} \cdot C_p &= 6000. \text{ J}/^\circ\text{C} \\ Q_{\text{VOL}} &= 12000. \text{ W} \\ T(o) &= 0.0 \text{ }^\circ\text{C} \\ h \cdot A &= 100.0 \text{ W}/^\circ\text{C} \\ T_\infty &= 0.0 \text{ }^\circ\text{C} \\ P_L &= 80.0 \text{ }^\circ\text{C} \\ P_H &= 100.0 \text{ }^\circ\text{C} \\ B &= \frac{h \cdot A}{\rho \cdot \text{Vol} \cdot C_p} = \frac{1}{60} \frac{1}{\text{sec}} \\ T(t) &= 120. \cdot (1 - e^{-Bt}) \\ T(107.5) &= 100.0 \text{ }^\circ\text{C} \end{aligned}$$

## Input File

```

SOL 159
CEND
ANALYSIS = HEAT
TITLE = MSC.Nastran job created on 08-Mar-02 at 15:29:11
ECHO = SORT
MAXLINES = 999999999
SPC = 1
IC = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : tran
    SUBTITLE=tran
    TSTEPNL = 1
    DLOAD = 2
    THERMAL(SORT2,PRINT)=ALL
    FLUX(SORT2,PRINT)=ALL
    OLOAD(SORT2,PRINT)=ALL
    SPCFORCES(SORT2,PRINT)=ALL
$ Direct Text Input for this Subcase
OUTPUT(XYLOT)
xgrid=yes
ygrid=yes
xtitle=time
yttitle=temp
xyplot temp/2(T1)
xtitle=time
yttitle=sensor temp
xyplot temp/1(T1)
BEGIN BULK
CONTRLT, 99, 1, T, TSTAT, 80.0, 100.0, 1, 1.0 +
+, 0.0, 0.0, 0.0

```

```

PARAM      SIGMA    1.714-9
PARAM      PRGPST   NO
TSTEPNL    1        200      2.      1      ADAPT    2      -10      UPW
           .01      .001    1.-6
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : solid
PSOLID     1        1        0
$ Pset: "solid" will be imported as: "psolid.1"
CHEXA      1        1        1        2        4        3        5        6
           8        7
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 08-Mar-02           Time: 15:24:54
MAT4       1        204.     60.     100.
$ Nodes of the Entire Model
GRID       1        0.      0.      0.
GRID       2        1.      0.      0.
GRID       3        0.      1.      0.
GRID       4        1.      1.      0.
GRID       5        0.      0.      1.
GRID       6        1.      0.      1.
GRID       7        0.      1.      1.
GRID       8        1.      1.      1.
$ Loads for Load Case : tran
TLOAD1     4        3              1
DLOAD      2        1.      1.      4
$ Fixed Temperatures of Load Set : convection
SPC        1        9        1      0.
$ Volumetric Heat Generation of Load Set : qvol
$QVOL      3        12000.      1
QVOL,3,12000.,99,1
$ Convection to Ambient of Load Set : convection
PCONV      1        1001     0      0.
CONV       100001    1        0      0      9
CONV       100002    1        0      0      9
CONV       100003    1        0      0      9
CONV       100004    1        0      0      9
CONV       100005    1        0      0      9
CONV       100006    1        0      0      9
$ Time Dependent Nodal Temperatures of Load Set : convection
$ Referenced Dynamic Load Tables
$ Constant Load Table
TABLED1    1
           0.      1.      1000.  1.      ENDT
$ Initial Temperatures from Temperature Load Sets
TEMP       1        9        0.
$ Default Initial Temperature
TEMPD      1        0.
$ CHBDYG Surface Elements
CHBDYG     100001      AREA4
           5        7        3        1
CHBDYG     100002      AREA4
           2        4        8        6
CHBDYG     100003      AREA4

```



```

1      2      6      5
CHBDYG 100004      AREA4
3      7      8      4
CHBDYG 100005      AREA4
1      3      4      2
CHBDYG 100006      AREA4
6      8      7      5

$ Free Convection Heat Transfer Coefficients
MAT4      1001      16.6667
$ Scalar Points
SPOINT 9
$ Referenced Coordinate Frames
ENDDATA 259d5d50
```

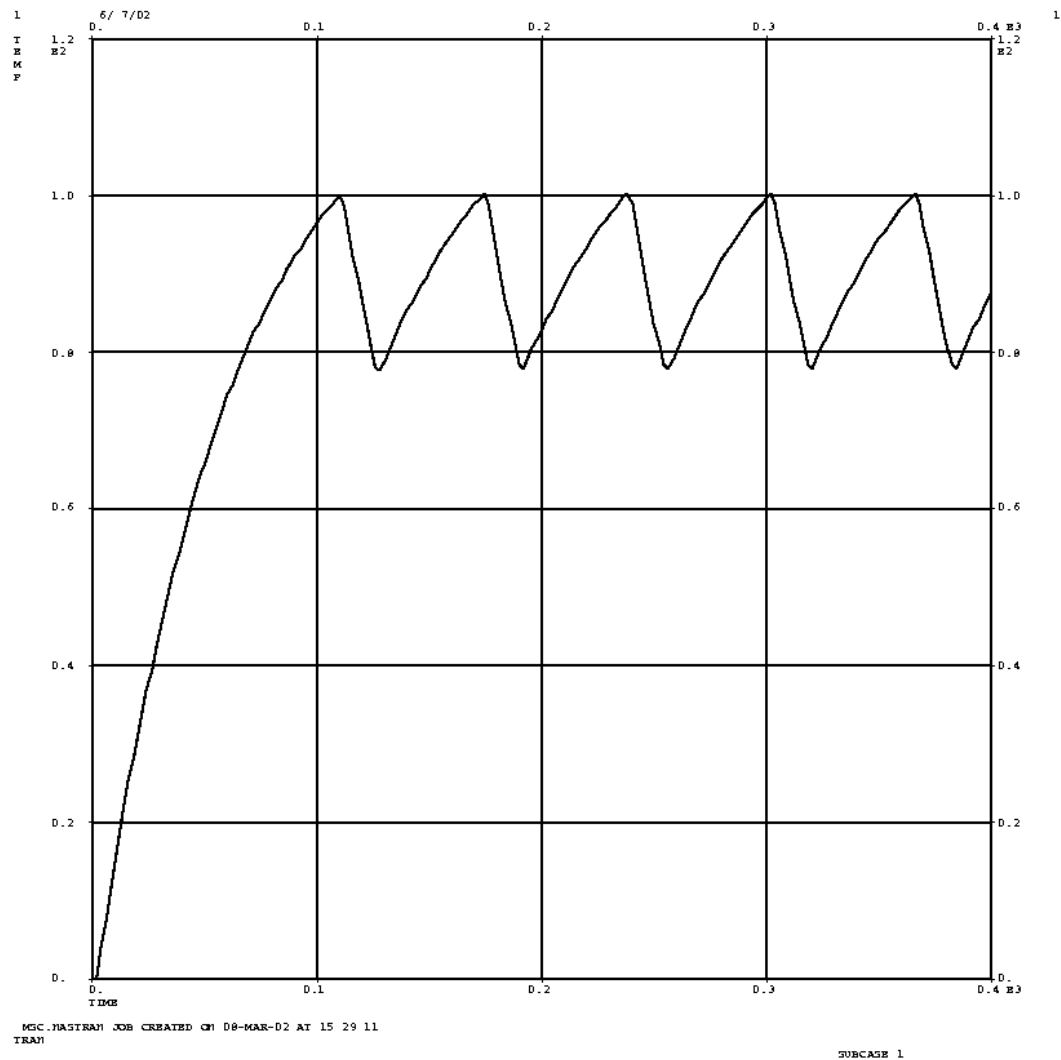
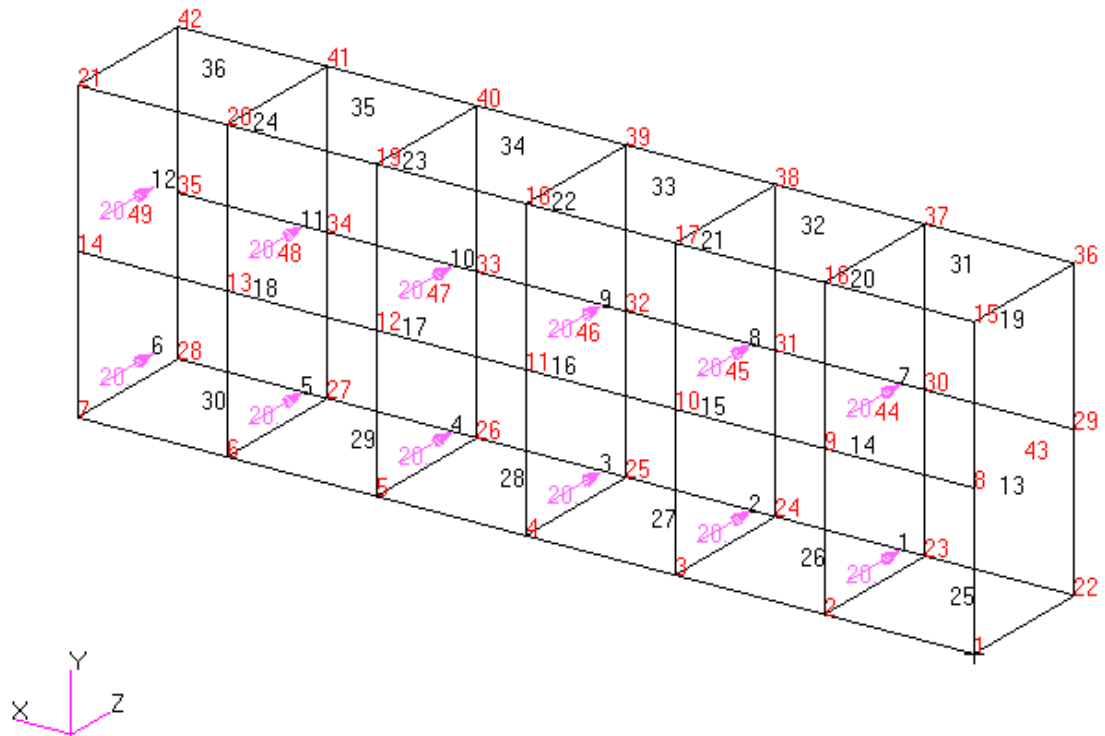


Figure 10-5 Sensor Temperature Profile

## Example 2 (contrlt2.dat)

The following example illustrates the use of an automatic flow regulator with feedback loop using the new CONTRLT Bulk Data entry. Often in the electronics industry, a control process is desired in the thermal model. The heat exchanger shown in **Figure 10-6**, with heat dissipation on one side of the plate, is used to demonstrate this application. There is no airflow initially until the temperature sensor at the left end of the plate (grid point 14) reaches 45 degrees F. When that occurs, the flow will begin from right to left, cooling down the device.



**Figure 10-6 Grid Point and Element IDs for the Heat Exchanger.**

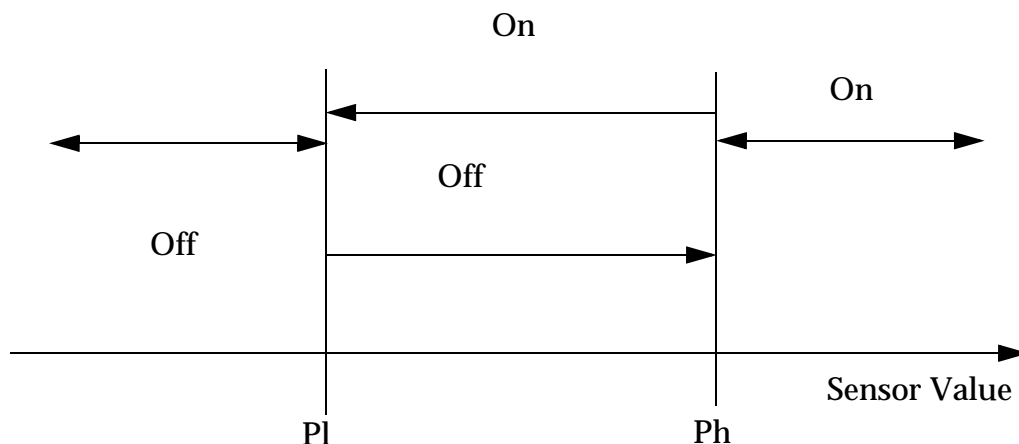
### Boundary conditions:

1. The airflow comes from the right as shown in **Figure 10-6**.
2. A constant heat flux of 20 watts/in is imposed on one side of the plate (far side.)
3. The mass flow rate is 0.0167 lbm/sec.
4. A temperature sensor is set up at grid point 14 (left hand side of Figure 8-6).
5. A 0.3 constant forced convection coefficient is used.

## Model Description

At time=0, the heat exchanger is at 20 degrees F with no airflow. Once the temperature sensor at grid point 14 reaches 45 degrees F, the air flow is automatically turned on. We will look at the temperatures as a function of time at the sensor (grid point 14) and at the exhaust (grid point 49).

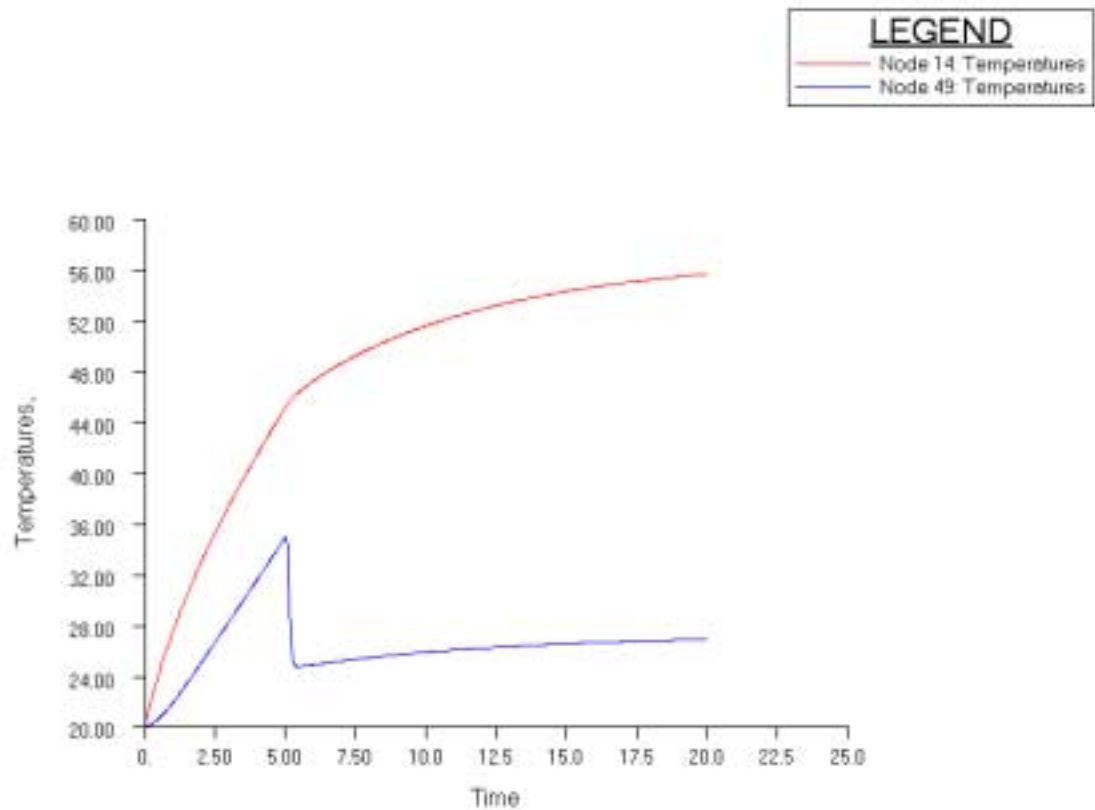
The Control node ID (99) on the CONTRLT entry is referenced on the CONVMM entry, and the sensor location is at grid point 14 using the temperature option (SFORM=T). The control element is a thermostat (CTYPE=TSTAT) and the lower and upper temperature limit are 40. and 45. degrees, respectively. The sensor type (PTYPE=2) is a classical cooling thermostat as shown in [Figure 10-7](#).



**Figure 10-7 Classical Cooling Thermostat.**

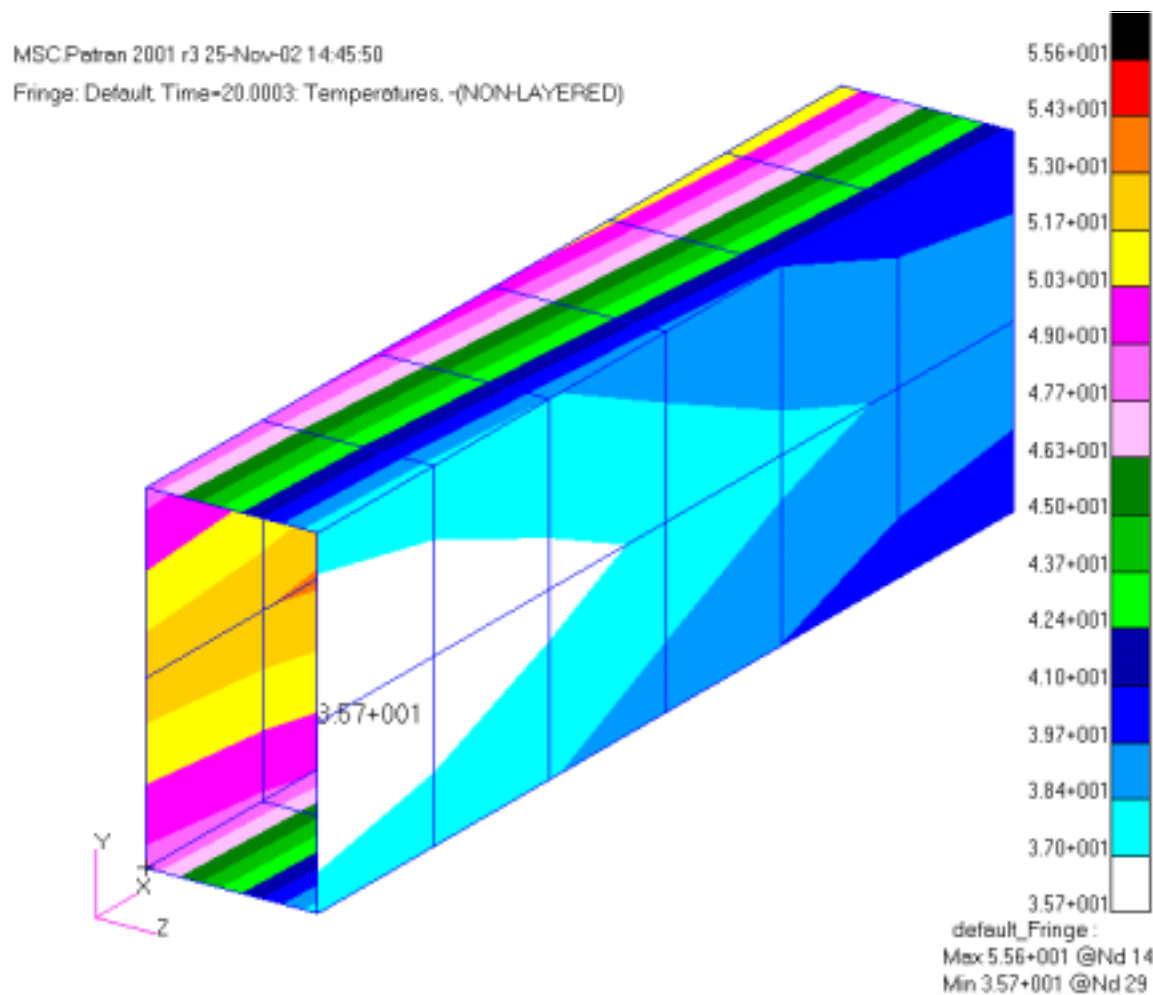
At  $t=5$  sec, the flow is turned ON because the sensor temperature at grid point 14 has reached 45 degrees F. Also notice in [Figure 10-8](#), that the slopes of the temperature at grid points 14 and 49 change due to the flow being turned on.

## Results/Discussion



**Figure 10-8 Temperature Profile at Control Point (Grid Point 14) and Exhaust (Grid Point 49)**

Note that the Ucntmdot is the toggle switch that turns the mass flow rate on or off, based on the condition described by PTYPE. The total mdot is equal to  $2.0 \times$  the control node on scalar point 99, which is  $2 \times .008333 = 0.0167$  lbm/sec. The total mass flow rate is  $\text{mdot} = \text{Ucntmdot} \times \text{CMDOT} \times \text{Ucontrol}$ . A temperature contour of the heat exchanger is shown in [Figure 10-9](#).



**Figure 10-9 Temperature Contour of the Heat Exchanger at t=20 sec**

The initial delta t is 0.02 sec, and the number of time steps is 1000. This yields a total analysis time of  $0.02 \times 1000 = 20$  sec. The corresponding tstepnl is shown below.

tstepnl,500,1000,0.02,1

A good way of estimating the time step is to initially run a transient thermal analysis without the CONTRLT entry. This transient thermal run will provide a good estimate of which initial time step to use.

Pertinent portion of the input file is shown below.

```
CONTRLT, ID, Sensor, SFORM, CTYPE, Pl, Ph, PTYPE, PZERO, +
+, DT, Delay, Taud
CONTRLT, 99, 14, T, TSTAT, 40.0, 45.0, 2, 1.0 +
+, 0.0, 0.0, 0.0
$ This test the case 5d in SRS
$ mdot=Ucntmdot*CMDOT*Ucontrol
$CONVM, EID, PCONID, FLMND, CNTMDOT, TA1, TA2, MDOT
```

CONVM	37	1	0	99	50	2.0
CONVM	38	1	0	99	50	2.0
CONVM	39	1	0	99	50	2.0
CONVM	40	1	0	99	50	2.0
CONVM	41	1	0	99	50	2.0
CONVM	42	1	0	99	50	2.0
SPC	1	99	1	.008333		

CHAPTER

**11**

## MSC.Nastran/MSC.Adams Interface

■ ADAMS/Flex Support

## 11.1 ADAMS/Flex Support

### Introduction

MSC.Nastran allows the use of residual structure only, or any superelement, or any part super element to be used as a component for an ADAMS/Flex flexible body. Flexible body analysis, including large rigid body motions, are available by combining the ADAMS/Flex formulation with reduced flexible component dynamics provide by MSC.Nastran. This allows designers take a component-level Finite Element Analysis and move its characteristic shapes to a system-level motion study for improved design accuracy.

Typical applications are: Elasto-kinematic analysis of vehicular suspensions, engine crankshaft-drivetrain interaction, aircraft landing conditions, and satellite panel deployment.

### Benefits

The MSC.Nastran/ADAMS integration provides an easy method to move from a FE analysis to a system analyses study by providing the direct generation within MSC.Nastran of the MSC.ADAMS Modal Neutral File (MNF) required for the ADAMS/Flex solver.

The interface is initiated by the simple MSC.Nastran Case Control command `ADAMSMNF FLEXBODY=YES` and the addition of a Bulk Data entry, `DTI,UNITS`. The interface is applicable in SOL 103, SOL 111, SOL 112, or SOL 103 from a nonlinear SOL 106 restart using standard MSC.Nastran `PARAM,NMLOOP`.

The user can easily define the flexible body attachment points in MSC.Nastran by either defining the component as a superelement or part super element, in which case, the physical external (a-set) grids become the attachment points; or for a residual only type model, using standard MSC.Nastran `ASET` Bulk Data entries to define the attachment points.

If the component requires prestiffening that does not entail large deformation standard MSC.Nastran Case Control, `STATSUB`, can be used in SOL 103, SOL 111, or SOL 112 to provide the required loading.

For a component that will experience nonlinear deformation, a standard SOL 106 run can be made and then using standard `PARAM,NMLOOP`, the component can be restarted into SOL 103 with the advantage that `ASET` Bulk Data entries can be used at restart time to define the attachment points.



The component modes determined by MSC.Nastran automatically represent a set of constraint modes for boundary coordinates plus a truncated set of fixed-interface normal modes; this combination is referred to as the Craig-Bampton modes. These modes, by default, are augmented by residual flexibility modes to give a correct representation of spatial loads. The spatial loads may include physical loads and inertia relief loads.

## Theory

### Basic Component Mode Equations

This section briefly describes the equations used in the MSC.Nastran/MSC.ADAMS interface.

In MSC.Nastran modal solutions, the physical nodal displacements  $\{u\}$  are related to modal amplitudes  $\{\zeta\}$  by the relationship:

$$\{u\} = [\Upsilon]\{\zeta\} \quad \text{Eq. 11-1}$$

where  $[\Upsilon]$  is the matrix of mode shapes containing the Craig-Bampton modes and any residual flexibility modes.

The above transformation is then applied to the physical mass, stiffness, and load matrices. The resulting matrices are then reduced to analysis size (a-set) matrices. For modal analysis, the a-set consists of the modal amplitudes and the physical boundary grids external to a superelement or the modal amplitudes and the physical grids defined using an ASET Bulk Data entry. The resulting form, symbolically using the stiffness matrix as an example is:

$$[K_{aa}] = [\Upsilon_{ga}]^T [K_{gg}] [\Upsilon_{ga}] \quad \text{Eq. 11-2}$$

with appropriate similar result for the mass and load matrices.

To isolate rigid body modes as well as any other mechanisms or poorly-conditioned modes and to insure orthonormalization, an a-set eigenvalue problem is solved. This results in a new matrix of shapes  $[\Phi_{a\lambda}]$ , where the  $\lambda$  may be equal or fewer than the initial a-set modes. The modal stiffness, modal mass, and modal load a-set matrices are then transformed as:

for stiffness

$$[K_{\lambda\lambda}] = [\Phi_{a\lambda}]^T [K_{aa}] [\Phi_{a\lambda}] \quad \text{Eq. 11-3}$$

for modal mass

$$[M_{\lambda\lambda}] = [\Phi_{a\lambda}]^T [M_{aa}] [\Phi_{a\lambda}] \quad \text{Eq. 11-4}$$

and for modal loads

$$[P_{\lambda}] = [\Phi_{a\lambda}]^T [P_a] \quad \text{Eq. 11-5}$$

Additionally, a diagonal matrix of a-set unit boundary displacements is formed and taken through standard MSC.Nastran transformations up to g-set size. This matrix, say  $[\Xi_{ga}]$ , is transformed as:

$$[\Xi_{g\lambda}] = [\Xi_{ga}] [\Phi_{a\lambda}] \quad \text{Eq. 11-6}$$

The above four orthonormal matrices and the physical mass matrix is then exported to MSC.ADAMS in MNF format. The matrix  $[\Xi_{g\lambda}]$  is also used to obtain stress and strain shapes.

## Mass Invariant Calculations

When executing the MSC.Nastran/ADAMS interface, MSC.Nastran will export to MSC.ADAMS up to nine mass invariants. Briefly, these invariants are obtained as follows:

The instantaneous location  $\{r_p\}$  of a grid point p can be expressed as:

$$\{r_p\} = \{x\} + [A]\{s_p\} + [A]\{u_p\} \quad \text{Eq. 11-7}$$

where

$\{x\}$  is the position vector from the origin of a ground reference frame to the origin of a reference frame attached to the flexible body.

$[A]$  is the direction cosine matrix between the ground reference and the reference frame attached to the flexible body.

$\{s_p\}$  is the position vector from the origin of the reference frame attached to the flexible body to the undeformed position of a grid point p.  $[A]\{s_p\}$  are the components of  $\{s_p\}$  in the ground reference frame.

$\{u_p\}$  is the translational deformation of the grid point p in the reference frame attached to the flexible body.  $[A]\{u_p\}$  are the components in the ground reference frame.

The velocity  $\{v_p\}$  is defined as:

$$\{v_p\} = \{\dot{x}\} + [\dot{A}](s_p + u_p) + [A]\{\dot{u}_p\} \quad \text{Eq. 11-8}$$

Since the columns of  $[A]$  are unit vectors along the coordinate axes of the reference frame attached to the flexible body, we can use the angular velocity  $\{\Omega\}$  of the reference frame attached to the flexible body relative to the ground reference to determine  $[\dot{A}]$ . Expressing the results in a body fixed system we get:

$$[\dot{A}] = [A][\tilde{\Omega}] \quad \text{Eq. 11-9}$$

where  $[\tilde{\Omega}]$  is a skew symmetric matrix containing the components of the angular velocity.

The generalized coordinates of the flexible body are defined as:

$$\{\xi\} = \begin{bmatrix} \{x\} \\ \{\psi\} \\ \{q\} \end{bmatrix} \quad \text{Eq. 11-10}$$

where  $\{\psi\}$  is a body fixed 3-1-3 set of Euler angles,  $\{q\}$  is the vector of modal coordinates used to define the generalized coordinates of the flexible body, and  $\{x\}$  is defined above.

Then  $\{\Omega\}$  can be defined as:

$$[\Omega] = [B]\{\dot{\psi}\} \quad \text{Eq. 11-11}$$

where  $[B]$  is the matrix of sines and cosines representing the Euler angles.

We can define the following transformations:

$$[u_p] = [\phi_p]\{q\} \quad \text{Eq. 11-12}$$

$$[\dot{u}_p] = [\phi_p]\{\dot{q}\} \quad \text{Eq. 11-13}$$

$$[\omega_p] = [B]\{\dot{\psi}\} + [\phi_p^*]\{\dot{q}\} \quad \text{Eq. 11-14}$$

where  $[\phi_p]$  is the partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid p and  $[\phi_p^*]$  is the partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid p.

Then the velocity  $\{v_p\}$  can be expressed as:

$$\{v_p\} = \begin{bmatrix} I & -A\tilde{X}B & A\phi_p \end{bmatrix} \{\xi\} \quad \text{Eq. 11-15}$$

where

$$[\tilde{X}] = [\tilde{s}_p] + [\tilde{u}_p] \quad \text{Eq. 11-16}$$

is a skew symmetric matrix of coordinate locations and translations.

The kinetic energy of the flexible body is then approximated by the relationship:

$$T = \frac{1}{2} \sum_p m_p [v_p] \{v_p\} + \frac{1}{2} \sum_p [\omega_p] I_p \{\omega_p\} \quad \text{Eq. 11-17}$$

where  $[ \ ]$  represent a row matrix and  $I_p$  is the inertia tensor of grid p.

It is this approximation to the kinetic energy and the need to keep the  $[A]$  matrix out of the mass invariant calculations that currently restricts the MSC.Nastran/ADAMS interface analysis to a lumped mass formulation.

Expanding the relationship for kinetic energy, we obtain the following nine mass invariants:

$${}^1_I = \sum_{p=1}^N m_p \quad \begin{matrix} 1 \\ 1 \times 1 \end{matrix} \quad \text{Eq. 11-18}$$

$${}^2_I = \sum_{p=1}^N m_p s_p \quad \begin{matrix} 2 \\ 3 \times 1 \end{matrix} \quad \text{Eq. 11-19}$$

$${}^3_{I_j} = \sum_{p=1}^N m_p \Phi_p \quad j = 1, \dots, M \quad \begin{matrix} 3 \\ 3 \times M \end{matrix} \quad \text{Eq. 11-20}$$

$${}^4_I = \sum_{p=1}^N m_p \tilde{s}_p + I_p \Phi_p^* \quad \begin{matrix} 4 \\ 3 \times M \end{matrix} \quad \text{Eq. 11-21}$$

$$\begin{matrix} 5 \\ 3 \times M \end{matrix} I_j = \sum_{p=1}^N m_p \check{\Phi}_{pj} \Phi_p \quad j = 1, \dots, M \quad \text{Eq. 11-22}$$

$$\begin{matrix} 6 \\ M \times M \end{matrix} I = \sum_{p=1}^N m_p \Phi_p^T \Phi_p + \Phi_p^{*T} \mathbf{I}_p \Phi_p^* \quad \text{Eq. 11-23}$$

$$\begin{matrix} 7 \\ 3 \times 3 \end{matrix} I = \sum_{p=1}^N m_p \tilde{s}_p^T \tilde{s}_p + \mathbf{I}_p \quad \text{Eq. 11-24}$$

$$\begin{matrix} 8 \\ 3 \times 3 \end{matrix} I_j = \sum_{p=1}^N m_p \tilde{s}_p \check{\Phi}_{pj} \quad j = 1, \dots, M \quad \text{Eq. 11-25}$$

$$\begin{matrix} 9 \\ 3 \times 3 \end{matrix} I_{jk} = \sum_{p=1}^N m_p \check{\Phi}_{pj} \Phi_{pk} \quad j, k = 1, \dots, M \quad \text{Eq. 11-26}$$

where  $s_p = [xyz]^T$  are the coordinates of grid point  $p$  in basic and

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix}$$

is the skew-symmetric vector cross product operator.  $\Phi_p$  is the partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid  $p$ ,  $\mathbf{I}_p$  is the inertia tensor of grid  $p$ ,  $\Phi_p^*$  is the partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid  $p$ .  $\check{\Phi}_{pf}$  is the skew-symmetric matrix formed for each grid translational degree of freedom for each mode.  $M$  is the number of modes and  $N$  is the number of grids.

## Inputs

If MSC.Nastran/ADAMS interface is desired, only **two** unique **entries** are **required**. They are:

## Case Control Unique to ADAMS/Flex Interface

ADAMSMNF FLEXBODY=YES which causes the ADAMS/Flex MNF to be generated.

## Bulk Data Entries Unique to ADAMS/Flex Interface

DTI,UNITS used to specify the unit system to be used by ADAMS/Flex.

## Typical File Management for ADAMS/Flex Interface

- ASSIGN MASTER='location\_of\_106\_database' - used in a SOL 103, SOL 111, or SOL 112 run to locate the SOL 106 data base for restart.
  - RESTART - used in a modal solution to indicate that run is a restart from a SOL 106 run. This is a standard MSC.Nastran restart.
  - DBLOC - used in a modal solution run to extract files from a SOL 106 data base. This is used for a nonstandard MSC.Nastran restart.

## Typical Case Control for ADAMS/Flex Interface

- ADAMSMNF FLEXBODY = YES - **required for interface run**. See the ADAMSMNF case description for a complete list of options and interface rules.
- STRESS(PLOT) - necessary for stress shapes.
- GPSTRESS(PLOT) - necessary for grid point stress shapes to be included in MNF.
- STRAIN(PLOT) - necessary for strain shapes.
- GPSTRAIN(PLOT) - necessary for grid point strain shapes to be included in MNF.
- METHOD = n - required above or in first subcase for modal solutions.
- RESVEC = character\_value - controls calculation of residual flexibility (including inertia relief) modes. SOL 111 and SOL 112, residual flexibility is defaulted to on for both component (o-set) and system modes (a-set). In SOL 103, residual flexibility is defaulted to on for only component modes (o-set).
- LOAD = n - used if modal loads are desired. A different subcase will define different load conditions.
- STATSUB(PRELOAD) = subcase\_ID - necessary if a linear preload condition is to be included in the MNF.

- SUPORT1=seid - necessary to select a static support set for a residual only linear preload run.
- SPCF(PLOT) = ALL - necessary with preload to ensure complete preload reaction condition is to be included in the MNF.
- MPCF(PLOT) = ALL - necessary with preload to ensure complete preload reaction condition is to be included in the MNF.
- OUTPUT(POST) - necessary to define volume and surface for grid stress or strain shapes.
  - SET nnn = list - set of elements for surface definition for grid stress or strain shapes.
  - SURFACE nnn SET nnn NORMAL z3 - surface definition - Only one FIBER selection is allowed for all SURFACES.
  - VOLUME nnn SET nnn - volume definition.
  - The default SYSTEM BASIC is required with SURFACE or VOLUME.
- OUTPUT(PLOT) - necessary to define elements used to select grids to display the component in ADAMS.
  - SET nj = list of elements used to select grids to display the component.
- NLPARM=n - used to select nonlinear parameters on a NLPARM Bulk Data entry.
- PARAM,NMLOOP,m - necessary for a SOL 103 restart from SOL 106. Describes the nonlinear iteration loop m to use for modal analysis.
- SUPER=n,SEALL=n - useful with multiple superelement model to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.

## Typical Bulk Data for ADAMS/Flex Interface

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME - **required for interface run.** See the ADAMSMNF case description for a complete description of ADAMS unit requirements and interface rules. For parts super elements, must reside in the main Bulk Data Section.
- SPOINT,id\_list - used to define and display modal amplitude.
- SESET,SEID,grid\_list - used to define a superelements (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.

- SEELT,SEID,element\_list - used to reassign superelement boundary elements to an upstream superelement.
- RELEASE,SEID,C,Gi - optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational DOFs from an analysis where only translational DOFs are required.
- SEQSET, SEID,spoint\_list - used to define modal amplitudes of a superelement (see SEQSET1).
- SENQSET,SEID,N - used to define modal amplitudes of a part super element. Must reside in the main Bulk Data Section.
- ASET,IDI,Ci - used to define attachment points for a residual only run (see ASET1). This entry cannot exist in the SOL 106 run of a restart into SOL 103. It must be added in the SOL 103 restart.
- QSET1,C,IDI - used to define modal amplitudes for the residual structure or modal amplitudes for a part super element (see QSET). This entry cannot exist in the SOL 106 run of a restart into SOL 103. It must be added in the SOL 103 restart.
- SUPORT1,SID,IDI,Ci - used to define the static support for a preload condition with a residual only run. This entry is case control selectable. **Do Not Use SUPORT.**
- PLOTTEL,EID,Gi - can be used, along with existing model elements, to define elements used to select grids to display the component in ADAMS.
- EIGR,SID,METHOD,... - used to obtain real eigenvalue extraction (see EIGRL).
- PARAM,GRDPNT,value - mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  will be computed using results of MSC.Nastran grid point weight generator execution in the basic coordinate system.
- PARAM,LGDISP - used to turn on large displacement theory in SOL 106.
- NLPARM,N,INC,... - used to select nonlinear parameters.

## Output

An output unique to the MSC.Nastran/ADAMS interface run is the MNF for export to the ADAMS/Flex run. If ADMOUT=YES is set on the ADAMSMNF Case Control, the op2 file described on the ADAMSMNF Case Control command description will also be created.



## Guidelines and Limitations

Guidelines for using MSC.Nastran/Adams interface is summarized below:

1. Superelements and parts super elements require a superelement license.
2. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.
3. p-elements are not allowed as they are always coupled mass formulation.
4. If for preload conditions a simple static support system and a self equilibrating load can be used, the SPCF(PLOT) and MPCF(PLOT) statements are not required.
5. For nonlinear SOL 106 runs, if the MATS1 entry is included, an incremental approach between MSC.Nastran and ADAMS/Flex should be adopted because of the path dependent load history.
6. For a standard MSC.Nastran restart (using RESTART file management command) from a SOL 106 (nonlinear) into SOL 103 (modal with MSC.Nastran/ADAMS interface), the SPOINTS must be in the SOL 106 run and the associated QSET1s must be in the SOL 103 run.
7. If a user has legacy SOL 106 runs or forgets to put the SPOINTS in the SOL 106 run, the SPOINTS and associated QSET1s can be placed in the modal run which then must use a nonstandard restart using the DBLOCATE file management command. The command **must contain** the following six data blocks: bulk, estnl, ugni, qgni, qmgni, and pgni. The command must also have a dbname that corresponds to the dbname on the ASSIGN command. The command takes the form:
  - dblocate datablk=(bulk,estnl,ugni,qgni,qmgni,pgni) where (version=1), logi=dbname
8. When **restarting** with a RESTART or DBLOCATE commands from a SOL 106 run into a modal run (SOL 103, SOL 111, SOL 112), **the first SUBCASE in the modal run must contain the LOAD= entry** that reflects the load condition of the chosen converged nonlinear solution. Other SUBCASES with their own load conditions may follow the first SUBCASE to be used for generation of modal load shapes. This is required for labeling reasons and proper column alignment.

9. When supplying SPOINTS/QSETS combinations, to capture accurately the modal shapes. If  $n$  is the number of modes specified on the EIGR or EIGRL Bulk Data entries and  $p$  is the number of load cases specified, then the number of SPOINTS(ns) should be at least  $ns = n + (6 + p)$ , assuming that residual flexibility is on. In general, there cannot be too many SPOINTS, as excess ones will be simply truncated with no performance penalty.
10. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in very accurate representation of the component shapes at little additional cost.
11. The current ADAMS/Flex does not support OMIT or OMIT1 Bulk Data entries.
12. PLOTETs or other elements may be used to define FE mesh detail for ADAMS/Flex using the PSETID=set\_entry on the ADAMSMNF Case Control command. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements grids attached only to a-set or exterior grids), a SEELT Bulk Data entry MUST be specified to keep that display element with the superelement component. The SEELT entry would not be required with parts superelements, as boundary elements stay with their component.
13. PARAM,WTMASS,value with a value other than 1.0 may be used with an MSC.Nastran/ADAM/Flex run when generating a MNF. It must have consistent units with regard to the DTI,UNITS Bulk Data entry as discussed on the ADAMSMNF Case Control command. Before generating the MNF, MSC.Nastran will appropriately scale the WTMASS from the physical mass matrix and the mode shapes.
14. The two types of loads in MSC.Nastran are those that are element specified (such as a PLOAD4 entry) and those that are grid specific (such as a FORCE entry). When doing a standard MSC.Nastran run, the distinction is not important. When doing an ADAMSMNF FLEXBODY=YES run a subtle distinction exists when superelements are used. Element specified type loads always stay with the superelement. The superelement sees the total element specified applied load. For grid specified loads, any grid specified load attached to an external grid will move down stream with the grid. That is to say, it is part of the boundary and not part of the superelement (or ADAMS/Flex component). This distinction applies only to a superelement run and not to a residual only or a parts super element run.

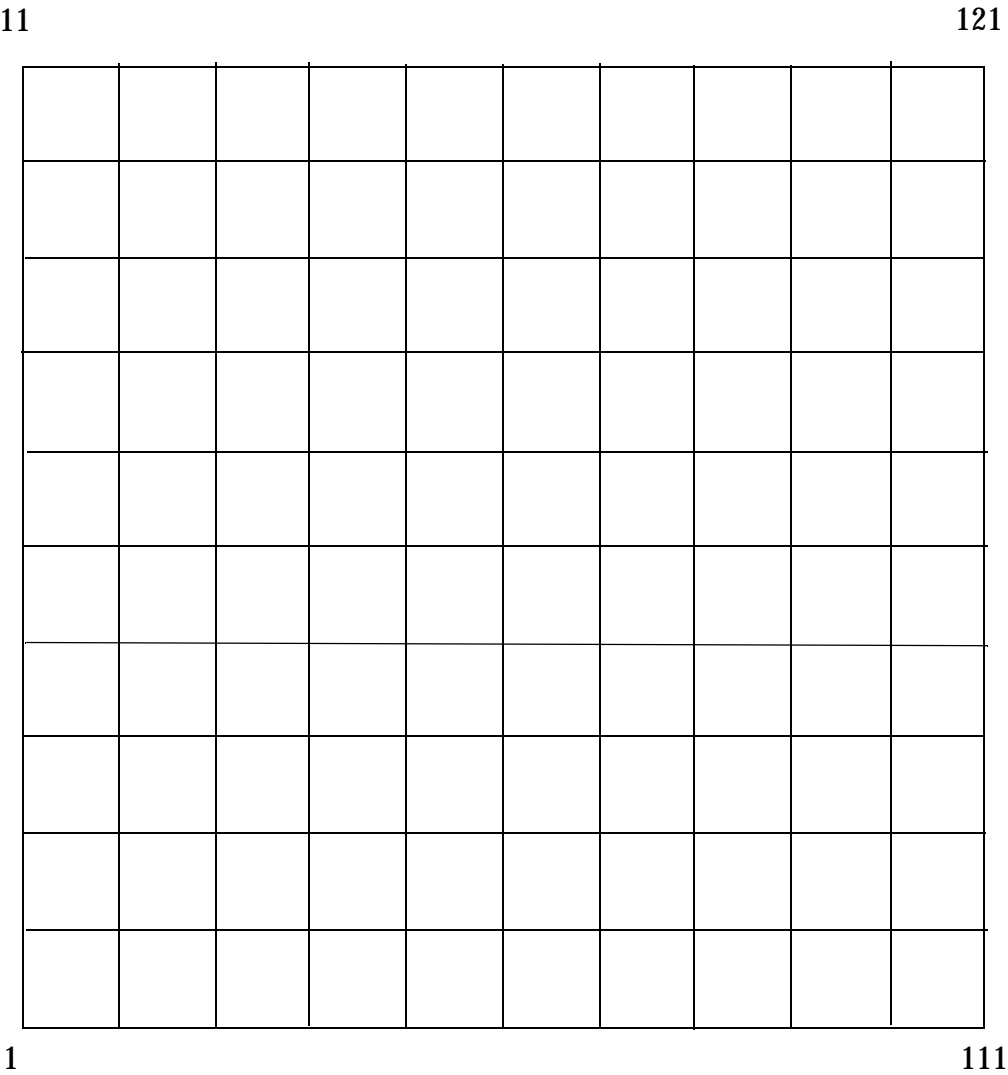
15. The loads specified in MSC.Nastran generally, fall into two categories: non follower or fixed direction loads (non circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of a fixed direction load is the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines the example of a follower load is the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in MSC.Nastran, the follower loads are always active in SOL 103 and SOL 106 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In an ADAMSMNF FLEXBODY=YES with superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement (or ADAMS/Flex component), the follower load will move downstream with the grid. Thus, for an ADAMSMNF FLEXBODY=YES run, the downstream follower contribution to the components stiffness will be lost. This could yield poor results. This caution applies only to a superelement run and not to a residual only or a parts superelement run.
16. In Examples 4-8, an in plane tensioning preload is applied to the plate structure. If a non follower type load such as a FORCE entry is used as in Example 4, two rigid body modes will be lost (a rotation about the y-axis and a rotation about the z-axis). This is due to the fact that when the pre load is applied, the FORCE entry, that is fixed in direction, does not rotate with the structure. Because a preload condition requires at least a set of static supports, the resulting internal stress system used to form the differential stiffness reflects this support state and when added the elastic stiffness, will add stiffness to two rotational rigid body modes. The FORCE1 entry used in Examples 5-8 are follower loads and are applied in such a manner as to rotate with the structure. The differential stiffness still has the constraints discussed for Example 4; however, for these examples a corrective follower stiffness is added that cancels out the differential stiffness and the component sees six rigid body modes.
17. For the MSC.Nastran 2004 release, the ADAMS/Flex tool kit does not support the combination of both preload and stress or strain shapes. Therefore, when using preload, do not include GPSTRESS or GPSTRAIN Case Control commands.
18. For MSC.Nastran 2004, to insure compatibility with the MSC.Adams msc2mnf tool kit, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the MSC.Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any output2 files generated will have a pre-MSC.Nastran 2004 format.

## Examples

The eight small examples discussed below are intended to show salient features of the MSC.Nastran/ADAMS interface with the following options:

1. Flexible body component as a superelement.
2. Flexible body component as a residual only run.
3. Flexible body component as a part superelement.
4. Flexible body component with preload as a superelement - no follower load.
5. Flexible body component with preload as a residual only run - with follower load.
6. Flexible body component with preload as a part superelement - with follower load.
7. A structural component with nonlinear effects run in SOL 106 and restarted into SOL 103 as a flexible body component using standard restart - with follower load.
8. A structural component with nonlinear effects run in SOL 106 and restarted into SOL 103 as a flexible body component using nonstandard dblocate restart - with follower load.

The basic model is a square plate divided into a 10 x 10 mesh. The four corner points, grid point 1 (0.,0.,0), grid point 11 (0.,1.,0.), grid point 111 (1.,0.,0.), and grid point 121 (1.,1.,0.) are considered the attachment points. The basic geometry, material, and loads are given by the include file shown at the end of the examples. The Case Control commands and Bulk Data entries shown in the examples try to highlight features important to the interface run.



**Example 1 - Component Modeled as a Superelement**

This example represents a MSC.Nastran/ADAMS interface run with the component modeled as superelement 200 as defined by the SESET Bulk Data entries. Grids listed on this entry represent the component, grids not listed (in this case grid point 1, grid point 11, grid point 111, and grid point 121) are exterior to the superelement and are used as the attachment points for ADAMS.

The ADAMSMNF Case Control command initiates the interface run with the **required** FLEXBODY=YES entry. Also included on this entry is the optional PSETID=7772, that provides a set of PLOTTEL elements whose grids are retained in the MNF and whose connectivity defines face geometry for the ADAMS display. The OUTPUT(PLOT)

section is used to define the set of elements for the PSETID. For this case, PLOTELS were used. These are defined at the end of the bulk section. The SEELT is used so that the plotels defined on the boundary, stay with the component.

Grid point stress and strain shapes are desired; thus, the sequence of STRESS, STRAIN, GPSTRESS, GPSTRAIN commands. The (PLOT) item suppresses output of these items to the f06 file. The OUTPUT(POST) section is used to define the surface for the grid point stress and strain requests.

The DTL,UNITS in the Bulk Data Section is **required** for the interface run. This defines the units to be used by ADAMS/Flex.

Standard SUBCASE is used to define two pressure load conditions. No constraints are applied to the component; hence, the modal analysis will represent a free structure.

The PARAM,WTMASS is not recommended with interface runs and this is noted by explicitly setting it to its default value of 1.

The PARAM,GRDPNT will cause the MSC.Nastran grid point weight generator (GPWG) to run and display a summary of the mass properties of the model. Any physical grid location may be chosen for the display purpose. However, with the ADAMS interface, if PARAM,GRDPNT is present, GPWG will be run a second time **always** at the origin of the basic system and its results will be used for mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  computations for MNF export.

The SPOINT and SEQSET1 Bulk Data entries are used to define the component modes. These entries must specify enough degrees of freedom or modal amplitudes to capture the shape of the component and residual flexibility for any loading conditions.

```
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
```

```
RESVEC = COMPONENT
$
$ Generate stress and strain grid shapes
$
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
$ Define surface for stress and strain grid shapes
$
OUTPUT(POST)
SET 9998 = ALL
SURFACE 9998 SET 9998 NORMAL X3
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$
$ Define interior grids as superelement 200
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
SESET,200,2,THRU,10
SESET,200,12,THRU,110
SESET,200,112,THRU,120
```

```

$
$ SCALAR Point and SEQSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80018
SEQSET1,200,0,80001,THRU,80018
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ SEELT to put element 1003 and 1004 into SE 200
$
SEELT 200    10003 10004
$
$ Get model data and load data
$
include 'modell.dat'
$
ENDDATA

```

## Example 2 - Component Modeled as a Residual Only Run

This example represents a MSC.Nastran/ADAMS interface run with the component modeled as residual only structure. Grid point 1, grid point 11, grid point 111, and grid point 121 are to be used as the attachment points for ADAMS. This is accomplished by use of the ASET1 Bulk Data entry.

The ADAMSMNF Case Control entry initiates the interface run with the **required** FLEXBODY=YES entry. Also included on this entry is the optional PSETID=7772 that provides a set of PLOTTEL elements whose grids are retained in the MNF and whose connectivity defines face geometry for ADAMS display. The OUTPUT(PLOT) section is used to define the set of elements for the PSETID. For this case, PLOTTELs were used and these are defined at the end of the Bulk Data Section.

For this run, in addition to the MNF for export to ADAMS, op2 files are desired for export into MSC.Fatigue. This op2 files generation is initiated by the ADMOUT=YES entry, the ADAMSMNF Case Control command. The ASSIGN statement in the MSC.Nastran File Management Section will cause the op2 file to be written with the file name residual\_op2.out to fortran unit 20.



The additional OUTSTRS=YES on the ADAMSMNF Case Control command will cause element stress shapes to be output to the op2. This requires the STRESS command in the Case Control command.

Grid point stress and strain shapes are desired for export to the MNF and thus, the sequence of STRESS, STRAIN, GPSTRESS, GPSTRAIN commands. The (PLOT) item suppresses output of these items to the f06 file. The OUTPUT(POST) section is used to define the surface for the grid point stress and strain requests.

The DTI,UNITS in the Bulk Data Section is **required** for the interface run. This defines the units to be used by ADAMS/Flex.

Standard SUBCASE is used to define two pressure load conditions. No constraints are applied to the component. Hence, the modal analysis will represent a free structure.

The PARAM,WTMASS is not recommended with interface runs and this is noted by explicitly setting it to its default value of 1.

The PARAM,GRDPNT will cause the MSC.Nastran grid point weight generator (GPWG) to run and display a summary of the mass properties of the model. Any physical grid location may be chosen for the display purpose. However, with the ADAMS interface, if PARAM,GRDPNT is present, GPWG will be run a second time, **always** at the origin of the basic system and its results will be used for mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  computations for MNF export.

The SPOINT and QSET1 Bulk Data entries are used to define the component modes. These entries must specify enough degrees of freedom or modal amplitudes to capture the shape of the component and residual flexibility for any loading conditions.

```
$ ADAMS/Durability with MSC.Fatigue requires op2 generation
$
ASSIGN OUTPUT2='e2_residual_op2.out' STATUS=UNKNOWN UNIT=20 FORM=UNIFORM
$
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$ admout=yes also output op2 file for ADAMS/Durability with MSC.Fatigue
$ outstrs=yes output element stress shapes to op2 file
$
ADAMSMNF flexbody=yes, psetid=7772, admout=yes,
        outstrs=yes
$
$ Select real eigen value parameters - must be above or in first subcase
$
```

```

METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
$ Generate stress and strain grid shapes
$
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
$ Define surface for stress and strain grid shapes
$
OUTPUT(POST)
SET 9998 = ALL
SURFACE 9998 SET 9998 NORMAL X3
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$

```

```
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
ASET1,123456,1,11,111,121
$
$ SCALAR Point and QSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80018
QSET1,0,80001,THRU,80018
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ Get model data and load data
$
include 'modell1.dat'
$
ENDDATA
```

### Example 3 - Component Modeled as a Part Superelement

This example represents a MSC.Nastran/ADAMS interface run with the component modeled as part superelement 200 as defined by the BEGIN BULK SUPER = 200 entry. Grids listed in this section represent the complete component as a substructure (including in this case grid point 1, grid point 11, grid point 111, and grid 121). The **main Bulk Data Section** identified by BEGIN BULK includes GRID entries for grid point 1, grid point 11, grid point 111, and grid point 121 that identify the boundaries and are exterior to the part super element, and are used as the attachment points for ADAMS.

The ADAMSMNF Case Control command initiates the interface run with the **required** FLEXBODY=YES entry. Also included on this entry is the optional PSETID=7772 that provides a set of PLOTTEL elements whose grids are retained in the MNF and whose connectivity defines face geometry for ADAMS display. The OUTPUT(PLOT) section is used to define the set of elements for the PSETID. For this case, PLOTTELs were used and these are defined at the end of the Bulk Data Section.

Grid point stress and strain shapes are desired; thus, the sequence of STRESS, STRAIN, GPSTRESS, GPSTRAIN commands. The (PLOT) item suppresses output of these items to the f06 file. The OUTPUT(POST) section is used to define the surface for the grid point stress and strain requests.

The DTI,UNITS in the Bulk Data Section is **required** for the interface run and **must be placed in the main bulk section**. This defines the units to be used by ADAMS/Flex.

Standard SUBCASE is used to define two pressure load conditions. No constraints are applied to the component; hence, the modal analysis will represent a free structure.

The PARAM,WTMASS is not recommended with interface runs. This is noted by explicitly setting it to its default value of 1.

The PARAM,GRDPNT will cause the MSC.Nastran grid point weight generator (GPWG) to run and display a summary of the mass properties of the model. Any physical grid location may be chosen for the display purpose. However, with the ADAMS interface, if PARAM,GRDPNT is present, GPWG will be run a second time, **always** at the origin of the basic system and its results will be used for mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  computations for MNF export.

The SENQSET Bulk Data entry is used to define the component modes and **must be placed in the main Bulk Data Section**. This entry must specify sufficient degrees of freedom or modal amplitudes to capture the shape of the component and residual flexibility for any loading conditions. Instead of the SENQSET in the main Bulk Data Section, SPOINT and QSET1 Bulk Data entries could have been used in the BEGIN BULK SUPER = 200 section.

Note that the EIGR entry occurs in both the BEGIN BULK and BEGIN BULK SUPER = 200 sections.

```
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
```

```
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
$ Generate stress and strain grid shapes
$
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
$ Define surface for stress and strain grid shapes
$
OUTPUT(POST)
SET 9998 = ALL
SURFACE 9998 SET 9998 NORMAL X3
$
BEGIN BULK
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
GRID      1             0.0      0.0      0.0
GRID     11             0.0      1.0      0.0
GRID    111             1.0      0.0      0.0
GRID    121             1.0      1.0      0.0
$
$ SENQSET to define DOFs to use for component modes -
$ must be in main Bulk Data Section
$
SENQSET 200      18
$
$ ADAMS REQUIRES following DTI -
$ with PARTS this must be in main Bulk Data Section
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
```

```

$ =====
$
BEGIN BULK SUPER = 200
$
$ This PART defines interior grids as superelement 200
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value -
$ ADAMS must use the DTI,UNITS defined in main Bulk Data Section
$
PARAM,WTMASS,1.0
$
$ Select number of modes - for PART (Substructure) 200
$
$ =====
EIGR      300      LAN                                10
$ =====
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ Get model data for PART 200 and load data
$
include 'modell1.dat'
$
ENDDATA

```

## Example 4 - Component Modeled as a Superelement with Preload

This example is the same as Example 1, but with the following additions because preload requires a static solution. Therefore, the following additions are needed:

In each of the SUBCASEs that define the pressure loads, SPC=100 entries have been added to supply static supports. Additionally, a SUBCASE 200 has been added with SPC=100 and LOAD=100 entries to define the preload condition. This condition is defined by the FORCE 100 entries added to the Bulk Data Section. The resultant differential stiffness matrix will add stiffness to modes with rotation about the y-axis

and z-axis and only four rigid body modes will be present. In this run, the FORCE entries should not be replaced with equivalent follower load FORCE 1 entries because the follower stiffness will move down stream with grids 1, 11, 111, and 121 and will not contribute to the component.

Also in the Bulk Data Section, SPC1 100 entries have been included to supply a static support system at the attachment points.

A SUBCASE 300 has been included that identifies the preload condition with the STATSUB(PRELOAD)=200 entry. Note that this subcase contains no SPC entry; hence, the modal analysis will represent a free - free solution.

```
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
SUBCASE 5
LABEL= one atu overpressure - uniform
spc=100 $
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
spc=100 $
load =50
SUBCASE 200
LABEL= static stiffening load in plane of plate use a preload
spc=100 $
load=100
$
$ Select preload and generate free-free component shapes
$
SUBCASE 300
LABEL= Select SC200 as preload - modes as free-free structure
STATSUB((PRELOAD) = 200
```

```

$
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Add in plate stiffening load
$
$ For an ADAMS/Flex run do not use follower
$ FORCE1 etc. entries with this example as the
$ loads at grids 1,11,111,121 will move
$ down stream with these boundary grids
$ and their contribution to the follower
$ stiffness will be lost to the compont
$
FORCE   100      111           500.    6.
FORCE   100      112           1000.   6.
FORCE   100      113           1000.   6.
FORCE   100      114           1000.   6.
FORCE   100      115           1000.   6.
FORCE   100      116           1000.   6.
FORCE   100      117           1000.   6.
FORCE   100      118           1000.   6.
FORCE   100      119           1000.   6.
FORCE   100      120           1000.   6.
FORCE   100      121           500.    6.
$
FORCE   100       1            500.   -6.
FORCE   100       2            1000.  -6.
FORCE   100       3            1000.  -6.
FORCE   100       4            1000.  -6.
FORCE   100       5            1000.  -6.
FORCE   100       6            1000.  -6.
FORCE   100       7            1000.  -6.
FORCE   100       8            1000.  -6.
FORCE   100       9            1000.  -6.
FORCE   100      10            1000.  -6.
FORCE   100      11            500.   -6.
$

```



```
$ static support set for preload -
$ note because of differential stiffness formulation,
$ only 4 zero modes now exist. The zero modes
$ for y-axis and z-axis rotation will be lost
$
SPC1      100      123      1
SPC1      100      13       11
SPC1      100      3       111
$
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$
$ Define interior grids as superelement 200
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
SESET,200,2,THRU,10
SESET,200,12,THRU,110
SESET,200,112,THRU,120
$
$ SCALAR Point and SEQSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80019
SEQSET1,200,0,80001,THRU,80019
$
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ SEELT to put element 1003 and 1004 into SE 200
$
SEELT      200      10003      10004
$
$ Get model data and load data
$
include 'modell.dat'
$
ENDDATA
```

## Example 5 - Component Modeled as a Residual only Run with Preload

This example is the same as Example 2, but due to the preload requirements for a static solution, the following additions are needed:

In each of the SUBCASEs that define the pressure loads, SUPORT1=100 entries have been added to supply static supports. Additionally, a SUBCASE 200 has been added, with SUPORT1=100 and LOAD=100 entries to define the preload condition. This condition is defined by the follower FORCE1 100 entries added to the Bulk Data Section. Also in the Bulk Data Section, SUPORT1 100 entries have been included to supply a static support system at the attachment points. This run requires the use of SUPORT1 entries rather than SPC or SPC1 entries in order to avoid conflict between s-set points defined via SPC or SPC1, and a-set points defined via the ASET1 entry.

A SUBCASE 300 has been included that identifies the preload condition with the STATSUB(PRELOAD)=200 entry. Note that this subcase contains no SUPORT1 entry; hence, the modal analysis will represent a free - free solution.

```
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
SUBCASE 5
LABEL= one atu overpressure - uniform
suport1=100 $
load=5
$
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
suport1=100 $
load =50
```

```

$
SUBCASE 200
LABEL= static stiffening load in plane of plate use a preload
suport1=100 $
load=100
$
$ Select preload and generate free-free component shapes
$
SUBCASE 300
LABEL= Select SC200 as preload - modes as free-free structure
STATSUB((PRELOAD) = 200
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Add in plate tensioning follower load
$
FORCE1 100      111      3000.   1      111
FORCE1 100      112      6000.   2      112
FORCE1 100      113      6000.   3      113
FORCE1 100      114      6000.   4      114
FORCE1 100      115      6000.   5      115
FORCE1 100      116      6000.   6      116
FORCE1 100      117      6000.   7      117
FORCE1 100      118      6000.   8      118
FORCE1 100      119      6000.   9      119
FORCE1 100      120      6000.  10      120
FORCE1 100      121      3000.  11      121
$
FORCE1 100      1      3000.  111      1
FORCE1 100      2      6000.  112      2
FORCE1 100      3      6000.  113      3
FORCE1 100      4      6000.  114      4
FORCE1 100      5      6000.  115      5
FORCE1 100      6      6000.  116      6
FORCE1 100      7      6000.  117      7
FORCE1 100      8      6000.  118      8
FORCE1 100      9      6000.  119      9

```

```

FORCE1  100      10      6000.  120      10
FORCE1  100      11      3000.  121      11
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
ASET1,123456,1,11,111,121
$
$ SCALAR Point and QSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80019
QSET1,0,80001,THRU,80019
$
$ Static support for preload
$
$ Note use of SUPORT1 entry and not SPC entry! This is to avoid
$ fatal conflict of grids 1, 11, and 111 of being in s-set and a-set
$ SUPORT1 places them in r-set which is a subset of a-set
$ SUPORT1 chosen over SUPORT as it is subcase selectable
$
SUPORT1 100      1      123
SUPORT1 100      11     13
SUPORT1 100     111     3
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ Get model data and load data
$
include 'modell.dat'
$
ENDDATA

```

## Example 6 - Component Modeled as a Part Superelement with Preload

This example is the same as Example 3, but because preload requires a static solution, the following additions are needed:

In each of the SUBCASEs that define the pressure loads, SPC=100 entries have been added to supply static supports. Additionally, a SUBCASE 200 has been added with SPC=100 and LOAD=100 entries to define the preload condition. This condition is defined by the follower FORCE1 100 entries added to the BEGIN BULK SUPER = 200 section. Also in the main BEGIN BULK Section, SPC1 100 entries have been included to supply a static support system at the attachment points. For this run, the SPC1 100 entries could have been placed in the BEGIN BULK SUPER = 200 Section.

A SUBCASE 300 has been included that identifies the preload condition with the STATSUB(PRELOAD)=200 entry. Note that this subcase contains no SPC entry, hence the modal analysis will represent a free - free solution.

```
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
SUBCASE 5
LABEL= one atu overpressure - uniform
spc=100 $
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
spc=100 $
load =50
SUBCASE 200
LABEL= static stiffening load in plane of plate use a preload
spc=100 $
load=100
$
$
$ Select preload and generate free-free component shapes
$
SUBCASE 300
```

```

LABEL= Select SC200 as preload - modes as free-free structure
STATSUB((PRELOAD) = 200
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
BEGIN BULK
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
GRID      1      0.0      0.0      0.0
GRID     11      0.0      1.0      0.0
GRID    111      1.0      0.0      0.0
GRID    121      1.0      1.0      0.0
$
$ static support set for preload
$
SPC1     100     123      1
SPC1     100     13       11
SPC1     100      3      111
$
$ SENQSET to define DOFs to use for component modes -
$ must be in main Bulk Data Section
$
SENGSET 200      19
$
$ ADAMS REQUIRES following DTI -
$ with PARTS this must be in main Bulk Data Section
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$
BEGIN BULK SUPER = 200
$
$ This PART defines interior grids as superelement 200
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value -
$ ADAMS must use the DTI,UNITS defined in main Bulk Data Section
$

```

```
PARAM,WTMASS,1.0
$
$ Add in plate tensioning follower load
$
FORCE1 100 111 3000. 1 111
FORCE1 100 112 6000. 2 112
FORCE1 100 113 6000. 3 113
FORCE1 100 114 6000. 4 114
FORCE1 100 115 6000. 5 115
FORCE1 100 116 6000. 6 116
FORCE1 100 117 6000. 7 117
FORCE1 100 118 6000. 8 118
FORCE1 100 119 6000. 9 119
FORCE1 100 120 6000. 10 120
FORCE1 100 121 3000. 11 121
$
FORCE1 100 1 3000. 111 1
FORCE1 100 2 6000. 112 2
FORCE1 100 3 6000. 113 3
FORCE1 100 4 6000. 114 4
FORCE1 100 5 6000. 115 5
FORCE1 100 6 6000. 116 6
FORCE1 100 7 6000. 117 7
FORCE1 100 8 6000. 118 8
FORCE1 100 9 6000. 119 9
FORCE1 100 10 6000. 120 10
FORCE1 100 11 3000. 121 11
$
$ Select number of modes - for PART (Substructure) 200
$
$ =====
EIGR 300 LAN 10
$ =====
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
$ Get model data for PART 200 and load data
$
include 'modell1.dat'
$
ENDDATA
```

## Example 7a - Nonlinear Static Analysis of Component for Standard Restart

This example represents a MSC.Nastran nonlinear SOL 106 run with the component a standard residual structure.

SUBCASE 200 defines with SPC=100 and LOAD=100 entries, a preload condition or stiffening load condition. This condition is defined by the follower FORCE1 100 entries added to the Bulk Data Section. Also in the Bulk Data Section, SPC1 100 entries have been included to supply a static support system at the plate corner points; these will become the attachment points in the restart SOL 103 run.

The NLPARM Case Control command selects the NLPARM Bulk Data entry to define non linear parameters.

The PARAM,LGDISP,1 is required to turn on large displacement.

The SPCF(PLOT) = ALL generates forces of constraints. These will be used to form the total reactive preload in SOL 103.

SPOINT entries are **required** to be in the nonlinear run. They have no effect on the nonlinear solution but are place holders for the restart. The associated QSET1 entries **must not** appear in the nonlinear run as they will cause SOL 106 to fail because by current definition in MSC.Nastran, the q-set and a-set are synonyms and nonlinear requires a physical a-set.

When the job is submitted, SCR=NO must be invoked to insure that the data base is saved.

```
$ The data base must be saved for this run therefore SCR=NO required
SOL 106
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Get nonlinear stress output
$
NLSTRESS = ALL
$
SUBCASE 200
LABEL= static stiffening load in plane of plate for preload
SPCF(PLOT) = ALL $ Generate forces of constraint
SPC = 100 $
LOAD=100
$
$ Select nonlinear parameters
$
NLPARM = 1
$
```



```

BEGIN BULK
$
$ Turn on large displacements
$
PARAM,LGDISP,1
$
$ Nonlinear parameters
$
NLPARM,1,4,,,,,UPW,YES
$
$ ADAMS REQUIRES following DTI
$ either in nonlinear run or Sol 103 run but not both
$
$DTI,UNITS,1,KG,N,M,SEC
$
$ If wanted, turn on gridpoint weight generator
$ either in nonlinear run or Sol 103 run but not both
$
$PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Add in plate tensioning follower load
$
FORCE1 100      111      3000.   1      111
FORCE1 100      112      6000.   2      112
FORCE1 100      113      6000.   3      113
FORCE1 100      114      6000.   4      114
FORCE1 100      115      6000.   5      115
FORCE1 100      116      6000.   6      116
FORCE1 100      117      6000.   7      117
FORCE1 100      118      6000.   8      118
FORCE1 100      119      6000.   9      119
FORCE1 100      120      6000.  10      120
FORCE1 100      121      3000.  11      121
$
FORCE1 100      1      3000.  111      1
FORCE1 100      2      6000.  112      2
FORCE1 100      3      6000.  113      3
FORCE1 100      4      6000.  114      4
FORCE1 100      5      6000.  115      5
FORCE1 100      6      6000.  116      6
FORCE1 100      7      6000.  117      7
FORCE1 100      8      6000.  118      8
FORCE1 100      9      6000.  119      9
FORCE1 100     10      6000.  120     10
FORCE1 100     11      3000.  121     11
$
$
$ static support set for preload
$
SPC1    100      123      1

```

```

SPC1      100      13      11
SPC1      100      3       111
$
$
$ =====
$
$ scalar points must be added here for modal sol103 nmloop restart
$ because to add them in the modal 103 run would cause restart
$ to generate whole model over since changing grids and elements
$ flags model as new
$ The Qsets and Asets must be added to the sol103 run
$ They can not exist in the sol106 run
$
SPOINT,80001,THRU,80019
$
$ Get model data and load data
$
include 'modell.dat'
$
ENDDATA

```

## Example 7b - Nonlinear Static Analysis Restart into SOL103

This example represents a restart from SOL 106 into SOL 103 to perform a MSC.Nastran/ADAMS interface run with the component modeled as residual only structure. Grid point 1, grid point 11, grid point 111, and grid point 121 are to be used as the attachment points for ADAMS. This is accomplished by use of the ASET1 Bulk Data entry included in this file. The **nonlinear run cannot contain** this ASET entry.

The restart is indicated by the presence of the ASSIGN MASTER and RESTART file management commands in the file management section of the MSC.Nastran input stream.

The PARAM,NMLOOP in Case Control selects the converged nonlinear solution to use for the restart.

The ADAMSMNF Case Control command initiates the interface run with the **required** FLEXBODY=YES command. Also included on this entry is the optional PSETID=7772 which provides a set of PLOTTEL elements whose grids are retained in the MNF and whose connectivity defines the face geometry for ADAMS display. The OUTPUT(PLOT) section is used to define the set of elements for the PSETID. For this case, PLOTTELs were used. These are defined at the end of the Bulk Data Section.

The DTI,UNITS in the Bulk Data Section is **required** for the interface run. This defines the units to be used by ADAMS/Flex. This entry could have been placed in the SOL 106 run, but it cannot be placed in both because of standard MSC.Nastran restart requirements.

Standard SUBCASE is used to define two pressure load conditions. No constraints are applied to the component; hence, the modal analysis will represent a free structure.

The PARAM,WTMASS is not recommended with interface runs. This is noted by explicitly setting it to its default value of 1 in the SOL 106 run. Hence, it is not reinserted in the SOL 103 restart run.

The PARAM,GRDPNT will cause the MSC.Nastran grid point weight generator (GPWG) to run and display a summary of the mass properties of the model. Any physical grid location may be chosen for the display purpose. However, with the ADAMS interface, if PARAM,GRDPNT is present, GPWG will be run a second time, **always** at the origin of the basic system and its results will be used for mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  computations for MNF export.

The SPOINT and QSET1 Bulk Data entries are used to define the component modes. The SPOINT, **as required**, were placed in the SOL 106 run. The QSET1 is **required** to be placed in the restart SOL 103 run. These entries must specify sufficient degrees of freedom or modal amplitudes to capture the shape of the component and residual flexibility for any loading conditions.

```
$
$ Get the data base for the nonlinear run
$
ASSIGN MASTER='e7a_residual_106.MASTER'
$
$ Indicate a restart run
$
RESTART VERSION=1 KEEP
$
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
```

```

$
$ Select the desired converged nonlinear solution
$
PARAM,NMLOOP,4
$
$ include preload condition from sol 106 for labeling
$ must be first subcase
$
SUBCASE 2
LABEL=Sol 106 preload
load=100
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$ either in nonlinear run or Sol 103 run but not both
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator in this run
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$ Already done is Sol 106 run
$
$PARAM,WTMASS,1.0
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
$ =====
$
$ define attachment points
$
ASET1,123456,1,11,111,121
$
$ SCALAR Points supplied in sol106 run should match following QSET
$ QSET to define DOFs to use for normal modes
$

```

```

QSET1,0,80001,THRU,80019
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
ENDDATA

```

## Example 8a - Nonlinear Static Analysis of Component for dblocate Restart

This example represents a MSC.Nastran nonlinear SOL 106 run with the component a nonlinear residual structure.

SUBCASE 200 defines with SPC=100 and LOAD=100 entries, a preload condition or stiffening load condition. This condition is defined by the follower FORCE1 100 entries added to the Bulk Data Section. Also in the Bulk Data Section, SPC1 100 entries have been included to supply a static support system at the plate corner points which will become the attachment points in the restart SOL 103 run.

The NLPARM Case Control selects the NLPARM Bulk Data entry to define nonlinear parameters.

The PARAM,LGDISP,1 is required to turn on large displacement.

The SPCF(PLOT) = ALL generates forces of constraints. These will be used to form the total reactive preload in SOL 103.

No SPOINT entries (**required** for a standard restart from SOL 106 to SOL 103) are given. Thus, a nonstandard restart into SOL 103, using dblocate in SOL 103 will be provided. QSET1 entries **must not** appear in the nonlinear run. They will cause SOL 106 to fail because by current definition in MSC.Nastran, the q-set and a-set are synonyms and nonlinear requires a physical a-set.

```

$ The data base must be saved for this run therefore SCR=NO required
SOL 106
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Get nonlinear stress output
$
NLSTRESS = ALL

```

```

$
SUBCASE 200
LABEL= static stiffening load in plane of plate use a preload
SPCF(PLOT) = ALL $ Generate forces of constraint
SPC = 100 $
LOAD=100
$
$ Select nonlinear parameters
$
NLPARM = 1
$
BEGIN BULK
$
$ Turn on large displacements
$
PARAM,LGDISP,1
$
$ Nonlinear parameters
$
NLPARM,1,4,,,,,UPW,YES
$
$ ADAMS REQUIRES following DTI
$either in nonlinear run or Sol 103 run but not both
$
$DTI,UNITS,1,KG,N,M,SEC
$
$ If wanted, turn on gridpoint weight generator
$ either in nonlinear run or Sol 103 run but not both
$
$PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Add in plate tensioning follower load
$
FORCE1 100      111      3000.   1      111
FORCE1 100      112      6000.   2      112
FORCE1 100      113      6000.   3      113
FORCE1 100      114      6000.   4      114
FORCE1 100      115      6000.   5      115
FORCE1 100      116      6000.   6      116
FORCE1 100      117      6000.   7      117
FORCE1 100      118      6000.   8      118
FORCE1 100      119      6000.   9      119
FORCE1 100      120      6000.  10      120
FORCE1 100      121      3000.  11      121
$
FORCE1 100      1      3000.  111     1
FORCE1 100      2      6000.  112     2
FORCE1 100      3      6000.  113     3
FORCE1 100      4      6000.  114     4
FORCE1 100      5      6000.  115     5

```

```

FORCE1  100      6      6000.  116      6
FORCE1  100      7      6000.  117      7
FORCE1  100      8      6000.  118      8
FORCE1  100      9      6000.  119      9
FORCE1  100     10      6000.  120     10
FORCE1  100     11      3000.  121     11
$
$ static support set for preload
$
SPC1     100     123      1
SPC1     100     13      11
SPC1     100      3     111
$
$ =====
$
include 'modell.dat'
$
ENDDATA

```

## Example 8b - Nonlinear Static Analysis dblocate Restart into SOL103

This example represents a restart for SOL 106 into SOL 103 to perform a MSC.Nastran/ADAMS interface run with the component modeled as a residual only structure. Grid point 1, grid point 11, grid point 111, and grid point 121 are to be used as the attachment points for ADAMS. This is accomplished by use of the ASET1 Bulk Data entry inserted in this file. The **nonlinear run cannot contain** this ASET entry

Because the SOL 106 run did not contain SPOINT Bulk Data entries, the nonstandard restart is indicated by the presence of the ASSIGN MASTER and DBLOCATE file management statements in the File Management Section of the MSC.Nastran input stream.

The PARAM,NMLOOP in case control selects the converged nonlinear solution to use for the restart.

The ADAMSMNF Case Control command initiates the interface run with the **required** FLEXBODY=YES entry. Also include on this entry is the optional PSETID=7772 that provides a set of PLOTTEL elements whose grids are retained in the MNF and whose connectivity defines face geometry for ADAMS display. The OUTPUT(PLOT) section is used to define the set of elements for the PSETID. For this case, PLOTTELs were used and these are defined at the end of the Bulk Data Section.

The DTI,UNITS in the Bulk Data Section is **required** for the interface run. This defines the units to be used by ADAMS/Flex. This entry could have been placed in the SOL 106 run but it cannot be placed in both because of standard MSC.Nastran restart requirements.

Standard SUBCASE is used to define two pressure load conditions. No constraints are applied to the component. Hence, the modal analysis will represent a free structure.

The PARAM,WTMASS is not recommended with interface runs. This is noted by explicitly setting it to its default value of 1 in the SOL 106 run. Hence, it is not reinserted in the SOL 103 restart run.

The PARAM,GRDPNT will cause the MSC.Nastran grid point weight generator (GPWG) to run and display a summary of the mass properties of the model. Any physical grid location may be chosen for the display purpose. However, with the ADAMS interface if PARAM,GRDPNT is present, GPWG will be run a second time **always** at the origin of the basic system, and its results will be used for mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  computations for MNF export.

The SPOINT and QSET1 Bulk Data entries are used to define the component modes. The SPOINT entry is placed in this restart run and is what causes the use of a DBLOCATE nonstandard restart run. The QSET1 must be placed in the restart SOL 103 run. These entries must specify sufficient degrees of freedom or modal amplitudes to capture the shape of the component and residual flexibility for any loading conditions.

```
$
$ Get the data base for the nonlinear run
$
ASSIGN MASTER='e8a_residual_106_db.MASTER'
$
$ A dblocate run
$
dblocate datablk=(bulk,estnl,ugni,
                  qgni,qmgni,pgni) where(version=1) logi=master
$
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$
ADAMSMNF flexbody=yes, psetid=7772
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2004, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
```



```
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
$ Select the desired converged nonlinear solution
$
PARAM,NMLOOP,4
$
$ include preload condition from sol 106 for labeling
$ must be first subcase
$
SUBCASE 2
LABEL=Sol 106 preload
load=100
$
$ Generate two modal loads
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
$ Define surface for stress and strain grid shapes
$
BEGIN BULK
$
param,respath,old
$
$ ADAMS REQUIRES following DTI
$ either in nonlinear run or Sol 103 run but not both
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator in this run
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$ Already done is Sol 106 run
$
$PARAM,WTMASS,1.0
$
$ Select number of modes:
$
$ =====
EIGR      300      LAN                                10
```

```

$ =====
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
ASET1,123456,1,11,111,121
$
$ SCALAR Point and QSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80019
QSET1,0,80001,THRU,80019
$
$ Define plotel's for set 7772 in output(plot) section
$
PLOTTEL,10001,1,12
PLOTTEL,10002,12,121
PLOTTEL,10003,121,111
PLOTTEL,10004,111,1
PLOTTEL,10006,2,10
PLOTTEL,10007,22,110
PLOTTEL,10008,120,112
PLOTTEL,10009,12,100
$
ENDDATA

```

## Model Data for the Examples

The following data defines the plate model data.

```

$
$ START GRID SPECS
$
GRID, 1,, 0.0, 0.0, 0.0
=,*11,,*.10,==
=9
GRID, 2,, 0.0, 0.1, 0.0
=,*11,,*.10,==
=9
GRID, 3,, 0.0, 0.2, 0.0
=,*11,,*.10,==
=9
GRID, 4,, 0.0, 0.3, 0.0
=,*11,,*.10,==
=9
GRID, 5,, 0.0, 0.4, 0.0
=,*11,,*.10,==
=9
GRID, 6,, 0.0, 0.5, 0.0
=,*11,,*.10,==
=9
GRID, 7,, 0.0, 0.6, 0.0
=,*11,,*.10,==
=9

```

```
GRID, 8,, 0.0, 0.7, 0.0
=,*11,*,*10,==
=9
GRID, 9,, 0.0, 0.8, 0.0
=,*11,*,*10,==
=9
GRID, 10,, 0.0, 0.9, 0.0
=,*11,*,*10,==
=9
GRID, 11,, 0.0, 1.0, 0.0
=,*11,*,*10,==
=9
$
$ START ELEMENT SPECS
$
CQUAD4, 1,1,1,2,13,12
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 11,1,2,3,14,13
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 21,1,3,4,15,14
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 31,1,4,5,16,15
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 41,1,5,6,17,16
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 51,1,6,7,18,17
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 61,1,7,8,19,18
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 71,1,8,9,20,19
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 81,1,9,10,21,20
=,*1,=,*11,*11,*11,*11
=8
CQUAD4, 91,1,10,11,22,21
=,*1,=,*11,*11,*11,*11
=8
$
$ ELEMENT PROPERTIES INPUT
$
PSHELL,1,1,0.010,1
$
$ MATERIAL PROPERTIES INPUT
$
MAT1,1,.2223+12,.0855+12,0.3,7803.0
$
```

```
$ Modal load set 5: one atu overpressure - uniform
$
PLOAD45      1      101344.          THRU  100
$
$
$ Modal load set 50: spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
$
RFORCE,50,1,,1.0,0.0,1.0,0.0
$
$
```

CHAPTER  
**12**

## Aeroelasticity

- Correction of a Machine Dependency in the ZONA51 Supersonic Aerodynamic Method for Planar and Nonplanar Configurations

## 12.1 Correction of a Machine Dependency in the ZONA51 Supersonic Aerodynamic Method for Planar and Nonplanar Configurations

### Introduction

A recent wing-tail-vertical fin air vehicle configuration has shown that the ZONA51 supersonic aerodynamic method may yield slightly different results on different computer platforms.

A new algorithm has now been incorporated in ZONA51 to remove this sensitivity. This fix is available in MSC.Nastran Versions 2001.0.9 and newer.

Broadly, there appear to be two solutions, but these solutions are not associated with the usual numerical differences among machines (e.g., IEEE vs. byte swapped). Rather, these are more related to the level of accuracy of the mantissa in floating point calculations and may also be related to (although not corrected by) specific compiler optimization strategies. MSC.Nastran Versions 2001.0.7 and earlier exhibit this behavior.

This code error was found in the algorithm that computes the intersection between the downstream and upstream Mach cone from a sending control point to a receiving control point. The particular problem that exposed the ZONA51 error was a case in which the receiving box was on the vertical fin and the Mach cone intersection of a sending box was on the wing. The interference term generated by this algorithm becomes sensitive to computer round off error in the subsequent decomposition and solution of the aerodynamic influence coefficients. Consequently, both steady and unsteady results may be affected, although, to date, the steady aerodynamics effects are negligible.

For some nonplanar configurations, flutter solutions that are generated with the new algorithm may differ by up to 5% when compared to results from the old algorithm. While the initial case was a nonplanar configuration, results in planar cases are also affected (because the Mach cone effects are present in all cases—even for single panel test cases). For the planar configurations, these two algorithms appear to yield practically the same result in the aerodynamics, but have generated “numerically notable” differences in flutter solutions (that is, automated testing criteria for differences greater than 0.1% are triggered).

We recommend that users rerun prior cases to ensure that they do not encounter unexpected differences from the ZONA51’s aerodynamic method.

## Benefits/Improvements

### Example 1: Flutter Results

To illustrate the change in ZONA51 aerodynamic data, flutter results are shown in [Figure 12-1](#). New and old V-G curves are presented for AIX and HP platforms.

---

**Note:** Even though the AIX and HP platforms were used to illustrate this problem, the error exists across all platforms. This problem is *not* a hardware issue, but a software error.

---

The plot shows that the old results are machine dependent. It also shows that the new results are different than the old results, but not machine dependent. The new results are considered to be more accurate than either of the old answers—both of which were affected by numerical sensitivities introduced by the error in computing interference effects in ZONA51.

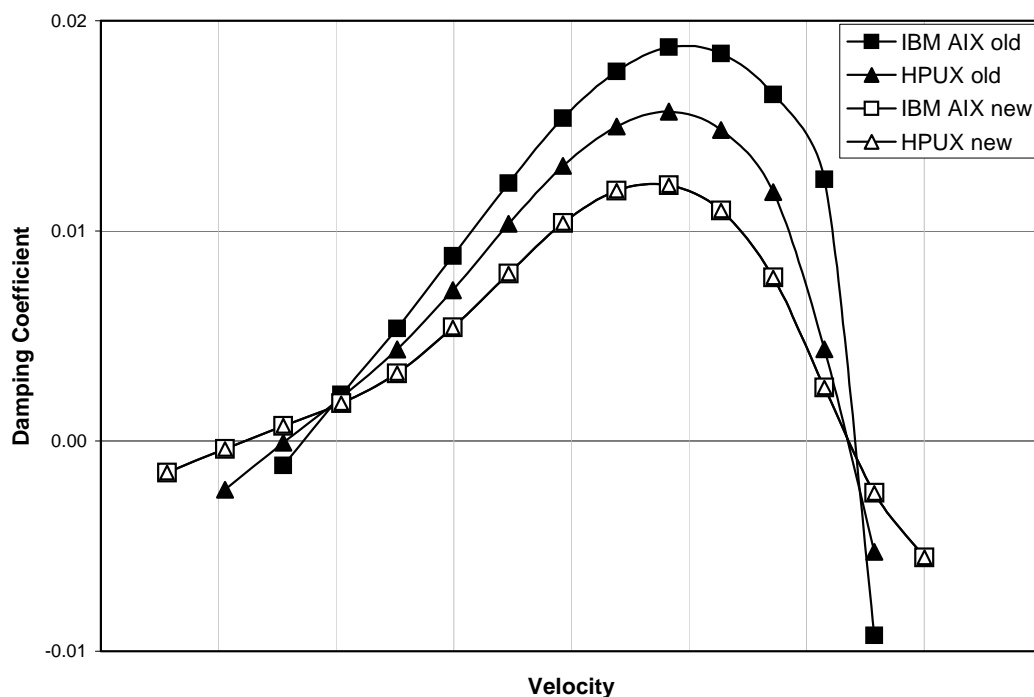
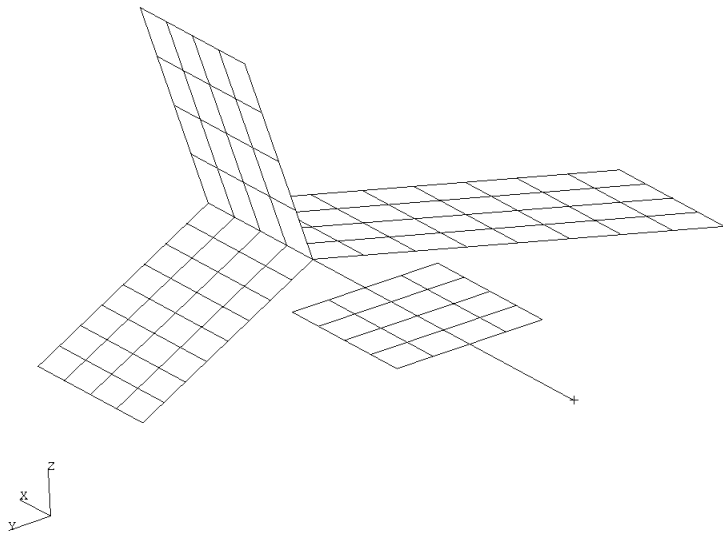


Figure 12-1 Comparison of Flutter Results

### Example 2: Dynamic Stability Derivatives

For the second example, dynamic stability derivatives<sup>10</sup> were calculated for the model shown in [Figure 12-2](#). The model was analyzed at Mach=1.2. The derivatives and their computed differences are listed in [Table 12-1](#). The reduced frequency for the results

presented in the table was 0.01. This table shows that results for MSC.Nastran Version 2001.0.7 are machine dependent and that results for MSC.Nastran Version 2001.0.9 may be different from results for 2001.0.7, but are not machine dependent.



**Figure 12-2 Model used to calculate dynamic stability derivatives.**

**Table 12-1 Dynamic Stability Derivatives**

	2001.0.7		2001.0.9		2001.0.7 vs 2001.0.09		AIX vs HP	
	AIX	HP	AIX	HP	% Difference AIX	% Difference HP	% Difference 2001.0.7	% Difference 2001.0.9
$C_{z_{\alpha}}$	0.01	0.01	0.01	0.01	0.00%	0.00%	0.00%	0.00%
$C_{m_{\alpha}}$	0.00	0.00	0.00	0.00	0.00%	0.00%	0.00%	0.00%
$C_{z_{\dot{\alpha}}}$	1193.10	1193.30	1193.30	1193.30	0.02%	0.00%	0.02%	0.00%
$C_{m_{\dot{\alpha}}}$	-12.93	-12.93	-12.93	-12.93	0.02%	0.00%	0.02%	0.00%
$C_{z_q}$	-1193.82	-1194.02	-1194.02	-1194.02	0.02%	0.00%	0.02%	0.00%



Table 12-1 Dynamic Stability Derivatives

	2001.0.7		2001.0.9		2001.0.7 vs 2001.0.09		AIX vs HP	
	AIX	HP	AIX	HP	% Difference AIX	% Difference HP	% Difference 2001.0.7	% Difference 2001.0.9
$C_{m_q}$	0.06	0.06	0.06	0.06	0.30%	0.00%	0.35%	0.00%
$C_{y_\beta}$	-8.22	-8.22	-8.22	-8.22	0.00%	0.00%	0.00%	0.00%
$C_{y_{\dot{\beta}}}$	152.60	148.42	147.96	147.97	3.04%	0.30%	2.74%	0.00%
$C_{l_\beta}$	-0.07	-0.07	-0.07	-0.07	0.00%	0.00%	0.00%	0.00%
$C_{l_{\dot{\beta}}}$	5.43	5.75	5.78	5.78	6.43%	0.49%	5.91%	0.00%
$n_\beta$	0.03	0.03	0.03	0.03	0.00%	0.00%	0.00%	0.00%
$C_{n_\beta}$	-4.00	-3.80	-3.78	-3.78	5.58%	0.51%	5.09%	0.01%
$C_{y_p}$	0.20	0.20	0.20	0.20	0.10%	0.05%	0.05%	0.00%
$C_{y_{\dot{p}}}$	-3.65	-3.11	-2.69	-2.69	26.22%	13.56%	14.65%	0.00%
$C_{l_p}$	-0.01	-0.01	-0.01	-0.01	0.00%	0.00%	0.00%	0.00%
$C_{l_{\dot{p}}}$	0.05	0.05	0.05	0.05	2.94%	0.75%	2.21%	0.00%
$C_{n_p}$	-0.01	-0.01	-0.01	-0.01	0.00%	0.00%	0.00%	0.00%
$C_{n_{\dot{p}}}$	0.12	0.10	0.09	0.09	19.62%	10.10%	10.59%	0.00%
$C_{y_r}$	152.67	148.48	148.03	148.04	3.04%	0.30%	2.74%	0.00%

Table 12-1 Dynamic Stability Derivatives

	2001.0.7		2001.0.9		2001.0.7 vs 2001.0.09		AIX vs HP	
	AIX	HP	AIX	HP	% Difference AIX	% Difference HP	% Difference 2001.0.7	% Difference 2001.0.9
$C_{l_r}$	-5.42	-5.74	-5.77	-5.77	6.44%	0.49%	5.92%	0.00%
$C_{n_r}$	-4.00	-3.80	-3.78	-3.78	5.57%	0.51%	5.08%	0.01%

# CHAPTER 13

## Miscellaneous Enhancements

---

- Enhancements to MATMOD Module Option 1
- Enhancements to MATMOD Module Option 2
- New PARAML Module Options For Modifying Table Data Blocks Via DMAP
- Option to Reduce Post Processing Data (POST Case Control Command)
- Additional Describers for the ASSIGN Statement
- Option to Print the CD Value in the F06 File
- Enhancements in the Specification of Parameters
- Enhancement for the DMAP String-based Editor
- Consistent Parameter Checking
- Punch Output Enhancement
- Scale Factor for X2GG and X2PP Matrices
- Direct Input of Structural Element Damping and Area Matrices
- MSC.Access, DDLADD Revisions, and XDB Defects Correction
- Enhanced Free Field Format
- Dynamic Element Forces
- Sparse Data Recovery

## 13.1 Enhancements to MATMOD Module Option 1

### Introduction

Option 1 in the MATMOD module was originally designed to extract a single specified column from a matrix. While this feature is useful, it is limited in its scope and applicability. In MSC.Nastran 2004 this option has now been enhanced so that the user can extract a range of columns with an increment. Furthermore, the user also has the option of obtaining a complementary matrix that contains all of the columns that are not extracted as above. This latter feature, in effect, can therefore be used to partition a matrix in a certain way without using explicit partitioning vectors. The details of the enhancement are given below.

### Input

The old usage of MATMOD module option 1 was:

```
MATMOD      MATIN, , , , /MATOUT, /1/P2 $
```

where MATIN is the input matrix and MATOUT is a single column output matrix that contains the P2-th column of MATIN.

There are two forms for the new usage of MATMOD module option 1. The first form is:

```
MATMOD      MATIN, , , , /MATOUT, /1/P2/P3/P4 $
```

In this case, matrix MATOUT will, in general, have multiple columns of MATIN as per the following scheme:

P2 (input-integer)	First column of MATIN that is to be extracted
P3 (input-integer)	Last column to be considered in the extraction process
P4 (input-integer)	Increment to be applied to P2 for subsequent columns to be extracted

The net result is that MATOUT will contain those columns of MATIN whose column numbers are  $P2$ ,  $P2 + P4$ ,  $P2 + 2 \cdot P4$ ,  $P2 + 3 \cdot P4$ , ... until the last column number  $\leq P3$ . Whether the last column extracted is less than or equal to  $P3$  is determined by the first column number  $P2$ , the increment  $P4$ , and the number of columns ( $N$ ) of MATIN.

The second form for the new usage of MATMOD module option 1 is:

```
MATMOD      MATIN, , , , /MATOUT1, MATOUT2/1/P2/P3/P4 $
```

In this case, matrix MATOUT1 will be generated just like MATOUT in the first form. However, in this case, the program will also generate matrix MATOUT2 which will contain those columns of MATIN that are NOT in MATOUT1. In effect, this form can be used to partition MATIN as per the scheme outlined above.

To facilitate the new usage and also preserve legacy usage, parameters P3 and P4 (which must both be non-negative integers), are handled as follows:

### Parameter P3

$P3 > 0$

If  $P3 < P2$ , P3 is assumed to be P2.

If  $P3 > N$ , P3 is assumed to be N.

$P3 = 0$  (default) or not specified

If  $P4 = 0$  (default) or not specified, P3 is assumed to be P2. This, in conjunction with the assumed value of 1 for P4 in this case (see below), preserves legacy usage.

If  $P4 > 0$ , then P3 is assumed to be N.

### Parameter P4

If  $P4 = 0$  (default) or is not specified, P4 is assumed to be 1.

## Examples

### Example 1

```
MATMOD      MATIN,,,,,/MATOUT,/1/4 $
```

This is exactly like the old usage. In this case, we have  $P2 = 4$ . Because neither P3 nor the increment P4 is specified, the program assumes  $P3 = P2$  and  $P4 = 1$ , resulting in just the 4<sup>th</sup> column of MATIN being extracted, thus preserving legacy usage.

### Example 2

```
MATMOD      MATIN,,,,,/MATOUT,/1/3/8/2 $
```

In this case, MATOUT will contain columns 3, 5 and 7 of MATIN.

### Example 3

```
MATMOD      MATIN,,,,,/MATOUT,/1/5/9 $
MATMOD      MATIN,,,,,/MATOUT,THEREST/1/5/9 $
```

In this case, because the increment P4 is not specified, the program assumes  $P4 = 1$  and MATOUT will therefore contain columns 5 through 9 of MATIN. In the second form, matrix THEREST will contain those columns of MATIN that are not in MATOUT.

### Example 4

```
$ PARTITION MATIN INTO ODD NUMBERED AND
$ EVEN NUMBERED COLUMNS
MATMOD    MATIN , , , , /ODDCOLS , EVENCOLS /1/1//2 $
```

or

```
MATMOD    MATIN , , , , /EVENCOLS , ODDCOLS /1/2//2 $
```

In this case, because P3 is not specified and  $P4 > 0$ , the program assumes  $P3 = N$ . Accordingly, matrix ODDCOLS will contain the odd numbered columns of MATIN and matrix EVENCOLS will contain the even numbered columns of MATIN.

### Example 5

Assume UDVT is the matrix resulting from a transient analysis solution, containing the displacement-velocity-acceleration triplet of columns for all time steps.

```
$ EXTRACT DISPLACEMENT COLUMNS FOR ALL TIME STEPS
MATMOD    UDVT , , , , /DISP , /1/1//3 $
$ EXTRACT VELOCITY COLUMNS FOR ALL TIME STEPS
MATMOD    UDVT , , , , /VELO , /1/2//3 $
$ EXTRACT ACCELERATION COLUMNS FOR ALL TIME STEPS
MATMOD    UDVT , , , , /ACCE , /1/3//3 $
```

## 13.2 Enhancements to MATMOD Module Option 2

### Introduction

The matrix element filter capability for MATMOD option 2 prior to MSC.Nastran 2004 was based on the magnitude of the element. An option to also filter on the algebraic value of the element has been added to MSC.Nastran 2004. An existing undocumented option to discard the lower or upper triangle of the input matrix is also described.

### Input

Option p1=2

Various matrix filter options

### Format:

(New features and newly-documented features in lower case)

MATMOD I1,,,,,,/O1,,,,,,/2/PURGE/UPLOW/TYPEF/FILTER

### Input Data Block:

Any matrix, real or complex

### Output Data Block:

Copy of I1 with some terms set to 0.0, based on the values of the input parameters.

### Parameter:

PURGE	Input-integer-default=0. If PURGE=0, and the output matrix has no nonzero terms, then the output matrix is purged. If PURGE $\neq$ 0 and the the output matrix has no nonzero terms than the output matrix is null.
UPLOW	Input-integer-default=0. If UPLOW<0 then all lower triangular terms in O1 are set to zero. If UPLOW>0 then all upper triangular in O1 are set to zero. If UPLOW=0 then this parameter is ignored.
TYPEF	Input-integer-default=0. Determines the type of filtering

Value	Action
0	O1(i,j) is set to zero when $\text{abs}(I1(i,j)) < \text{abs}(\text{FILTER})$ . Magnitude filter
1	O1(i,j) is set to zero when $I1(I,j) < \text{FILTER}$ . Algebraic filter.
-1	O1(I,j) is set to zero when $I1(I,j) > \text{FILTER}$ . Algebraic, complement of prior option.

**Examples:**

Print terms in A larger in magnitude than 100.0

```
MATMOD A,,,,,/AFILTER,/2///100.0 $
MATPRN A// $
```

Print terms in the upper triangle of B that are less than -200.

```
TYPE PARM,,I,N,UPLOW=1,TYPEF=-1 $
MATMOD      B,,,,,/BFILTER,/2//UPLOW/TYPEF/-200. $
MATPRN      B// $
```



## 13.3 New PARAML Module Options For Modifying Table Data Blocks Via DMAP

### Introduction

New options have been added to the PARAML module to allow for the modification of table data blocks via DMAP. The new options allow the user to perform the following modifications to table data blocks via DMAP:

1. Replace any entry in any record (via the new TABREPx options).
2. Insert an entry in any record or insert a new record (via the new TABINSx options).
3. Delete any number of entries from any record, delete the trailing portion of any record or delete an entire record (via the new TABDEL option).

### Benefits

The above features provide an easy and elegant means for the user to make modifications to table data blocks via DMAP without resorting to the use of the TABEDIT module whose usage is rather cumbersome and not very user friendly.

The full details of the new capability are given later in this section under the discussion of the usage of the new options.

### Input

The format of the PARAML module as used in the following discussion is as follows:

```
PARAML      TABIN/TABOUT/new_option//  
            RECNUM/WORDNUM/REAL $
```

or

```
PARAML      TABIN/TABOUT/new_option//  
            RECNUM/WORDNUM//INTEG $
```

or

```
PARAML      TABIN/TABOUT/new_option//  
            RECNUM/WORDNUM/////CHAR $
```

The input and output data blocks and the parameters have the following meanings.

## Input Data Block

TABIN	Any table data block
-------	----------------------

## Output Data Block

ABOUT	Table data block exactly identical to TABIN (including its trailers) except for the modifications performed
-------	---

## Parameters

**RECNUM** Input-integer  $\geq 0$ . A record number of TABIN. If RECNUM is greater than the number of records in TABIN, it will result in a fatal error. (RECNUM=0 refers to the header record of TABIN.)

**WORDNUM** Input-integer. If this is non-zero, its absolute value references a word number in record **RECNUM**. If this value is greater than the number of words in record number **RECNUM**, it will result in a fatal error.

REAL	Input-real.
------	-------------

INTEG	Input-integer.
-------	----------------

CHAR	Input-character.
------	------------------

## Usage of the New PARAML Module Options

The usage of the new options is discussed below under three categories.

## Category 1 - Replace Any Entry in Any Record of a Table

(There are four options in this category. They are of the form TABREPx, for TABLE REPlacement.)

## Option TABREPR

**Purpose:** Replace any entry in any record by a real number

Usage:      PARAML    TABIN/TABOUT/"TABREPR"/  
             RECNUM/WORDNUM/REAL \$

Word number WORDNUM in record number RECNUM is replaced by real number REAL.

## Option TABREPI

Purpose: Replace any entry in any record by an integer number

Usage: PARAML TABIN/TABOUT/'TABREPI'/  
RECNUM/WORDNUM//INTEG \$

Word number WORDNUM in record number RECNUM is replaced by integer number INTEG.

## Option TABREPC

Purpose: Replace any entry in any record by the first four characters of a character parameter

Usage: PARAML TABIN/TABOUT/'TABREPC'/  
RECNUM/WORDNUM////CHAR \$

Word number WORDNUM in record number RECNUM is replaced by the first four characters of the character parameter CHAR.

## Option TABREP2C

Purpose: Replace any two consecutive entries in any record by the two 4-character values comprising a character parameter

Usage: PARAML TABIN/TABOUT/'TABREP2C'/  
RECNUM/WORDNUM////CHAR \$

Word numbers WORDNUM and WORDNUM+1 in record number RECNUM are replaced by the two 4-character values comprising the character parameter CHAR.

---

**Note:** For TABREPx options: It is the user's responsibility to ensure that the type of entry that is being replaced is the same as the type of entry that is replacing it. The program does not check for this condition.

---

# Category 2 - Insert an Entry in Any Record of a Table or Insert a New Record After Any Record of a Table

(There are four options in this category. They are of the form TABINSx, for TABLE INSection)

## Option TABINSR

Purpose:	Insert a real number in any record or insert a new record containing a real number
Usage:	PARAML    TABIN/TABOUT/'TABINSR'/ RECNUM/WORDNUM/REAL \$
WORDNUM > 0	Real number REAL is inserted AFTER word number WORDNUM in record number RECNUM.
WORDNUM < 0	Real number REAL is inserted BEFORE word number ABS(WORDNUM) in record number RECNUM.
WORDNUM = 0	A new record containing real number REAL is inserted AFTER record number RECNUM.

## Option TABINSI

Purpose:	Insert an integer number in any record or insert a new record containing an integer number
Usage:	PARAML    TABIN/TABOUT/'TABINSI'/ RECNUM/WORDNUM//INTEG \$
WORDNUM > 0	Integer number INTEG is inserted AFTER word number WORDNUM in record number RECNUM.
WORDNUM < 0	Integer number INTEG is inserted BEFORE word number ABS(WORDNUM) in record number RECNUM.
WORDNUM = 0	A new record containing integer number INTEG is inserted AFTER record number RECNUM.

## Option TABINSC

Purpose:	Insert the first four characters of a character parameter in any record or insert a new record containing the first four characters of a character parameter
Usage:	PARAML    TABIN/TABOUT/'TABINSC' / RECNUM/WORDNUM/////CHAR \$
WORDNUM > 0	The first four characters of CHAR are inserted AFTER word number WORDNUM in record number RECNUM.
WORDNUM < 0	The first four characters of CHAR are inserted BEFORE word number ABS(WORDNUM) in record number RECNUM.
WORDNUM = 0	A new record containing the first four characters of CHAR is inserted AFTER record number RECNUM.

## Option TABINS2C

Purpose:	Insert the two 4-character values comprising a character parameter in any record or insert a new record containing the two 4-character values comprising a character parameter.
Usage:	PARAML    TABIN/TABOUT/'TABINS2C' / RECNUM/WORDNUM/////CHAR \$
WORDNUM > 0	The two 4-character values of CHAR are inserted AFTER word number WORDNUM in record number RECNUM.
WORDNUM < 0	The two 4-character values of CHAR are inserted BEFORE word number ABS(WORDNUM) in record number RECNUM.
WORDNUM = 0	A new record containing the two 4-character values of CHAR is inserted AFTER record number RECNUM.

**Category 3 - Delete Any Number of Entries in Any Record of a Table, Delete the Trailing Portion of Any Record of a Table or Delete an Entire Record of a Table**

(There is only one option in this category. It is called TABDEL, for TABLE DEletion)

**Option TABDEL**

Purpose:	Delete any number of entries in any record, delete the trailing portion of any record or delete an entire record
Usage:	PARAML TABIN/TABOUT/'TABDEL'/ RECNUM/WORDNUM//NWORDS \$
WORDNUM > 0	A total of NWORDS entries starting from word number WORDNUM and going forward are deleted from record number RECNUM. If NWORDS is not specified, it is internally assumed to be 1.
WORDNUM = 0	The entire record number RECNUM is deleted. NWORDS is ignored in this case.
WORDNUM < 0	The trailing portion of record number RECNUM starting from word number ABS(WORDNUM) and going forward is deleted. NWORDS is ignored in this case.

---

**Note:** The previous usage implies that the use of WORDNUM=0 and WORDNUM=-1 will both result in the deletion of the entire record number RECNUM.

---

## 13.4 Option to Reduce Post Processing Data (POST Case Control Command)

### Introduction

For MSC.Nastran 2004, the POST Case Control command has been added that gives the user some subcase-level control over the amount of output data stored on the postprocessing file.

Finite element analysis of large structural components sometimes requires the consideration of many different loading scenarios to complete the simulation. This can be especially true for static analysis, that may require evaluation of several thousand different load cases. During this process, recovery and postprocessing of large amounts of response data usually occurs. Many analysis organizations use commercially available pre and postprocessors such as MSC.Patran to graphically display results data. These programs typically read the results data from a FORTRAN file generated by the OUTPUT2 module in MSC.Nastran. This file contains all of the output requested by the case control data recovery requests in each and every subcase. Occasionally, output data are required for a subcase that does not need to be handled by the postprocessing program. Until now, there has been no convenient way to eliminate the output data for a particular subcase from the data file that is read by the postprocessor.

### Benefits and Features

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

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

The amount of data recovered for each subcase is dependent upon the present Case Control commands. If a request is placed above all subcases, that request applies to all of the subcases. Data are generated and produced one time for all subcases by examining the case control requests in each subcase. Each separate type of response is stored in its own data block. For example, displacement results are computed and stored separately from element stress results. These data blocks will include results for any subcase that contains a data recovery request with a print, plot or punch destination. The data blocks are then written to an external file by the OUTPUT2 module. There is no way to control the placement of the results data on the output file, except through the case control request for those data. For example, if there are three subcases and displacement results for the first and third subcases, but not the second subcase are required for postprocessing, one could simply request displacement output for only the first and third subcases. However, if punched displacement results for the second subcase are also required, displacements for all three subcases would be placed on the output file, even though results for the second subcase are not needed, nor wanted in the postprocessor. This applies to all output quantities requested by case control commands. The only way to handle this situation is to run the analysis twice and change the data recovery requests. A simple method is needed to exercise more control over the contents of the output file.

The new POST Case Control command provides better control over the contents of the output file produced by the MSC.Nastran program when a PARAM,POST Bulk Data entry is present in the input. It does not affect the contents of the results data blocks, only the amount of data that is transferred from those data blocks to the output file. This is accomplished by giving the OUTPUT2 module access to the case control requests for each subcase. The POST command is examined for the output requested in each subcase. Only that output requested by the POST command is placed on the output file for the subcase. Note that the data must be made available by the presence of a data recovery command in the subcase. For example, the POST command cannot output stress data if no stress data have been requested.

The operations associated with the POST Case Control command are closely related to the operations described under the PARAM POST discussion in “[Parameters](#)” on page 573 of the *MSC.Nastran Quick Reference Guide*. All of the DMAP parameters discussed in the *The MSC.Nastran Quick Reference Guide* related to the POST parameter are honored and can be used to control the output produced. The POST parameter value itself is supplied by the POST Case Control command through selection of one of the supported postprocessor programs shown in the following table.



ppname	Product	PARAM,POST,Value
PATRAN	MSC.Patran V3	-1
SDRC	SDRC IDEAS	-2
NF	MSC/LMS NF	-4
FEMTOOLS	DDS/FemTools	-5
UNIGRAHICS	EDS/Unigraphics	-6

## Inputs

The POST Case Control command is designed to provide a way to limit the output of data recovery results for a particular subcase. The general format of the POST command is:

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

The complete description of the POST command can be found in “[Statements, Commands, Entries, and Parameters](#)” on page 621. The POST command recognizes only a sub-set of all of the results data that can be produced by MSC.Nastran. This sub-set is summarized in the following table.

Output Item	oplist Keyword	Case Command
Displacements	[NO]DISPLACE	DISP
Forces of Single Point Constraint	[NO]SPCFORCE	SPCFORCE
Element Forces	[NO]FORCES	ELFO/FORCE
Element Stresses	[NO]STRESS	ELST/STRESS
Element Strain Energy	[NO]ESE	ESE
Grid Point Force Balance	[NO]GPFORCE	GPFORCE
Stress at Grid Points	[NO]GPSIGMA	STRESS
Strain/Curvature at Grid Points	[NO]GPEPSILON	STRAIN
Composite Element Failure Indices	[NO]PLYFAILURE	STRESS
Element Kinetic Energy	[NO]EKE	EKE
Element Energy Loss	[NO]EDE	EDE

Output Item	oplist Keyword	Case Command
Multi-point Constraint Forces	[NO]MPCFORCE	MPCFORCE
Composite Lamina Stresses	[NO]PLYSIGMA	STRESS
Composite Lamina Strains	[NO]PLYEPSILON	STRAIN
Element Strains	[NO]STRAIN	STRAIN
Grid Point Stresses	[NO]GPSTRESS	GPSTRESS
Grid Point Strains	[NO]GPSTRAIN	GPSTRAIN
Applied Loads	[NO]LOAD	OLOAD
No items to be output	NONE	-----

The POST command permits the optional selection of the target postprocessor program as well as the target fortran file unit reference number where the output will be stored.

## Output

There are no special output considerations when using the POST Case Control command.

## Guidelines and Limitations

The new POST Case Control command is intended primarily for static analysis. It is considered to be an extension of the parameters associated with the POST DMAP parameter discussed in “[Parameters](#)” on page 573 of the *MSC.Nastran Quick Reference Guide*. The POST command recognizes specification of any of the commercial postprocessor products currently supported by the MSC.Nastran solution sequences. The following guidelines should be kept in mind when using the POST command.

- Use of the POST command is entirely optional. If it is not used and a PARAM,POST,x entry is present in the input file, a postprocessor output file is generated just as in previous releases of the program.
- The default postprocessor program is MSC.Patran Version 3.0.
- If any POST command operations are to be performed, a POST command must be specified above all subcases.
- When a POST command is encountered, a “PARAM,POST,x” entry is inserted in the Case Control at that point. All other POST-related DMAP parameters are available to the user to control output of data as before.

- Once a particular postprocessor program is specified, it cannot be changed in any subsequent subcases.
- The POST command does not allow a change in specification of the output fortran unit reference number if it is associated with a form=formatted OUTPUT2 file.
- POST command options are limited to SORT1 formatted output data at the present time.
- As is presently the case, only data that have been generated by the presence of a case control data recovery request command will be available for postprocessing output.
- There is no control over any geometry data output or over any data blocks that are not generated from a case control request. These data blocks will continue to be placed on the OUTPUT2 unit specified by the OUNIT2 parameter, depending upon the value of the POST parameter (and the values of any related parameters).
- The placement of the POST command above the subcase level causes a cumulative effect on POST commands in subsequent subcases. Any options specified above the subcase level propagate down into the POST command within a subsequent subcase. Thus, if a POST command specifies NODISP (no displacement output wanted) above the subcase level, then a POST command with the DISP option would be required within a subcase to produce any output on the OUTPUT2 file for displacements. This also implies that changing the OUTPUT2 file name or unit reference number with the TOFILE option in a subcase causes all output quantities currently scheduled for output to be switched to the new unit number or file, not just those in the option list for the current POST command.

## Example

A simple model is presented to demonstrate the usage of the POST Case Control command to eliminate unwanted output from being placed on the postprocessor data file generated by MSC.Nastran. The model data itself are not important for this example. It is the subcase structure that is the essence of the discussion. As an example of how this new feature might be used, consider the case of a static simulation performed on a model requiring three load cases. The required output is displacements at three grid points in the model. Furthermore, the output for the second load case must also include punched displacement data for all grid points. Displacement results for subcases 100 and 300 are to be placed on the default

OUTPUT2 file for use by the MSC.Patran postprocessor. No output for subcase 200 is required for postprocessing by MSC.Patran. The subcase structure might look something like:

```

      SET 1000  =  10, 11, 12
      DISP = 1000
      $
      POST  TOFILE 12 $ use defaults: Patran v3.0
      $
      SUBCASE      100
      LOAD = 100
      $
      SUBCASE      200
      LOAD = 200
      DISP(PUNCH) = ALL
      POST  NODISP  $  stop any displacement output from going to POST file
      $
      SUBCASE      300
      LOAD = 300

```

In this example, the presence of the POST command above all of the subcases indicates that all output requested (DISP for SET 1000 is the only output requested and generated) is to be stored on the default OUTPUT2 fortran unit for use by MSC.Patran V3.0. The presence of the POST command in SUBCASE 200 with the NODISP option prevents the large output produced by the DISP(PUNCH)=ALL request in SUBCASE 200 from being placed on the OUTPUT2 file.

## Model Description

The model used for this example is very simple, as the intent is to focus on the subcase structure. It is a cantilever plate structure consisting of only six CQUAD4 shell elements and fourteen grid points. There are three subcases used to apply loads and recover displacements.

## Input File

```

$
$*****
$
$Minor Enhancement A01130/A01131 - new POST case control capability
$VERSION: 2002
$TEST DECK NAME: postrnex.dat
$
$PURPOSE:
$  Simple case control setup to demonstrate usage of the new POST Case
$  Control Command for the Release Notes
$
$DESCRIPTION:
$  An OP2 file is to be generated for the MSC.Patran V3.0 post-

```

```

$ processor. Three subcases are present in case control.
$ Displacement output is requested above the subcase level.
$ Displacements for subcases 100 and 300 are to be placed on the
$ default OP2 file. Displacements for subcase 200 are to be placed on
$ system punch file but not on the OP2 file.
$
$EXPECTED RESULTS:
$ The following UIM 4114 should be present in the .f06 output:
$
$*** USER INFORMATION MESSAGE 4114 (OUTPX2)
$ DATA BLOCK OUGV1 WRITTEN ON FORTRAN UNIT 12, TRL =
$ 101 0 160 0 0 0 0
$ (MAXIMUM POSSIBLE FORTRAN RECORD SIZE = xxxxx WORDS.)
$ (MAXIMUM SIZE OF FORTRAN RECORDS WRITTEN = 146 WORDS.)
$ (NUMBER OF FORTRAN RECORDS WRITTEN = 30 RECORDS.)
$ (TOTAL DATA WRITTEN FOR DATA BLOCK = 381 WORDS.)
$ (DATA FOR 2 CASE(S) WAS OUTPUT TO THIS UNIT. CASE ID LIST FOLLOWS.)
$ 100 300
$
$*****
$
ID MSC,POSTRNEX
SOL 101
TIME 5
CEND
TITLE = POST CASE CONTROL COMMAND EXAMPLE FOR RELEASE NOTES
SUBTITLE = DEMONSTRATE POST COMMAND USAGE
$
SPC = 1
SEALL = ALL
SET 1000 = 10,11,12
DISP = 1000
$
$ place post command above subcase level. Nothing specified causes
$ all defaults to be taken:
$ POST PATRAN TOFILE 12 (all outputs requested via CC commands)
$ or, for the example here where only DISP is requested,
$ POST PATRAN TOFILE 12 DISP
$ Note that POST command options specified above the subcase level
$ flow down into all of the subcases.
$
POST patran tofile 12
$
$-----
subcase 100
LOAD = 100
$-----
subcase 200
LOAD = 200
DISP(PUNCH)=ALL $ request punched displacement output for all grids
POST NODISP $ no displacement output to default OP2 file.
$-----
subcase 300
LOAD = 100

```

```

$-----
BEGIN BULK
CQUAD4  1      1      1      4      5      2
CQUAD4  2      1      5      6      3      2
CQUAD4  3      1      4      7      8      5
CQUAD4  4      1      8      9      6      5
CQUAD4  5      1      7     10     11      8
CQUAD4  6      1     11     12      9      8
$
PSHELL  1      100     0.001   100
MAT1     100     10.7+6      0.33
$
FORCE   1      10              1.          10.
FORCE   1      11              1.          20.
FORCE   1      12              1.          10.
GRID    1              0.      1.75      0.
GRID    2              0.      2.275     0.
GRID    3              0.      2.8        0.
GRID    4              0.6      1.75      0.
GRID    5              0.6      2.275     0.
GRID    6              0.6      2.8        0.
GRID    7              1.2      1.75      0.
GRID    8              1.2      2.275     0.
GRID    9              1.2      2.8        0.
GRID   10              1.8      1.75      0.
GRID   11              1.8      2.275     0.
GRID   12              1.8      2.8        0.
GRID   13              0.0      1.75      1.
GRID   14              1.0      1.75      1.
LOAD   100      1.      1.      1
LOAD   200      1.      2.      1
LOAD   300      1.      3.      1
PARAM  AUTOSPC  YES
SPC     1      1      13456
SPC     1      2      123456
SPC     1      3      13456
ENDDATA

```

## 13.5 Additional Describers for the ASSIGN Statement

### Introduction

Two additional describers (RECL and SIZE) have been added to the ASSIGN FMS statement.

### Input

The format of the revised ASSIGN statement is as follows:

$$\text{ASSIGN } \text{logical-key} = \text{'filename2'} \left[ \text{[STATUS} = \left\{ \begin{array}{c} \text{NEW} \\ \text{OLD} \\ \text{UNKNOWN} \end{array} \right\} \text{UNIT} = \text{u}, \right.$$

$$\text{FORM} = \left[ \begin{array}{c} \text{FORMATTED} \\ \text{UNFORMATTED} \end{array} \right] \text{TEMP DELETE SYS - 'sys-spec',}$$

$$\left. \text{RECL} = l, \text{SIZE} = s \right]$$

Describers	Meaning
log-name	The name of a DBset member name. log-name is also referenced on an INIT statement after the LOGICAL keyword.
filename <i>i</i>	The physical filename assigned to the DBset member.
TEMP	Requests that filename <i>i</i> be deleted at the end of the run.
DELETE	Requests that filename <i>i</i> , if it exists before the start of the run, is to be deleted.
logical-key	Specifies defaults for STATUS, UNIT, and FORM of FORTRAN files for other FMS statements, DMAP modules, punching and plotting operations.
filename2	The physical file name assigned to the FORTRAN file.
STATUS	Specifies whether the FORTRAN file is being created (STATUS=NEW) or has been created prior to the run (STATUS=OLD). If its status is not known, then STATUS=UNKNOWN is specified.

Describers	Meaning
UNIT=u	u is the FORTRAN unit number of the FORTRAN file.
FORM	Indicates whether the FORTRAN file is written in ASCII (FORM=FORMATTED) or binary (FORM=UNFORMATTED) format.
sys-spec	System specific or machine-dependent controls for IBM/MVS-type computers only. See the <a href="#">MSC.Nastran 2004 Installation and Operations Guide</a> for further information on machine-dependent aspects of the ASSIGN statement.
RECL = l	The size of a block of input/output information specified in words. The current RECL keyword is used by the DBC module and has a default minimum of 1024 words. The maximum allowed is 65536 words and is used to increase the database capacity.
SIZE = s	The number of blocks allocated to the DBC database. The SIZE keyword is used by the DBC module and has a default of 16777215. The maximum allowed is 2147483647 and is used to increase the database capacity. MSC.Patran releases before 2001 should use the defaults for RECL and SIZE or database verification failures will occur.

## Example

1. Assign DBC with a size of 1,000,000,000 blocks and blocksize of 65536.  
ASSIGN DBC='test.xdb', UNIT=40, RECL=65536, SIZE=1000000000



## 13.6 Option to Print the CD Value in the F06 File

### Introduction

MSC.Nastran has a unique capability that allows the user to define the grid in one coordinate system and request the output, such as DISP, in yet another coordinate system. Although this capability is convenient for the person who developed the FEA model, it can be very confusing for others to properly interpret the output. To remove the confusion and to provide an aid for output interpretation, the user defined output coordinate system ID, on field 7 (CD) of GRID entry, can be printed along with the responses.

### Inputs

An example of a request to print CD is as follows

DISP(CID) = ALL

### Outputs

A sample for the new displacement output is shown as follows.

0										SUBCASE 100									
DISPLACEMENT VECTOR																			
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	OCS ID											
1	G	0.0	0.0	0.0	0.0	0.0	0.0	0											
2	G	0.0	0.0	0.0	0.0	0.0	0.0	0											
3	G	0.0	0.0	0.0	0.0	0.0	0.0	0											
4	G	0.0	0.0	7.152507E-04	4.307964E-04	-2.239286E-03	0.0	0											
5	G	0.0	0.0	8.153501E-04	-2.642959E-05	-2.554608E-03	0.0	0											
6	G	6.945020E-04	0.0	0.0	0.0	2.171725E-03	-4.589293E-04	1000											
7	G	0.0	0.0	2.655400E-03	4.538104E-04	-4.071037E-03	0.0	0											
8	G	0.0	0.0	2.762245E-03	-4.022141E-05	-3.784631E-03	0.0	0											
9	G	0.0	0.0	2.608432E-03	-5.534015E-04	-4.051088E-03	0.0	0											
10	G	0.0	0.0	5.265094E-03	2.327628E-04	-4.473832E-03	0.0	0											

The output coordinate system will be printed as the last column and labeled as 'OCS ID'. If the user does not specify output coordinate system, '0' for basic coordinate system is printed.

### Guidelines and Limitations

When utilizing this new feature, please observe the following:

1. Output coordinate system ID will be printed only in the standard .f06 output file. All other forms of output, such as punch, plot, or the data block itself, e.g. OUGV1, remain the same as before which is without the coordinate system ID attached.

2. A CID keyword in the output request(s) need(s) only appear once in the Case Control Section. It will affect ALL grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce, and MPCForce.
3. When 'CID' appears in the output request, it is functional for grid related output in SORT1, SORT2, REAL/IMAG, and/or MAG/PHASE format.

## 13.7 Enhancements in the Specification of Parameters

Three new enhancements have been added for the specification of parameters:

1. Default parameter values can now be defined in the `nast2004rc` (unix machines) or the `nast2004.rcf` (windows) file similar to the command line submittal keywords, FMS commands, and nastran statements.
2. Parameter values can be specified on the command line if corresponding keywords are defined for these parameters. This feature allows the user to change the value of the parameters during runtime. These keywords are defined in the `nast2004.params` file and they must be placed in the `.../msc2004/bin/` subdirectory. Comment records are defined by starting with a "\$", "#", or ";" in column one. The format of this parameter file is as follows:

```
keyword      :      parameter name      :      param value      comment
```

where:

keyword is a user-defined keyword representing the parameter and can be the same as the parameter name

parameter name is the name of the parameter (e.g, snorm)

param value is either a number (integer or real) or acceptable value list, e.g., {"yes", "no"}

comment is an optional comment field

### Example

For example, if the `nast2004.params` file contains the following two lines

```
mysnorm      : snorm      : number
followk      : followk    : {"yes", "no"}
```

then the following submittal command

```
nastran sample mysorm=0. followk=no
```

will set the snorm value to "0." and followk to "no". A sample `nast2004.params` file, containing a list of commonly used parameters, is placed in the `../msc2004/bin/` subdirectory. The user can modify this file to add/delete keywords as needed.

3. A general keyword list can also be defined in a new `nast2004.kwds` file. This file must also be placed in the `../msc2004/bin/` subdirectory. The format of this general keyword file is as follows:

```
keyword      :      value list
```

where:

keyword is any user-defined keyword, value lists are acceptable values for the corresponding keyword. These values can then be used as a branching mechanism in the rc file.

### Example

For example, if the nast2004.kwds file contains the following lines

```
$ Keyword                      Value
$
Runtype      : {"stage1","stage2","final"}
```

and a different default set of parameters is desired during different stages of analyses, then a possible rc file may be as follows:

```
;
;
#   Section 1
$
[ runtype = stage1]
checkout=yes
$  plus other desired parameters
$
$   Section 2
$
[ runtype = stage2]
spcgen=1
$  plus other desired parameters
$
$   Section 3
$
[ runtype = final]
autospc=no
grdpnt=0
$  plus other desired parameters
$
```

then the user can use the runtype keyword to set the parameters. For example, the following submittal command:

```
nastran sample runtype=stage2
```

is equivalent to setting “param,spcgen,1” plus other parameters listed in section 2 for the run.

The following submittal command:

```
nastran sample runtype=final
```

is equivalent to setting “param,autospc,no”, “param,grdpnt,0”, plus other parameters listed in section 3 for the run.

The general keyword concept can be expanded for running jobs over a remote system. See the *MSC.Nastran 2004 Installation and Operations Guide* for further details.

If a parameter is specified on the submittal line, then a message similar to the following will appear at the end of the .f06 file. Note that parameter values set in the input file will take precedence.

The following PARAM values were set in the Control File:

```
PARAM,AUTOSPC,NO
```

Note that these values may have been changed by PARAM statements in Bulk Data or Case Control.

In addition, for general keywords, if the value descriptor starts or ends with the string "env" specified in any case and separated from the rest of the value descriptor with a comma (unless the value descriptor is only "env"), the keyword value will be set using the value associated with the environment variable having the same name as the keyword. The environment value will be subjected to the same syntax-checking rules as an INI file, RC file, or command line specification, with a warning message generated if syntax checking fails. This occurs even if the keyword is specified on the command line. Note that for UNIX systems, because environment variable names are case-sensitive, the keyword name must be specified exactly the same way as the environment variable name. This is the only time that the keyword name is case-sensitive. For Windows systems, because environment variable names are not case-sensitive, this restriction does not apply. Keyword values set from environment variables over-ride keyword values set in II or RC files, but do not over-ride keyword values set on the command line.

## 13.8 Enhancement for the DMAP String-based Editor

### Introduction

The MALTER and ALTER Executive Control statements were implemented in MSC.Nastran 68.2 to allow modifications of DMAP statements in the solution sequences using a string-based searching technique. MSC.Nastran 2004 extends this feature by supporting the commonly available line-based, as well as full screen, editors such as vi, sed, awk, and perl in the UNIX or linux environment by extending the string-base DMAP alter capability. This feature allows users to edit the alter, including interactively, during the development of the DMAP alters. This capability is derived by merging the ISHELL and INCLUDE features.

The ISHELL is an MSC.Nastran DMAP module that allows execution of an external program by spawning off a process. The INCLUDE command inserts the referenced file at the location where the INCLUDE command is placed. The “!” command combines the ISHELL/INCLUDE command by executing the external program and including the derived output into the MSC.Nastran input stream.

### Benefits

The ISHELL-INCLUDE statements allow the user to add, delete, or modify any DMAP statement interactively and save it to continue the run.

### Limitations

This feature is only supported on the unix and linux platforms.

### Examples

Two examples illustrate the implementation of this feature.

#### Example 1

```
Sol 101
Compile ifps1 $
alter 1,999 $
! xterm -e vi ishell.stdin
$
cend
.
.
begin bulk
.
enddata
```

The above example holds the MSC.Nastran job and opens up an xterm window with the Submap IFPS1 using the vi editor. All changes made to Subdmap IFPS1 and saved afterward just as one would normally edit the Subdmap in a batch mode. Once the changes are saved and the user exits the window, the MSC.Nastran job continues using the modified Subdmap.

## Example 2

```
Compile sestatics $
alter 1,999 $
! sed -e 's/K6ROT=0./K6ROT=1./' ishell.stdin
$
cend
.
.
begin bulk
.
enddata
```

The above example holds the MSC.Nastran job and replaces the K6ROT value from 0. to 1. in the main Subdmap SESTATICS using the sed editor. Once the changes are made, the MSC.Nastran job continues using the modified Subdmap.

.

## 13.9 Consistent Parameter Checking

In MSC.Nastran 2004, if a parameter is undefined, then either a FATAL or WARNING message similar to the following will be issued.

```
*** USER FATAL MESSAGE 439 (CPARAM)
      THE PARAMETER NAMED xxxx      (TYPE=  1) APPEARS TO BE AN INPUT PARAMETER BUT IS NOT
PREVIOUSLY INITIALIZED IN A TYPE STATEMENT OR SPECIFIED IN THE SUBDMAP ARGUMENT LIST.
      USER INFORMATION: IF THIS IS A WARNING THEN A DEFAULT VALUE OF  yyyyyy  WILL BE ASSUMED.
      USER ACTION:      IF THE PARAMETER IS INTENDED TO BE AN OUTPUT PARAMETER, THEN THE "S,N,"
PREFIX SHOULD BE SPECIFIED IN FRONT OF THE PARAMETER. IF THE PARAMETER IS INTENDED TO BE AN
INPUT PARAMETER THEN IT SHOULD BE INITIALIZED ON A TYPE STATEMENT OR SPECIFIED IN THE SUBDMAP
ARGUMENT LIST.
```

If the undefined parameter is an input parameter, then it should either be initialized in a TYPE statement or specified in the Subdmap argument list. If the offending parameter is an output parameter, then the “S,N,” prefix should be specified. To disable this checking (not recommended), set system 372 to 1.



## 13.10 Punch Output Enhancement

### Introduction

An enhancement was performed for the OFP and XYTRAN punch capability to correct deficiencies in tables supported, formatting of the entry information, and correct identification of the context of the punch output block. In addition, the MSC.Nastran system cell usage was expanded for the "new" punch option to remove the line numbering in columns 73-80, thereby reducing the file size. A brief summary of the enhancements are:

1. Recognition of the EKE and EDE output requests and correction of the eigenvalue associated with the mode.
2. Identification of the element type processed for the ESE, EKE and EDE punch options.
3. For Frequency Response, add the option selected to the ESE, EKE and EDE output header section.
4. Correction of mislabelling of the MPCF output as SPCF.
5. Addition of the identifier "LOAD FACTOR" to the nonlinear solution sort 1 output.
6. Addition of the superelement identifier when not the residual structure.
7. Elimination of the SUBCASE entry for SORT 2 STATICS, because SUBCASE is part of the entry and not global for the block information.
8. For an element that can select different invariant calculations from the Case Control, the selected invariant is indicated.
9. For QUAD and TRIA elements indicate the MATERIAL system for the local tensor is indicated rather than the assumed element system.
10. Correction of the Table Descriptors for thermal analysis rather than the structural descriptors.
11. For thermal punch output, other than for temperature, shorten the entry output to a single value to represent scalar.
12. Support of additional tables such as Grid Point Stress and discontinuities that have complex entry item formats for the "new" option.
13. Addition of element type names as well as their enumerated values.

## Inputs

### Description of the MSC.Nastran System Cell

System cell 210 is used to control the punch formatting, OFP tables being processed, and determine if the line number is placed in columns 73 through 80. It should be noted that this only effects the OFP punch capability and does not impact other punch options such as ECHO = PUNCH, Composite Element, synthesized element, and material property values, and other areas addressed outside of the OFP environment. The system cell values are:

Value	Selection
0	"OLD" Punch, default in V2001 and earlier
1	"NEW" Punch, default in V2004 and uses NDDL
2	Same as 1 except the line number is eliminated

The term "OLD" punch refers to the algorithm used to process the entries of the output table. In MSC.Nastran releases prior to the current release, assumed was that the first item of an entry could be either integer or real depending upon the table sort condition. The remaining entry items were assumed to be real. This led to incorrect translation of the binary when the entry item was either integer or character in data value. This could cause "not a number" or numerical range value exceptions in the translated punch file.

The "New" punch has been implemented in the time frame of the new executive system and uses the Nastran Data Definition Language (NDDL) interface to correctly translate the entry data items. The implication is that the reading program must better understand the true format of the entry within an output table class.

### Review of OFP Table Codes and Punch Header Conventions

**Table 13-1 Structural Punch Table Support and Descriptor Entry**

Table Code	Table Support	Table Content and Descriptor
1	Old	\$DISPLACEMENTS
2	Old	\$SOLOADS
3	Old	\$SPCF
4	Old	\$ELEMENT FORCES
5	Old	\$ELEMENT STRESSES

**Table 13-1 Structural Punch Table Support and Descriptor Entry**

Table Code	Table Support	Table Content and Descriptor
6	none	\$EIGENVALUE SUMMARY
7	Old	\$EIGENVECTOR
8	none	\$GRID POINT SINGULARITY TABLE
9	none	\$EIGENVALUE ANALYSIS SUMMARY
10	Old	\$VELOCITY
11	Old	\$ACCELERATION
12	Old	\$NON-LINEAR-FORCES
13	none	\$GRID POINT WEIGHT OUTPUT
14	Old	\$EIGENVECTOR (SOLUTION SET)
15	Old	\$DISPLACEMENTS (SOLUTION SET)
16	Old	\$VELOCITY (SOLUTION SET)
17	Old	\$ACCELERATION (SOLUTION SET)
18	Old	\$ELEMENT STRAIN ENERGIES
19	Old	\$GRID POINT FORCE BALANCE
20	none	\$STRESS AT GRID POINTS
21	none	\$STRAIN/CURVATURE AT GRID POINTS
22	none	\$ELEMENT INTERNAL FORCES AND MOMENTS
23	none	\$ELEMENT ORIENTED FORCES
24	none	\$ELEMENT PRESSURES
25	none	\$COMPOSITE FAILURE INDICIES
26	New	\$GRID POINT STRESS/PLANE STRESS
27	New	\$GRID POINT STRESS VOLUME DIRECT
28	New	\$GRID POINT STRESS VOLUME PRINCIPAL
29	New	\$ELEMENT STRESS DISCONTINUITIES
30	New	\$ELEMENT STRESS DISCONTINUITIES DIRECT

**Table 13-1 Structural Punch Table Support and Descriptor Entry**

Table Code	Table Support	Table Content and Descriptor
31	New	\$ELEMENT STRESS DISCONTINUITIES PRINCIPAL
32	New	\$GRID POINT STRESS DISCONTINUITIES
33	New	\$GRID POINT SRESS DISCONTINUITIES DIRECT
34	New	\$GRID POINT STRESS DISCON PRINCIPAL
35	New	\$GRID POINT STRESS/PLAIN STRAIN
36	Old	\$ELEMENT KINETIC ENERGY
37	Old	\$ELEMENT ENERGY LOSS PER CYCLE
38	New	\$MAX/MIN SUMMARY INFORMATION
39	Old	\$MPCF
40	Old	\$MODAL GRID POINT KINETIC ENERGY

**Notes:**

1. The "new" table format supports all of the "old" table formats for punch.
2. The word, "stress" in referenced table codes can be replaced by "strain".
3. The "none" table format implies that there is no Case Control interface to select punch as an option, only print is supported.
4. When RANDOM is selected, then the Table Descriptor of Table codes 1 through 5 can be augmented by the strings of - PSDF, - AUTO, - RMS, - NO, or - CRMS, to denote the entry formulation.

Other impacted aspects of RANDOM analysis are the replacement of the \$SUBCASE ID entry with \$RANDOM ID entry and two additional Table Descriptions that have no base table augmentation. The table descriptors are:

```
$CROSS-PSDF
$CROSS-CORRELATION FUNCTION
```

Starting with the current version, when thermal analysis has been selected, the structural table descriptor strings have been replaced by their correct thermal designation. **Table 13-2** associates the table code to the thermal table descriptor string.

In addition, with the exception of temperature, which formats the punch output to bulk data conventions, the scalar class entries have been shortened to a single data item.

**Table 13-2 Alternate Punch Descriptor for Thermal Analysis**

Table Code	Thermal Descriptor Entry
1	\$TEMPERATURE
2	\$HEAT FLOW AT LOAD POINTS
3	\$HEAT FLOW AT CONTSTRAINT POINTS
4	\$ELEMENT GRADIENTS AND FLUXES
10	\$ENTHALPY
11	\$H DOT

### Description of an OFP Punch Header Block

The OFP Punch header is composed of six to nine entries, depending upon sort conditions, analysis approach, grid versus element responses, and other conditions. In addition, column one has the following meaning. A "\$" indicates the entry belongs to the header section. A "-" indicates the continuation of item information for an entry. A blank in column one implies the start of an entry. In general, the punch header entries are:

```
$TITLE = Case Control Title
$SUBTITLE = Case Control Subtitle
$LABEL = Case Control Lable
$Table Descriptor
$MAXMIN (Table code = 38) additional information
$Output Format
$SUBCASE ID or RANDOM ID, missing for SORT 2 STATICS
$ELEMENT TYPE = missing for grid table codes
```

### Approach or Sort Specific Information

Note that entries one through four are always be present. The MAXMIN additional information entry is described in “[MAX/MIN/RMS Methodology in Data Recovery](#)” on page 134 because this is a new feature for the MSC.Nastran 2004 release.

The Table Descriptor entry has been augmented for the energy recovery tables 18, 36, and 37 to delineate the element type for non frequency response analysis approaches, and to include the calculation option for the Frequency Response formulation.

\$ELEMENT STRAIN ENERGIES

BAR

OPT=AVERAGE  
=AMPLITUDE  
=PEAK

The Output Format entry has been augmented, when necessary, to include the superelement identification. The area remains blank when a residual structure is detected.

\$REAL OUTPUT

SEID=10

The SUBCASE entry has been augmented to include the SURFACE or VOLUME designations when Grid Point Stress or Discontinuity tables are being processed. The keyword FIBER or STRCUR will be present for strain table recovery.

\$SUBCASE ID =

1 SURFACE=

1 FIBER

VOLUME=

The Element Type line has the most significant augmentation. The element name has now been appended to further identify the element enumeration. Because certain elements may contain variable user selectable responses, additional keywords are appended, when necessary, to reflect the entry content. The keywords VONM and SHEAR deal with the invariant selected; STRCUR or FIBER for the strain recovery, CUBIC for the QUAD4 corner extrapolation method, and MATERIAL for shell elements, when not in the element reference system.

\$ELEMENT TYPE = 144 QUAD4C VONM SRCUR BILIN MATERIAL  
SHEAR FIBER SGAGE  
CUBIC

The last augmentation is for the sort 2 table condition and deals with the issues of approach and whether the entry identification is integer or real.

\$SPOINT ID =

1

IDENTIFIED BY SUBACASE

FREQUENCY

TIME

LOAD FACTOR

## Example

The following are examples of header and entry changes between the current and previous releases of the OFP punch formats.

Sort2 Statics, the SUBCASE line was deleted since the SUBCASE identifies the entry and SUBCASE is not global for all entries.

\$TITLE	=CANTILEVER BEAM CONSTRUCTED OF PLATES, STRESS CONTOURS	F10	1
\$SUBTITLE	= REF: SINGER,STRENGTH OF MATERIALS, ART 5-2,PG 133		2
\$LABEL	=		3
\$DISPLACEMENTS			4
\$REAL OUTPUT			5
\$SUBCASE ID =	1		6
\$POINT ID =	1		6
	1	G	0.000000E+00
			0.000000E+00
			0.000000E+00
-CONT-			0.000000E+00
			8
\$TITLE	=CANTILEVER BEAM CONSTRUCTED OF PLATES, STRESS CONTOURS	F10	1
\$SUBTITLE	= REF: SINGER,STRENGTH OF MATERIALS, ART 5-2,PG 133		2
\$LABEL	=		3
\$DISPLACEMENTS			4
\$REAL OUTPUT			5
\$POINT ID =	1	IDENTIFIED BY SUBCASE	5
	1	G	0.000000E+00
			0.000000E+00
			0.000000E+00
-CONT-			0.000000E+00
			7

In superelement processing the block content was not identified.

\$TITLE	=RESULT CASE NAME		1
\$SUBTITLE	=LOAD CASE NAME 1		2
\$LABEL	=RESTRAINT CASE NAME 1		3
\$DISPLACEMENTS			4
\$REAL OUTPUT			5
\$SUBCASE ID =	1		6
	101	G	0.000000E+00
			9.994075E-04
			0.000000E+00
-CONT-			0.000000E+00
			8
\$TITLE	=RESULT CASE NAME		1
\$SUBTITLE	=LOAD CASE NAME 1		2
\$LABEL	=RESTRAINT CASE NAME 1		3
\$DISPLACEMENTS			4
\$REAL OUTPUT	SEID=10		5
\$SUBCASE ID =	1		6
	101	G	0.000000E+00
			9.994075E-04
			0.000000E+00
-CONT-			0.000000E+00
			8

When Multi-point Constraint Forces were recovered, they were incorrectly identified.

\$SUBTITLE=LOAD CASE NAME 1	2
\$LABEL =RESTRAINT CASE NAME 1	3
\$SPCF	4
\$REAL OUTPUT	5
\$SUBCASE ID = 1	6
1 G 0.000000E+00 0.000000E+00 0.000000E+00	7
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	8
\$TITLE =RESULT CASE NAME	1
\$SUBTITLE=LOAD CASE NAME 1	2
\$LABEL =RESTRAINT CASE NAME 1	3
\$MPCF	4
\$REAL OUTPUT	5
\$SUBCASE ID = 1	6
1 G 0.000000E+00 0.000000E+00 0.000000E+00	7
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	8

To assist the user with the element type enumeration, the GPTABD name values were appended to the white area of the ELEMENT TYPE entry.

\$TITLE =RESULT CASE NAME	1
\$SUBTITLE=LOAD CASE NAME 1	2
\$LABEL =RESTRAINT CASE NAME 1	3
\$ELEMENT FORCES	4
\$REAL OUTPUT	5
\$SUBCASE ID = 1	6
\$ELEMENT TYPE = 33	7
1 -3.906332E+01 -9.898468E+02 -3.342360E+01	8
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	9
-CONT- 0.000000E+00 0.000000E+00	10
\$TITLE =RESULT CASE NAME	745
\$SUBTITLE=LOAD CASE NAME 1	746
\$LABEL =RESTRAINT CASE NAME 1	747
\$ELEMENT FORCES	748
\$REAL OUTPUT	749
\$SUBCASE ID = 1	750
\$ELEMENT TYPE = 33 QUAD4	751
1 -3.906332E+01 -9.898468E+02 -3.342360E+01	752
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	753
-CONT- 0.000000E+00 0.000000E+00	754



When element classes had optional output selections the user could select from the Case Control entry, these selections were not indicated in the punch heading block.

\$TITLE	=RESULT CASE NAME			287
\$SUBTITLE	=LOAD CASE NAME 1			288
\$LABEL	=RESTRAINT CASE NAME 1			289
\$ELEMENT	STRAINS			290
\$REAL	OUTPUT			291
\$SUBCASE	ID =	1		292
\$ELEMENT	TYPE =	33		293
	1	-5.000000E-02	8.596358E-05	-3.260426E-04
-CONT-		-2.896712E-05	-2.010852E+00	8.647210E-05
-CONT-		-3.265511E-04	2.515292E-04	5.000000E-02
-CONT-		8.596358E-05	-3.260426E-04	-2.896712E-05
-CONT-		-2.010852E+00	8.647210E-05	-3.265511E-04
-CONT-		2.515292E-04		
				299
\$TITLE	=RESULT CASE NAME			287
\$SUBTITLE	=LOAD CASE NAME 1			288
\$LABEL	=RESTRAINT CASE NAME 1			289
\$ELEMENT	STRAINS			290
\$REAL	OUTPUT			291
\$SUBCASE	ID =	1		292
\$ELEMENT	TYPE =	33	QUAD4 VONM FIBER	293
	1	-5.000000E-02	8.596358E-05	-3.260426E-04
-CONT-		-2.896712E-05	-2.010852E+00	8.647210E-05
-CONT-		-3.265511E-04	2.515292E-04	5.000000E-02
-CONT-		8.596358E-05	-3.260426E-04	-2.896712E-05
-CONT-		-2.010852E+00	8.647210E-05	-3.265511E-04
-CONT-		2.515292E-04		
				299
\$TITLE	=RESULT CASE NAME			287
\$SUBTITLE	=LOAD CASE NAME 1			288
\$LABEL	=RESTRAINT CASE NAME 1			289
\$ELEMENT	STRAINS			290
\$REAL	OUTPUT			291
\$SUBCASE	ID =	1		292
\$ELEMENT	TYPE =	33	QUAD4 VONM STRCUR	293
	1	0.000000E+00	8.596358E-05	-3.260426E-04
-CONT-		-2.896712E-05	-2.010852E+00	8.647210E-05
-CONT-		-3.265511E-04	2.515292E-04	-1.000000E+00
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00
-CONT-		0.000000E+00		
				299

When an element had other than real data items, the values were incorrectly formatted. In this example the integer are outside the IEEE exponent range.

```

$TITLE      =RESULT CASE NAME      1
$SUBTITLE=LOAD CASE NAME 1        2
$LABEL      =RESTRAINT CASE NAME 1 3
$ELEMENT STRAINS 4
$REAL OUTPUT 5
$SUBCASE ID = 1 6
$ELEMENT TYPE = 144 7
1 1.973054E+02 5.605194E-45 0.000000E+00 8
-CONT- 8.596358E-05 -3.260426E-04 -2.896712E-05 9
-CONT- -2.010852E+00 8.647210E-05 -3.265511E-04 10
-CONT- 2.515292E-04 -1.000000E+00 0.000000E+00 11
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 12
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 13
-CONT- 1.401298E-45 0.000000E+00 8.924844E-05 14
-CONT- -4.056804E-04 -1.781213E-04 -9.896718E+00 15
-CONT- 1.047868E-04 -4.212187E-04 3.214852E-04 16
-CONT- -1.000000E+00 0.000000E+00 0.000000E+00 17
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 18
-CONT- 0.000000E+00 0.000000E+00 2.802597E-45 19
-CONT- 0.000000E+00 8.894981E-05 -2.488935E-04 20
-CONT- -1.678175E-04 -1.320753E+01 1.086421E-04 21
-CONT- -2.685857E-04 2.242232E-04 -1.000000E+00 22
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 23
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 24
-CONT- 0.000000E+00 1.261169E-44 0.000000E+00 25
-CONT- 8.297735E-05 -2.488935E-04 1.065242E-04 26
-CONT- 8.897819E+00 9.131589E-05 -2.572320E-04 27
-CONT- 2.086957E-04 -1.000000E+00 0.000000E+00 28
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 29
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 30
-CONT- 1.121039E-44 0.000000E+00 8.267872E-05 31
-CONT- -4.056804E-04 1.236546E-04 7.104448E+00 32
-CONT- 9.038459E-05 -4.133863E-04 3.101407E-04 33
-CONT- -1.000000E+00 0.000000E+00 0.000000E+00 34
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 35
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 36

```

```

$TITLE      =RESULT CASE NAME      1
$SUBTITLE=LOAD CASE NAME 1        2
$LABEL      =RESTRAINT CASE NAME 1 3
$ELEMENT STRAINS 4
$REAL OUTPUT 5
$SUBCASE ID = 1 6
$ELEMENT TYPE = 144 7
1 CEN/ QUAD4C VONM STRCUR BILIN 4 0.000000E+00 8
-CONT- 8.596358E-05 -3.260426E-04 -2.896712E-05 9
-CONT- -2.010852E+00 8.647210E-05 -3.265511E-04 10
-CONT- 2.515292E-04 -1.000000E+00 0.000000E+00 11
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 12
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 13
-CONT- 1 0.000000E+00 1.131398E-04 14
-CONT- -4.066659E-04 -2.896712E-05 -1.594806E+00 15
-CONT- 1.135430E-04 -4.070691E-04 3.160993E-04 16
-CONT- -1.000000E+00 0.000000E+00 0.000000E+00 17
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 18
-CONT- 0.000000E+00 0.000000E+00 2 19
-CONT- 0.000000E+00 6.580506E-05 -2.497893E-04 20
-CONT- -2.896712E-05 -2.622126E+00 6.646837E-05 21
-CONT- -2.504526E-04 1.929787E-04 -1.000000E+00 22
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 23
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 24
-CONT- 0.000000E+00 9 0.000000E+00 25
-CONT- 5.983260E-05 -2.479976E-04 -2.896712E-05 26
-CONT- -2.687879E+00 6.051255E-05 -2.486775E-04 27
-CONT- 1.892093E-04 -1.000000E+00 0.000000E+00 28
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 29
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 30
-CONT- 8 0.000000E+00 1.065701E-04 31
-CONT- -4.046950E-04 -2.896712E-05 -1.621391E+00 32
-CONT- 1.069800E-04 -4.051049E-04 3.119066E-04 33
-CONT- -1.000000E+00 0.000000E+00 0.000000E+00 34
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 35
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00 36

```

Again the coordinate system was also not correctly indicated.

\$TITLE	=	MSC/NASTRAN JOB CREATED ON 11-NOV-99 AT 14:47:27			24
\$SUBTITLE	=	SUBCASE1			25
\$LABEL	=				26
\$ELEMENT		STRESSES			27
\$REAL		OUTPUT			28
\$SUBCASE	ID =	1			29
\$ELEMENT	TYPE =	33			30
	1	-5.000000E-02	3.225000E+04	3.225000E+04	31
-CONT-		-2.775000E+04	-4.500000E+01	6.000000E+04	32
-CONT-		4.500000E+03	5.788134E+04	5.000000E-02	33
-CONT-		-3.225000E+04	-3.225000E+04	2.775000E+04	34
-CONT-		4.500000E+01	-4.500000E+03	-6.000000E+04	35
-CONT-		5.788134E+04			36
\$TITLE	=	MSC/NASTRAN JOB CREATED ON 11-NOV-99 AT 14:47:27			24
\$SUBTITLE	=	SUBCASE1			25
\$LABEL	=				26
\$ELEMENT		STRESSES			27
\$REAL		OUTPUT			28
\$SUBCASE	ID =	1			29
\$ELEMENT	TYPE =	33	QUAD4	VONM	MATERIAL
	1	-5.000000E-02	3.225000E+04	3.225000E+04	31
-CONT-		-2.775000E+04	-4.500000E+01	6.000000E+04	32
-CONT-		4.500000E+03	5.788134E+04	5.000000E-02	33
-CONT-		-3.225000E+04	-3.225000E+04	2.775000E+04	34
-CONT-		4.500000E+01	-4.500000E+03	-6.000000E+04	35
-CONT-		5.788134E+04			36

This checks that SHEAR versus VONM is correctly identified.

\$TITLE	=RESULT CASE NAME				7509
\$SUBTITLE	=LOAD CASE NAME 1				7510
\$LABEL	=RESTRAINT CASE NAME 1				7511
\$ELEMENT	STRESSES				7512
\$REAL	OUTPUT				7513
\$SUBCASE	ID =	1			7514
\$ELEMENT	TYPE =	33			7515
	1	-5.000000E-02	-3.906332E+02	-9.898468E+03	7516
-CONT-		-3.342360E+02	-2.010852E+00	-3.788981E+02	7517
-CONT-		-9.910203E+03	4.765653E+03	5.000000E-02	7518
-CONT-		-3.906332E+02	-9.898468E+03	-3.342360E+02	7519
-CONT-		-2.010852E+00	-3.788981E+02	-9.910203E+03	7520
-CONT-		4.765653E+03			7521
\$TITLE	=RESULT CASE NAME				8433
\$SUBTITLE	=LOAD CASE NAME 1				8434
\$LABEL	=RESTRAINT CASE NAME 1				8435
\$ELEMENT	STRESSES				8436
\$REAL	OUTPUT				8437
\$SUBCASE	ID =	1			8438
\$ELEMENT	TYPE =	33	QUAD4	SHEAR	8439
	1	-5.000000E-02	-3.906332E+02	-9.898468E+03	8440
-CONT-		-3.342360E+02	-2.010852E+00	-3.788981E+02	8441
-CONT-		-9.910203E+03	4.765653E+03	5.000000E-02	8442
-CONT-		-3.906332E+02	-9.898468E+03	-3.342360E+02	8443
-CONT-		-2.010852E+00	-3.788981E+02	-9.910203E+03	8444
-CONT-		4.765653E+03			8445

Check if element class name exists for Laminated Composite and that the invariant is eliminated since SHEAR context is only available.

```
$TITLE      =SQUARE PLATE WITH CIRCULAR HOLE - COMPOSITES SOLUTION      1
$SUBTITLE=MEMBRANE PROPERTIES OF QUAD4 AND TRIA3 ELEMENTS                2
$LABEL      =UNIFORM LOAD ALONG X=5.0.                                  3
$ELEMENT STRESSES                                                         4
$REAL OUTPUT                                                                5
$SUBCASE ID =          1                                                  6
$ELEMENT TYPE =          95                                              7
      1          1          6.713920E+03      -2.537445E+03      8
-CONT-          1.848406E+02      0.000000E+00      0.000000E+00      9
-CONT-          1.144151E+00      6.717611E+03      -2.541137E+03     10
-CONT-          4.629374E+03                                             11

$TITLE      =SQUARE PLATE WITH CIRCULAR HOLE - COMPOSITES SOLUTION      1
$SUBTITLE=MEMBRANE PROPERTIES OF QUAD4 AND TRIA3 ELEMENTS                2
$LABEL      =UNIFORM LOAD ALONG X=5.0.                                  3
$ELEMENT STRESSES                                                         4
$REAL OUTPUT                                                                5
$SUBCASE ID =          1                                                  6
$ELEMENT TYPE =          95      QUAD4LC                                7
      1          1          6.713920E+03      -2.537445E+03      8
-CONT-          1.848406E+02      0.000000E+00      0.000000E+00      9
-CONT-          1.144151E+00      6.717611E+03      -2.541137E+03     10
-CONT-          4.629374E+03                                             11
```

The GPSDCON entry caused unidentified output to be created in the punch stream.

```

$TITLE      =RESULT CASE NAME                                831
$SUBTITLE=LOAD CASE NAME 1                                  832
$LABEL      =RESTRAINT CASE NAME 1                          833
$REAL OUTPUT                                834
$SUBCASE ID =          1                                     835
          1          1.358237E-19          3.892658E+02          2.825602E+03          836
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          837
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          838
          1          1.358237E-19          3.892658E+02          2.825602E+03          839
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          840
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          841
          1          1.738842E-19          3.892658E+02          2.825602E+03          842
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          843
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          844

$TITLE      =RESULT CASE NAME                                831
$SUBTITLE=LOAD CASE NAME 1                                  832
$LABEL      =RESTRAINT CASE NAME 1                          833
$GRID POINT STRESS DISCONTINUITIES                      834
$REAL OUTPUT                                835
$SUBCASE ID =          1          SURFACE=          1          836
          1          Z1          3.892658E+02          2.825602E+03          837
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          838
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          839
          1          Z2          3.892658E+02          2.825602E+03          840
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          841
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          842
          1          MID          3.892658E+02          2.825602E+03          843
-CONT-          5.154498E+02          3.298798E+02          2.884989E+03          844
-CONT-          1.277554E+03          2.729644E+03          1.673444E+03          845

```

The ELSDCON entry caused unidentified output to be created in the punch stream.

```

$TITLE      =RESULT CASE NAME                                1867
$SUBTITLE=LOAD CASE NAME 1                                1868
$LABEL      =RESTRAINT CASE NAME 1                        1869
$REAL OUTPUT                                1870
$SUBCASE ID =          1                                1871
          1          5.724519E+10          1.491285E-07          1.358237E-19 1872
-CONT-          3.240547E+02          2.101422E+03          6.456028E+02 1873
-CONT-          2.751211E+02          2.147283E+03          9.770737E+02 1874
-CONT-          2.051576E+03          1.282938E+03                                1875

$TITLE      =RESULT CASE NAME                                1869
$SUBTITLE=LOAD CASE NAME 1                                1870
$LABEL      =RESTRAINT CASE NAME 1                        1871
$ELEMENT STRESS DISCONTINUITIES                                1872
$REAL OUTPUT                                1873
$SUBCASE ID =          1          SURFACE=          1          1874
          1          QUAD4          Z1          1875
-CONT-          3.240547E+02          2.101422E+03          6.456028E+02 1876
-CONT-          2.751211E+02          2.147283E+03          9.770737E+02 1877
-CONT-          2.051576E+03          1.282938E+03                                1878

```

The GPSTRESS entry caused unidentified output to be created in the punch stream.

```

$TITLE      =RESULT CASE NAME                                287
$SUBTITLE=LOAD CASE NAME 1                                288
$LABEL      =RESTRAINT CASE NAME 1                        289
$REAL OUTPUT                                290
$SUBCASE ID =          1                                291
          1          0.000000E+00          1.358237E-19          -7.940824E+02 292
-CONT-          -1.270989E+04          1.011802E+03          4.819154E+00 293
-CONT-          -7.087778E+02          -1.279519E+04          6.043207E+03 294
-CONT-          1.245593E+04                                295

$TITLE      =RESULT CASE NAME                                2913
$SUBTITLE=LOAD CASE NAME 1                                2914
$LABEL      =RESTRAINT CASE NAME 1                        2915
$GRID POINT STRESS/PLANE STRESS                                2916
$REAL OUTPUT                                2917
$SUBCASE ID =          1          SURFACE=          1          2918
          1          0          Z1          -7.940824E+02 2919
-CONT-          -1.270989E+04          1.011802E+03          4.819154E+00 2920
-CONT-          -7.087778E+02          -1.279519E+04          6.043207E+03 2921
-CONT-          1.245593E+04                                2922
          1          0          Z2          -7.940824E+02 2923
-CONT-          -1.270989E+04          1.011802E+03          4.819154E+00 2924
-CONT-          -7.087778E+02          -1.279519E+04          6.043207E+03 2925
-CONT-          1.245593E+04                                2926
          1          0          MID          -7.940824E+02 2927
-CONT-          -1.270989E+04          1.011802E+03          4.819154E+00 2928
-CONT-          -7.087778E+02          -1.279519E+04          6.043207E+03 2929
-CONT-          1.245593E+04                                2930

```

The GPSTRAIN entry caused unidentified output to be created in the punch stream. The GPSTRESS phase II module also did not react to FIBER/STRCUR correctly.

```

$TITLE      =RESULT CASE NAME                               5665
$SUBTITLE=LOAD CASE NAME 1                                   5666
$LABEL      =RESTRAINT CASE NAME 1                           5667
$REAL OUTPUT                                         5668
$SUBCASE ID = 1                                           5669
      1      0.000000E+00      1.358237E-19      1.006295E-04      5670
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5671
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5672
-CONT-      4.801584E-04                                           5673
      1      0.000000E+00      1.358237E-19      1.006295E-04      5674
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5675
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5676
-CONT-      4.801584E-04                                           5677
      1      0.000000E+00      1.738842E-19      1.299615E+04      5678
-CONT-      -2.920491E+03      -1.808289E+03      -6.400696E+00      5679
-CONT-      1.319900E+04      -3.123345E+03      8.161175E+03      5680
-CONT-      1.500647E+04                                           5681

$TITLE      =RESULT CASE NAME                               5673
$SUBTITLE=LOAD CASE NAME 1                                   5674
$LABEL      =RESTRAINT CASE NAME 1                           5675
$GRID POINT STRAIN/PLANE STRESS                         5676
$REAL OUTPUT                                         5677
$SUBCASE ID = 1      SURFACE= 1 FIBER                     5678
      1      0      Z1      1.006295E-04      5679
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5680
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5681
-CONT-      4.801584E-04                                           5682
      1      0      Z2      1.006295E-04      5683
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5684
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5685
-CONT-      4.801584E-04                                           5686
      1      0      MID      1.006295E-04      5687
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5688
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5689
-CONT-      4.801584E-04                                           5690

$TITLE      =RESULT CASE NAME                               5673
$SUBTITLE=LOAD CASE NAME 1                                   5674
$LABEL      =RESTRAINT CASE NAME 1                           5675
$GRID POINT STRAIN/PLANE STRESS                         5676
$REAL OUTPUT                                         5677
$SUBCASE ID = 1      SURFACE= 1 STRCUR                     5678
      1      0      MID      1.006295E-04      5679
-CONT-      -4.157221E-04      8.768948E-05      4.819154E+00      5680
-CONT-      1.043260E-04      -4.194186E-04      5.237446E-04      5681
-CONT-      4.801584E-04                                           5682
      1      0      CURV      0.000000E+00      5683
-CONT-      0.000000E+00      0.000000E+00      0.000000E+00      5684
-CONT-      0.000000E+00      0.000000E+00      0.000000E+00      5685
-CONT-      0.000000E+00                                           5686

```



The GPSTRESS entry caused unidentified output to be created in the punch stream.

```

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM                198
$SUBTITLE= ELASTIC LOADING BY FORCE                             199
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS           200
$REAL OUTPUT                                     201
$SUBCASE ID =                2                                202
          3401                9.471862E+02          5.677020E+01    203
-CONT-                1.723746E+01          -NaNQ                204

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM                199
$SUBTITLE= ELASTIC LOADING BY FORCE                             200
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS           201
$GRID POINT STRESS/PLAIN STRAIN                        202
$REAL OUTPUT                                     203
$SUBCASE ID =                2 SURFACE= 34                    204
          3401                9.471862E+02          5.677020E+01    205
-CONT-                1.723746E+01                                206

```

The GPSTRESS entry caused unidentified output to be created in the punch stream.

```

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM                235
$SUBTITLE= ELASTIC LOADING BY FORCE                             236
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS           237
$REAL OUTPUT                                     238
$SUBCASE ID =                2                                239
          3901                1.979894E+03          6.972885E+01    -2.529273E-14    240
-CONT-                2.071350E-13          -3.198958E+00          2.251117E+01    241
-CONT-                -6.832076E+02          1.946365E+03            242

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM                237
$SUBTITLE= ELASTIC LOADING BY FORCE                             238
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS           239
$GRID POINT STRESS VOLUME DIRECT                        240
$REAL OUTPUT                                     241
$SUBCASE ID =                2 VOLUME= 39                    242
          3901                1.979894E+03          6.972885E+01    -2.529273E-14    243
-CONT-                2.071350E-13          -3.198958E+00          2.251117E+01    244
-CONT-                -6.832076E+02          1.946365E+03            245

```

The GPSTRESS entry caused unidentified output to be created in the punch stream.

```

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM      336
$SUBTITLE= ELASTIC LOADING BY FORCE                  337
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS 338
$REAL OUTPUT 339
$SUBCASE ID =          2                            340
    3901      1.980150E+03    -4.018156E-01    6.987475E+01 341
-CONT-      9.999354E-01    -1.135504E-02    -5.369717E-04 342
-CONT-     -1.903498E-05     4.556393E-02    -9.989614E-01 343
-CONT-      1.136771E-02     9.988969E-01     4.556077E-02 344
-CONT-     -6.832076E+02     1.946365E+03                        345

$TITLE      = LINEAR ELEMENT CODE CHECK PROBLEM      339
$SUBTITLE= ELASTIC LOADING BY FORCE                  340
$LABEL      = CHECK GRID POINT STRESS OUTPUT FORMATS 341
$GRID POINT STRESS VOLUME PRINCIPAL 342
$REAL OUTPUT 343
$SUBCASE ID =          2    VOLUME= 39              344
    3901      1.980150E+03    -4.018156E-01    6.987475E+01 345
-CONT-      9.999354E-01    -1.135504E-02    -5.369717E-04 346
-CONT-     -1.903498E-05     4.556393E-02    -9.989614E-01 347
-CONT-      1.136771E-02     9.988969E-01     4.556077E-02 348
-CONT-     -6.832076E+02     1.946365E+03                        349

```

The element class did not identify the Element Strain Energy output.

```

$TITLE      =RESULT CASE NAME                        3043
$SUBTITLE=LOAD CASE NAME 1                          3044
$LABEL      =RESTRAINT CASE NAME 1                  3045
$ELEMENT STRAIN ENERGIES                            3046
$REAL OUTPUT 3047
$SUBCASE ID =          1                            3048
    1          2.976484E-03    6.479465E-02    1.632466E+00 3049
    2          1.780166E-03    3.875217E-02    5.816491E-01 3050

$TITLE      =RESULT CASE NAME                        8977
$SUBTITLE=LOAD CASE NAME 1                          8978
$LABEL      =RESTRAINT CASE NAME 1                  8979
$ELEMENT STRAIN ENERGIES                            8980
$REAL OUTPUT 8981
$SUBCASE ID =          1                            8982
    1          2.976484E-03    6.479465E-02    1.632466E+00 8983
    2          1.780166E-03    3.875217E-02    5.816491E-01 8984

```

In eigenvalue analysis the eigenvalue was wrong and element class identifier missing.

```

$TITLE      =RESULT_CASE_NAME                                2871
$SUBTITLE=LOAD CASE NAME                                    2872
$LABEL      =RESTRAINT CASE NAME                            2873
$ELEMENT STRAIN ENERGIES                                    2874
$REAL OUTPUT                                           2875
$SUBCASE ID =                      1                        2876
$EIGENVALUE =  .3390093E+13  MODE =          1              2877
      1                      5.414903E+02      6.852106E-02      4.061989E+02      2878
      2                      3.771790E+04      4.772884E+00      2.829408E+04      2879

$TITLE      =RESULT_CASE_NAME                                4101
$SUBTITLE=LOAD CASE NAME                                    4102
$LABEL      =RESTRAINT CASE NAME                            4103
$ELEMENT STRAIN ENERGIES                                    4103
$REAL OUTPUT                                           4104
$SUBCASE ID =                      1                        4105
$EIGENVALUE =  0.1580508E+07  MODE =          1              4106
      1                      5.414903E+02      6.852106E-02      4.061989E+02      4107
      2                      3.771790E+04      4.772884E+00      2.829408E+04      4108

```

The EKE entry did not identify the output, the eigenvalue was incorrect and the element class type was missing.

```

$TITLE      =RESULT_CASE_NAME                                3389
$SUBTITLE=LOAD CASE NAME                                    3390
$LABEL      =RESTRAINT CASE NAME                            3391
$REAL OUTPUT                                           3392
$SUBCASE ID =                      1                        3393
$EIGENVALUE =  .3390093E+13  MODE =          1              3394
      1                      6.581564E+04      8.328419E+00      4.937160E+04      3395
      2                      3.031419E+02      3.836007E-02      2.274019E+02      3396

$TITLE      =RESULT_CASE_NAME                                4821
$SUBTITLE=LOAD CASE NAME                                    4822
$LABEL      =RESTRAINT CASE NAME                            4823
$ELEMENT KINETIC ENERGY                                4823
$REAL OUTPUT                                           4824
$SUBCASE ID =                      1                        4825
$EIGENVALUE =  0.1580508E+07  MODE =          1              4826
      1                      6.581565E+04      8.328419E+00      4.937161E+04      4827
      2                      3.031419E+02      3.836007E-02      2.274019E+02      4828

```

ESE in Frequency Response did not identify the method used for calculating the response quantity. The element class was also missing.

\$TITLE	=				1
\$SUBTITLE	=				2
\$LABEL	=				3
\$ELEMENT STRAIN ENERGIES					4
\$REAL OUTPUT					5
\$SUBCASE ID	=	2			6
\$FREQUENCY	=	.1000000E+01			7
101		2.102882E-05	1.820742E-06	1.051441E-07	8
102		2.893353E-02	2.505157E-03	1.446677E-04	9
103		2.029759E-01	1.757429E-02	1.014879E-03	10

\$TITLE	=				1
\$SUBTITLE	=				2
\$LABEL	=				3
\$ELEMENT STRAIN ENERGIES			BAR	OPT=AVERAGE	4
\$REAL OUTPUT					5
\$SUBCASE ID	=	2			6
\$FREQUENCY	=	1.0000000E+00			7
101		3.086982E-05	2.673122E-06	1.543491E-07	8
102		2.927053E-02	2.534634E-03	1.463527E-04	9
103		2.036543E-01	1.763512E-02	1.018272E-03	10

In EKE output in Frequency Response the block was not identified, the method was not indicated, and the element class was missing.

\$TITLE	=				5666
\$SUBTITLE	=				5667
\$LABEL	=				5668
\$REAL OUTPUT					5669
\$SUBCASE ID	=	2			5670
\$FREQUENCY	=	.1000000E+01			5671
101		4.627901E+01	5.389437E+00	2.313951E-01	5672
102		3.740206E+01	4.355669E+00	1.870103E-01	5673
103		2.966045E+01	3.454118E+00	1.483023E-01	5674

\$TITLE	=				5668
\$SUBTITLE	=				5669
\$LABEL	=				5670
\$ELEMENT KINETIC ENERGY			BAR	OPT=AVERAGE	5671
\$REAL OUTPUT					5672
\$SUBCASE ID	=	2			5673
\$FREQUENCY	=	1.0000000E+00			5674
101		4.627080E+01	5.390246E+00	2.313540E-01	5675
102		3.739541E+01	4.356322E+00	1.869771E-01	5676
103		2.965508E+01	3.454624E+00	1.482754E-01	5677

In EDE output in Frequency Response the block was not identified, the method was not indicated, and the element class was missing.

\$TITLE	=			9635
\$SUBTITLE	=			9636
\$LABEL	=			9637
\$REAL OUTPUT				9638
\$SUBCASE ID	=	2		9639
\$FREQUENCY	=	.1000000E+01		9640
		3042	6.970902E+02	2.664647E+01
		5042	1.918979E+03	7.335353E+01
				9642
\$TITLE	=			9799
\$SUBTITLE	=			9800
\$LABEL	=			9801
\$ELEMENT ENERGY LOSS PER CYCLE DAMP2 OPT=AVERAGE				9802
\$REAL OUTPUT				9803
\$SUBCASE ID	=	2		9804
\$FREQUENCY	=	1.0000000E+00		9805
		3042	6.970200E+02	2.665085E+01
		5042	1.918356E+03	7.334915E+01
				9807

Nonlinear output blocks were not identified by the LOAD FACTOR.

\$TITLE	=RESULT_CASE_1				1
\$SUBTITLE=					2
\$LABEL	=				3
\$ELEMENT STRESSES					4
\$REAL OUTPUT					5
\$SUBCASE ID =	1				6
\$ELEMENT TYPE =	89				7
1	1.905082E+02	1.905082E+02	6.350274E-06		8
-CONT-	6.350274E-06	0.000000E+00	0.000000E+00		9

\$TITLE	=RESULT_CASE_1				1
\$SUBTITLE=					2
\$LABEL	=				3
\$ELEMENT STRESSES					4
\$REAL OUTPUT					5
\$SUBCASE ID =	1				6
\$ELEMENT TYPE =	89	RODNL			7
\$LOAD FACTOR =	1.000000E-01				7
1	1.905082E+02	1.905082E+02	6.350274E-06		8
-CONT-	6.350274E-06	0.000000E+00	0.000000E+00		9

Punch

format maintained, however block identifier changed to reflect content.

\$TITLE	= RADIATION BOUNDARY CONDITION				1
\$SUBTITLE=					2
\$LABEL	=				3
\$DISPLACEMENTS					4
\$REAL OUTPUT					5
\$SUBCASE ID =	1				6
\$POINT ID =	1				7
TEMP*	1	1	7.270000E+02		8
TEMP*	2	1	7.270001E+02		9
TEMP*	3	1	7.270000E+02		10

\$TITLE	= RADIATION BOUNDARY CONDITION				1
\$SUBTITLE=					2
\$LABEL	=				3
\$TEMPERATURE					4
\$REAL OUTPUT					5
\$SUBCASE ID =	1				6
\$POINT ID =	1	IDENTIFIED BY TIME			7
TEMP*	1	1	7.270000E+02		8
TEMP*	2	1	7.270001E+02		9
TEMP*	3	1	7.270000E+02		10

When THERMAL analysis, the problem is a single degree of freedom system, therefore when the true content is identified, the scale of the output can be reduced.

\$TITLE = RADIATION BOUNDARY CONDITION	305
\$SUBTITLE=	306
\$LABEL =	307
\$VELOCITY	308
\$REAL OUTPUT	309
\$SUBCASE ID = 1	310
\$POINT ID = 1	311
0.000000E+00 S 1.817500E+00 0.000000E+00 0.000000E+00	312
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	313
\$TITLE = RADIATION BOUNDARY CONDITION	1409
\$SUBTITLE=	1410
\$LABEL =	1411
\$ENTHALPY	1412
\$REAL OUTPUT	1413
\$SUBCASE ID = 1	1414
\$POINT ID = 1 IDENTIFIED BY TIME	1415
0.000000E+00 S 1.817500E+00	1416

Same issues as previous.

\$TITLE = RADIATION BOUNDARY CONDITION	801
\$SUBTITLE=	802
\$LABEL =	803
\$ACCELERATION	804
\$REAL OUTPUT	805
\$SUBCASE ID = 1	806
\$POINT ID = 1	807
0.000000E+00 S 1.066395E-06 0.000000E+00 0.000000E+00	808
-CONT- 0.000000E+00 0.000000E+00 0.000000E+00	809
\$TITLE = RADIATION BOUNDARY CONDITION	2817
\$SUBTITLE=	2818
\$LABEL =	2819
\$H DOT	2820
\$REAL OUTPUT	2821
\$SUBCASE ID = 1	2822
\$POINT ID = 1 IDENTIFIED BY TIME	2823
0.000000E+00 S 2.730277E-04	2824

Same issues as ENTHLAPY.

\$TITLE	=LARGE PROBLEM				85
\$SUBTITLE	=				86
\$LABEL	=				87
\$LOADS					88
\$REAL OUTPUT					89
\$SUBCASE ID	=	1			90
101	S	1.500000E+03	0.000000E+00	0.000000E+00	99
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00	100
\$TITLE	=LARGE PROBLEM				85
\$SUBTITLE	=				86
\$LABEL	=				87
\$HEAT FLOW AT LOAD POINTS					88
\$REAL OUTPUT					89
\$SUBCASE ID	=	1			90
\$LOAD FACTOR	=	1.0000000E+00			90
101	S	1.500000E+03			95

Same issues as ENTHLAPY.

\$TITLE	=LARGE PROBLEM				247
\$SUBTITLE	=				248
\$LABEL	=				249
\$SPCF					250
\$REAL OUTPUT					251
\$SUBCASE ID	=	1			252
98	S	-2.466861E+02	0.000000E+00	0.000000E+00	257
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00	258
99	S	-1.051691E+04	0.000000E+00	0.000000E+00	259
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00	260
\$TITLE	=LARGE PROBLEM				169
\$SUBTITLE	=				170
\$LABEL	=				171
\$HEAT FLOW AT CONTRAINT POINTS					172
\$REAL OUTPUT					173
\$SUBCASE ID	=	1			174
\$LOAD FACTOR	=	1.0000000E+00			174
98	S	-2.466861E+02			177
99	S	-1.051691E+04			178



Thermal element data recovery contains mis-labeled and possible numerical errors.

\$TITLE	=	RADIATION BOUNDARY CONDITION					1649
\$SUBTITLE	=						1650
\$LABEL	=						1651
\$ELEMENT FORCES							1652
\$REAL OUTPUT							1653
\$SUBCASE ID	=	1					1654
\$ELEMENT TYPE	=	33					1655
\$ELEMENT ID	=	4					1656
		0.000000E+00	5.724519E+10	1.491285E-07	0.000000E+00		1657
-CONT-		0.000000E+00	1.401298E-45	0.000000E+00			1658
-CONT-		0.000000E+00	1.401298E-45				1659
\$TITLE	=	RADIATION BOUNDARY CONDITION					6233
\$SUBTITLE	=						6234
\$LABEL	=						6235
\$ELEMENT GRADIENTS AND FLUXES							6236
\$REAL OUTPUT							6237
\$SUBCASE ID	=	1					6238
\$ELEMENT TYPE	=	33	QUAD4				6239
\$ELEMENT ID	=	4	IDENTIFIED BY TIME				6240
		0.000000E+00	QUAD4		0.000000E+00		6241
-CONT-		0.000000E+00		1	0.000000E+00		6242
-CONT-		0.000000E+00		1			6243

## 13.11 Scale Factor for X2GG and X2PP Matrices

The K2GG, M2GG, B2GG, and P2G Case Control commands will accept the specifications of real scale factor(s) in MSC.Nastran 2004.

Example:

```
K2GG = 1.25*KDMIG
M2gg = 2.8*MDMIG1, 3.9*MDMIG2
```

where KDMIG, MDMIG1, and MDMIG2 are defined on the DMIG Bulk Data entries.

Similarly, the K2PP, M2PP, and B2PP Case Control commands will accept the specifications of complex or real scale factor(s) in MSC.Nastran 2004. Mixture of real and complex scale factors is not allowed.

Example:

```
K2PP = KDMIG
K2PP = KDMIG1, KDMIG2, KDMIG3
K2PP = 2.04*KDMIG1, 0.82*KDMIG2
K2PP = (2.04, .5)*KDMIG1, (0.82, 0.)*KDMIG2
```

## 13.12 Direct Input of Structural Element Damping and Area Matrices

MSC.Nastran has the capability of adding stiffness (K2PP or K2PP), mass (M2PP or M2PP), or damping matrices (B2GG or B2PP) directly to the model by referencing the appropriate DMIG entries for many versions. Starting in MSC.Nastran 2004, the user may also add structural element damping by using the K42GG Case Control command.

Example:

```
K42GG = KDMIG  
K42GG = KDMIG1, KDMIG2,KDMIG3  
K42GG = 2.04*KDMIG1,0.82*KDMIG2
```

Similarly, the Area matrix may be added by using the A2GG Case Control command.

Example:

```
A2GG = ADMIG  
A2GG = ADMIG1, ADMIG2,ADMIG3  
A2GG = 2.04*ADMIG1,0.82*ADMIG2
```

## 13.13 MSC.Access, DDLADD Revisions, and XDB Defects Correction

Starting with MSC.Nastran 2001.0.5, the DBC module has installed the Data Definition Language into the XDB for heterogeneous platform usage, thus reducing the required use of the TRANS/RECEIVE utilities to migrate the data base. An error was discovered concerning the CBAR, CBEAM, CBUSH, and CGAP objects that may cause unpredictable errors when operating in the heterogeneous environment. Using XDBs on the same platform, or using the TRANS/RECEIVE utilities, will eliminate the possible event. The DDLADD utility has been enhanced for MSC.Nastran 2004 to correct this error when these objects are present in the data base. A "mode" keyword has been added to the execution line that allows for optional replacement versus the default of insert. For example:

```
ddladd mode=replace my.xdb
```

Once complete, the xdb can be executed in a heterogeneous fashion. One needs to execute the DDLADD utility on the same platform upon which the xdb was created

## 13.14 Enhanced Free Field Format

The free field data entry capability in the MSC.Nastran has been enhanced to support easy to use data input formats. The following examples illustrate the possible forms of the free field data input and the resulting translation to the fixed-field format.

### Example 1 - Entry with or without user continuation mnemonics.

```
MATT9,1101,2 ,3 ,4 ,,,,8 ,+P101
+P101,9 ,,,,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1101	2	3	4				8	+P101
+P101	9				13				

```
GRID,100,,1.0,0.0,0.0,,456
```

Translates to:

GRID	100		1.0	0.0	0.0		456		
------	-----	--	-----	-----	-----	--	-----	--	--

### Example 2 - Entry with Automatic Continuation.

```
MATT9,1151,2 ,3 ,4 ,,,,8
,9 ,,,,13
```

Translates to:

MATT9	1151	2	3	4				8	
	9				13				

```
SPC1,100,12456,1,2,3,4,5,6,7,8,9,10
```

Translates to:

SPC1	100	12456	1	2	3	4	5	6	
	7	8	9	10					

Example 3 - Entry with continuation mnemonics are included with the data.

SPC1,100,12456,1,2,3,4,5,6,+SPC-A,+SPC-A,7,8,9,10

Translates to:

1	2	3	4	5	6	7	8	9	10
SPC1	101	12456	1	2	3	4	5	6	
	7	8	9	10					

Example 4 - Entry with single field continuations.

MATT9,1301,2,3,4,,,,8,+  
+,9,,,,13

Translates to:

MATT9	1301	2	3	4				8	+
+	9				13				

Example 5 - Entry with mixed single field, double field continuations.

MATT9\*,1302,2,,4,+  
+,,,,,13

Translates to:

MATT9*	1302	2				4		+
+					13			

MATT9,1303,2,3,4,,,,8,+  
\*,9,,,,+  
\*,13

Translates to:

MATT9	1303	2	3	4				8	+
*	9								+
*	13								

MATT9\*,1304,2 ,3 ,,+  
\*,,,,8

Translates to:

MATT9*	1304	2	3		+
*				8	

MATT9,1355,2 ,3 ,5 ,,,8 ,+  
\*,,10 ,,,+  
+,17

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1355	2	3		5			8	+
*			10					8	+
+	17								

Example 6 - Entry continued by terminating the parent entry with a comma.

For this feature, system cell 363 needs to be turned on or set to 1  
( i.e. system(363)=1 or STRICTUAI=1 )

CHEXA, 200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,  
17, 18, 19, 20

Translates to:

CHEXA	200	200	1	2	3	4	5	6	
	7	8	9	10	11	12	13	14	
	15	16	17	18	19	20			

**Note:** Because of the feature allowing more than 10 fields of data to be entered on one free field entry, IT IS NOT ALLOWED to terminate a single free field entry with a comma.

The "/" character may be entered at the beginning of the continuation data record to indicate the beginning of a new physical entry to simplify the data input.

MATT9,1153,2 ,3 ,4 ,,,,8 ,  
/,9 ,,,,13

MATT9	1153	2	3	4				8	
	9				13				



## 13.15 Dynamic Element Forces

The CBUSH, CELAS, and CDAMP elements now include the effect of damping in the element force calculation in dynamic analysis.

For the CBUSH, the force equations are:

Frequency:

$$F_e = [(1 + i g_t) K_e + i \omega B_e] U_e$$

where

$$g_t = g + g_e$$

Static:

$$F_e = K_e U_e$$

Transient:

$$F_e = K_e U_e(t) + \left[ B_e + \left( \frac{g}{w3} + \frac{g_e}{w4} \right) K_e \right] \dot{U}_e(t)$$

For the CELAS, the force equations are:

Static:

$$F = k(u_1 - u_2)$$

Frequency:

$$F = (1 + i(g + g_e)) k \bar{U} \quad ; \quad \bar{U} = (u_1 - u_2)_{\text{real}} + i(u_1 - u_2)_{\text{img}}$$

Transient:

$$F = (u_1 - u_2) k + \left( \frac{g}{W3} + \frac{g_e}{W4} \right) k (\dot{u}_1 - \dot{u}_2)$$

where  $k$  is the stiffness coefficient for the scalar element and  $u_1$  is the displacement of the first degree-of-freedom listed on its connection entry.

For the CDAMP, the force equations are:

Frequency:

$$F = i\omega B(u_1 - u_2)$$

Transient:

$$F = B(\dot{u}_1 - \dot{u}_2)$$

## 13.16 Sparse Data Recovery

In earlier versions of MSC.Nastran, the displacements are computed at all grid points regardless of what the user requested for data recovery in Case Control, OUTPUT(PLOT), and OUTPUT(XY\*) sections. For large models in the modal frequency and transient response solutions, the CPU time spent and disk space used in data recovery can far exceed the resources used in other operations regardless of whether the matrix method of data recovery (PARAM,DDRMM,0) is selected or not and, also, regardless of how many elements or grid points are requested for data recovery. In MSC.Nastran 2004, by default, the sparse data recovery method queries the Case Control for those elements and grid points requested for data recovery. Then displacements are computed at only those points. This can result in considerable savings in CPU and disk space usage especially if the user requests data recovery at a few grid points and/or elements. This method is also extended to SOL 200 and is particularly advantageous for small design models and a small set of design responses.

The sparse method involves creation of a partitioning vector with ones at those rows corresponding to degrees-of-freedom at which data recovery is requested. The partitioning vector is then used to partition transformation matrices involved in matrix multiplications used to perform load reduction and solution expansion. This results in smaller more dense matrices thereby enabling better method selection in the matrix multiplications.

### User Interface

The sparse data recovery method is controlled by four user parameters described under parameter “**SPARSEDR**” on page 677 of the *MSC.Nastran Quick Reference Guide*.

### Changes in Output Due to Sparse Data Recovery

The following changes to the f06 file should be noted with the sparse data recovery method. These changes also occur when PARAM,DDRMM,-1 is specified in MSC.Nastran 2001.

- In acoustic output under “COMPLEX ACCELERATIONS VELOCITIES AND PRESSURE LEVELS” output the “PRESSURE (DB)” column is no longer “N/A”.
- Complex force output is computed for viscous damping elements.

- The SORT1 formatted output of displacements, velocities, and accelerations appears in a different order between MSC.Nastran 2001 with PARAM,DDRMM,-1 and MSC.Nastran 2004:

V2001 SORT1 output order with PARAM,DDRMM,-1:

DISPLACEMENT at time step 1  
 VELOCITY at time step 1  
 ACCELERATION at time step 1  
 DISPLACEMENT at time step 2  
 VELOCITY at time step 2  
 ACCELERATION at time step 2  
 ... etc. ...

V2004 SORT1 output order:

DISPLACEMENT at time step 1  
 DISPLACEMENT at time step 2  
 ... etc. ...  
 VELOCITY at time step 1  
 VELOCITY at time step 2  
 ... etc. ...  
 ACCELERATION at time step 1  
 ACCELERATION at time step 2  
 ... etc. ...

## Limitations:

- Sparse data recovery is not available in aeroelastic analysis (in SOL 146), flutter analysis (in SOLs 145 and 200), complex eigenvalue analysis (in SOLs 107, 110, 145, and 200), nonlinear analysis (in SOLs 106, 129, 153, 159, 400, and 600) and cyclic symmetry analysis (in SOLs 114, 115, and 118).
- Sparse data recovery is deactivated when the following Case Control commands are specified: EKE, ESE, EDE, and CMSENERGY.
- PARAM,DDRMM,0 is no longer available with scr=mini and PARAM,EXTDROUT.

## DMAP Alters:

If you have DMAP alters which rely on the full solution matrix then you may specify the following alter to deactivate sparse data recovery:

```
compile fdrngen  
alter 'dospdr1=', ''  
dospdr1=false $  
endalter
```



# CHAPTER 14

## Upward Compatibility

- Results or Output Changes
- Modifications to OUTPUT2 (INPUTT2) Transport Formats
- Summary of DMAP Module Changes from MSC.Nastran 2001 to MSC.Nastran 2004
- Summary of Data Block Changes from MSC.Nastran 2001 to MSC.Nastran 2004
- Removal of Old Features
- MSC.Nastran Error List

## 14.1 Results or Output Changes

There are numerous enhancements of existing capabilities in MSC.Nastran 2004 that improve the results. The answers may change compared to MSC.Nastran 2001. The major sources for results changes are listed here.

- The default for the parameter K6ROT is changed from 0. to 100. to improve robustness and accuracy, see Section 5.10. The displacements of models containing QUAD4/TRIA3 elements may change slightly. In most practical problems, the change in displacements is below 0.1%. The changes in rotations are higher. In addition, the a-set size increases, see section 5.12 for more details.
- A new QUADR/TRIAR formulation has been implemented, see Section 5.10. The answers are more accurate. The user can, however, still run the old QUADR/TRIAR by adding "NASTRAN SYSTEM(370)=1" to the file.
- The enforced motion capability is now based on the relative motion approach rather than the absolute motion approach used in MSC.Nastran 2001, see Section 3.6. Results are more accurate in case of large base motion and modal damping. In addition, SPC and MPC forces are more accurate.
- With the exception of normal modes (SOL 103), residual vector processing is now the default in all modal dynamic analysis solution sequences. In MSC.Nastran 2001 residual vectors were employed only if the user requested it, see Section 3.5. In addition, the residual vector generation has been improved for better accuracy.
- A new coupling algorithm for fluid-structure interaction analysis has been developed, see Chapter 7. The accuracy is significantly enhanced. The algorithm always guarantees force balance at the fluid structure interface.
- Calculations of the modal matrices in the GKAM module and the calculations associated with the uncoupled path in the FRRD1 module are performed in machine precision (see Section 3.11). Results of modal solution sequences are more accurate.
- The CBUSH and CELAS elements now include the effect of damping in the element force results output in dynamic analysis.



- The Lagrange method has been implemented for rigid elements that produces differential stiffness contributions and allows large rotation, see Section 5.9. The results for pre-stressed normal modes, buckling, and geometric nonlinear analysis are more accurate if the Lagrange method is selected.
- Optimization (SOLution 200) results are likely to be affected by the enhancement under “**Elimination of Constraints on Properties that are a Linear Function of a Single Design Variable**” on page 88. It has been observed that this not only enhances performance for the optimizer, but that it also sometimes improves the quality of the results. As part of this enhancement, the default value for the DELX parameter has been changed from 1.0 to 0.5.
- Temperature dependent composite materials are no longer ignored in nonlinear analysis (SOL 106), see Section 5.8. The answers are more accurate.
- Two new connectivity formats have been added to the CWELD element. The results change compared to the formats offered in MSC.Nastran 2001. The new formats preserve symmetry, see Section 5.11.
- BAR torsional mass moment of inertia can now be included in the mass matrix by setting NASTRAN SYSTEM(398)=1. This will provide results comparable to the BEAM element for an equivalent structure.
- A new axisymmetric view factor capability has been integrated into the existing adaptive 3-D view factor process. These capabilities provide for the calculation of gray diffuse radiation view factors between CHBDYi of REV type, see Section 10.2. Heat transfer results for axisymmetric systems will change accordingly.
- The time step control in nonlinear transient analysis (SOL 129) has been improved for NOLINis. Answers are more accurate see Section 8.3.
- SOL 600 has been introduced which spawns MSC.Marc from MSC.Nastran. Results of the MSC.Marc computations may be brought back into MSC.Nastran and output to the .f06, .op2, .xdb, or punch files using the OUTR option on the SOL 600 Executive Control statement. These results will differ from those calculated internally by MSC.Nastran due to differences in element formulations and other factors for those classes of problems which can be solved by both programs. For fine-grid models, these differences are usually small. In addition, various warning messages may be produced by the internal MSC.Nastran-to-MSC.Marc translator. The .f06 file also contains DMAP generated on the fly to bring the results back into MSC.Nastran (if requested) as well as a summary of the t16op2 inverse translator as it

converts MSC.Marc's t15 results into MSC.Nastran's results. Finally, if the COPYR option on the SOL 600 Executive Control statement is used, MSC.Marc's output file will be copied directly to the .f06 file with or without text string substitutions.

- BAR torsional mass moment of inertia can now be included in the mass matrix by setting NASTRAN SYSTEM(398)=1. This will now provide results comparable to the BEAM element for an equivalent structure.
- A new axisymmetric view factor capabilities have been integrated into the existing adaptive 3D view factor process. This capability provides for the calculation of gray diffuse radiation view factors between CHBDYi of REV type.

There are numerous enhancements in MSC.Nastran 2004 which create additional output and which have changed existing output tables.

- When the number of modes used in a modal dynamic analysis is a subset of the computed modes, an additional eigenvalue table identifying the actual modes used in the dynamic analysis is output, see Section 3.11.
- The Case Control Command ELSUM is extended and the element summary table output is enhanced, see Section 5.3.
- Ply failure indices stored in the OEFIT table are output to the .op2 file and strength ratios are output, table OEFIT, see Section 5.6. Any post processor supporting op2 can now display ply failure indices.
- The coordinate system identification number (CSTM CID) is printed in the .f06 output when DISP, SPCF, etc. output is requested using the CID option, see Section 13.6 for a sample. The CID is not stored on the OFP data block and it is not punched, only printed.
- The existing tables GEOM1, GEOM2, GEOM3, GEOM4, EPT and AXIC have changed from 2001 to 2004. Post processors supporting the op2 output format and read the tables above must make adjustments. A system cell OP2NEW can be set so that MSC.Nastran 2004 puts out the tables in 2001 format, see Section 14.2.

Several new capabilities generate new output data blocks or files.

- Modal contribution fractions, table OMCFRAC, see Section 3.10.
- Acoustic modal participation factors, table MPF, see Section 7.3.
- Modal strain and kinetic energy, tables SNRGYPLT and KNRGYPLT, see Section 3.8.

- Output of maximum and minimum values using the MAXMIN Case Control command, table OMM, see Section 3.12.
- Force resultants for the equilibrium balance check using the EQUILIBRIUM Case Control command, table OEQUIL, see Section 5.4.
- Modal Neutral File (MNF) for Adams, see Section 11.1.
- Ply strength ratios, table OESRT, see Section 5.6.

## 14.2 Modifications to OUTPUT2 (INPUTT2) Transport Formats

Starting with MSC.Nastran 2004, the OUTPUT2 file transport format has been enhanced to better delineate the content to allow external applications to identify version and stride values for IFP class data blocks. Users are reminded of the parameter OMACHPR that impacts the IFP data block content by optionally executing the MAKEOLD DMAP module. Since MSC.Nastran Version 69, the purpose of OMACHPR has been to remove the machine precision controls embedded in the IFP data block record locate positions (first three words) and to change the RX entry items to RS matching the Version 68 and earlier releases. The roll of MAKEOLD for MSC.Nastran 2004 has changed to now edit or drop entry items to conform to the previous systems definitions. External applications are encouraged to migrate to the more modern entry formats and precision, otherwise information concerning the entry is lost. The role of MAKEOLD is perceived as a temporary rather than a permanent solution because improvements, such as increased precision to be used by the remaining processing structures, will cause an entry length change from the IFP output table perspective.

The new MSC.Nastran keyword for either the MSC.Nastran entry or RC file inclusion is OP2NEW and is linked to system cell 403. This is an integer value with a default of one (1), indicating that the new transport format is used. A value of zero (0) removes all version information and IFP locate word changes from the transport file. Version modification occurs both in the Label Record and within the Data Block Name record.

First the tape label. (See the *MSC.Nastran 2004 DMAP Programmer's Guide*.)

Physical Record	Length	Content
1	1	KEY=3
2	KEY	Date (3 words, month-day-year) - integer
3	1	KEY=7
4	KEY	Nastran Header (7 words, Character A4)
5	1	KEY=2
6	KEY	LABEL (2 words, Character A4)
7	1	KEY=3
8	KEY	VERSION (3 words, Major, Minor, Special, Character A4)
9	1	KEY=-1 (EOR)

Physical Record	Length	Content
10	1	KEY=0 (EOF)
11	1	KEY = 2
12	KEY	Data Block Name (2 words, Character A4)
13	1	KEY=3
14	KEY	VERSION (3 words, Major, Minor, Special, Character A4)
15	1	KEY=-1 (EOR)

Records 7 and 8 are present when OP2NEW=1 and VERSION would be the title page breakup into Major=2004, Minor=0, Special=0, for 2004.0.0.

Because the tape label is optional, the data block name area also requires modifications to denote the version. The reason is that open-ended IFP entries require special application coding to correctly process the item information. Records 7 and 8 from the tape label will be repeated as records 13 and 14 contained in the data block name record.

The next change to the OUTPUT2 transport format is only related to IFP data blocks such as GEOM2, and not to OFP or Matrices except for those changes indicated previously. Each IFP data block record beyond the name identification supplied as the first record, the entry content is preceded by three words to indicate the type of entry. External applications had to code explicit entry lengths tied to an unidentified version. This is changed for MSC.Nastran 2004 via the new OP2NEW keyword, where for the fixed entries, the length and precision are now passed as an encrypted value within the third word. The first and second words are unchanged. This definition is:

sssspiiii where iiii (4 digits) is the previous systems value, p ( 1 digit ) represents the real precision, and ssss ( high digits ) is the length of the entry

The value of precision (p) is either one (1) for RS or two (2) for RD. When set, all real items of the entry are of the indicated precision. The four digit iiii field with the top values are removed matches the enumeration supplied by previous systems and is the unique index to the bulk data entry. The [MSC.Nastran Quick Reference Guide](#) number contains 599 entries. Values of index above 599 are internally generated entries that have been merged into the data block. These records can only be present after IFP is completed and other modules have been executed. The ssss or stride is measured in words, where RD is counted as two words in the entry. The ssss and p values are zero for single precision open-end entries and only the p value is supplied

for open-ended entries of machine precision, such as the MPC entry on the GEOM4 data block. An application can recognize the third word encryption by testing against 10000. Values below follow previous release conventions, values above can utilize modular arithmetic to extract the old index value. It is suggested that a test be performed against the stride, when present, to allow the application to recognize entry rule base changes by changing entry length.

While the above is sufficient for fixed length entries, it does not recognize the effect of the OMACHPR parameter setting to cause the optional execution of the MAKEOLD module and the conversion of the open-ended records. To indicate conversion to “OLD” entry structure, the character “O” is placed as the last character of the Special of the Version field. When present in the Version, the format would be Special Character, two blanks, and the next character “O” for old or blank for the current version. The IFP data blocks currently being converted are:

Generic Old NDDL Version Converted records

GEOM1	GEOM168	Machine Precision GRID, CORD2j
GEOM2	GEOM201	Shell element corner thickness and BAR/BEAM offset and orientation coordinate systems
GEOM3	GEOM301	Extension to PLOAD4 entry
GEOM4	GEOM4705	Machine Precision MPC coefficients and Rigid Elements
EPT	EPT01	Number of Damping Coefficients for PBUSH and BUSHT entries
AXIC	AXIC68	Various entry such as POINTAX, SECTAX which contain machine precision geometric values

The known entry changes to the IFP data blocks for MSC.Nastran 2004 are:

GEOM2  
CQUAD8, CTRIA6, CWELD extended by one word  
CQUAD4, CTRIA3, CQUADR, CTRIAR, CBAR, CBEAM have entry item changes

EPT  
PBUSH, PBUSHT extended by 5 words

GEOM4  
RBAR, RBE1, RBE2, RBE3, RROD, RTRPLT extended by one word or have opened-ended record process implications

GEOM3  
PLOAD4 extended by four words

For applications interested in new records on the IFP data blocks and changes to others, it is suggested that compiled NDDL output from the current system and the previous system be differenced to fully identify these variations. The “Old NDDL” data block descriptions are also present and reflect the current version record content.

## 14.3 Summary of DMAP Module Changes from MSC.Nastran 2001 to MSC.Nastran 2004

This section summarizes DMAP module changes from MSC.Nastran 2001 (2001.0.9) to MSC.Nastran 2004 which could affect DMAP alters and solution sequences. This information is intended to help the user convert MSC.Nastran 2001 DMAP alters and solution sequences to run in MSC.Nastran 2004. The format of the following modules has been modified in MSC.Nastran 2004 such that the MSC.Nastran 2001 format is not upward compatible with MSC.Nastran 2004 and/or their behavior is not upward compatible. The changes are described in the next section.

ASDR	ASG	DCMP	DDRMM	DPD	DSADJ	DSAL
DSVG1	EFFMASS	INPUTT2	MODACC	MODEPOUT	MTRXIN	OFP
ORTHOG	OUTPRT	OUTPUT2	SDR2	XYTRAN		

The following is a list of existing modules with new features in MSC.Nastran 2004. The new features are documented in the *MSC.Nastran 2004 DMAP Programmer's Guide*.

BCDR	BDYINFO	CASE	DBC	DBVIEW	DECOMP	DISOPT
DISUTIL	DOM10	DOM11	DOM12	DOM9	DOPR1	DOPR3
DSAD	DSAF	DSAH	DSAR	DSPRM	DSTAP2	DSVGP4
ELTPRT	EMA	EMG	GETCOL	GKAM	GP2	GP3
GP4	GP5	GPFDR	GPSP	IFP	IFP1	INPUTT2
LCGEN	MATMOD	MCE1	MCE2	MDCASE	MODGM2	MODGM4
NLITER	NLTRD2	OFP	OUTPUT2	PARAML	PRESOL	RANDOM
SDRP	SDSA	SEDR	SEP1	SEP1X	SEP4	SEQP
SSG1	SSG2	TA1	TASNP2	TRD1	TRLG	VDR
VECPLT	XSORT	XYTRAN				



The following is a list of new modules in MSC.Nastran 2004. They are documented in the *MSC.Nastran 2004 DMAP Programmer's Guide*.

ADAMSMNF	ADJMOD	CMSENGY	DISPARM	DLT2SLT	DOPR3X	DSADX
GYROLD	IFP10	INDXBULK	MATOFP	MCFRAC	MODENRGY	MODQSET
NEWUSET	NLRSLOOP	NLRSMAP	NLSOLV	NLTRLG	NSMEPT	RMDUPBLK
ROTOR	ROTRDR1	ROTRDR2	SEEFMBND	SEEFMCLF	SEEFMDMP	SEEFMLST
SEEFMNOR	SEEFMOUT	SEEFMXIT	ST2DYN			

## DMAP Module Changes

The following pages show the changes for DMAP module instructions which were updated in MSC.Nastran 2004. The module change descriptions are presented as differences with respect to the *MSC.Nastran 2001 DMAP Programmers' Guide* which is available on the "MSC.Nastran 2001 - MSC.Patran 2001 Documentation" CD-ROM. The change description includes the MSC.Nastran 2004 format of the module with changes in bold text. Any new or changed data blocks and parameters are also described below the format.

# ASDR Module

The MPAERP data block has been inserted at the 16th input position.

## Format:

ASDR            CASEA , UXDAT , AECTRL , FFAJ , ACPT , PAK , AEUSET , AEBGPDT ,  
                 AECOMP , MONITOR , MPSR , MPSEr , MPSIR , MPSRP , MPSErP , **MPAERP** ,  
                 AEMONPT , MPAER , AERO , CSTMA / / MACH / Q / AECONFIG /  
                 SYMXY / SYMXZ / IUNIT SOL   \$

## Input Data Block:

MPAERP        Total elastic restrained loads on aerodynamic monitor points at trim  
                 due to static applied loads.

## ASG Module

The MPAERP data block has been inserted at the 9th input position.

### Format:

```
ASG          CASEA , AEMONPT , MONITOR , MPAERV , MPSEVP , MPSIR , AEDBUXV ,
              MPSEVP , MPAERP , AECTRL , EDT , PRBDOFS , DIT , AEDBINDX / UX ,
              UXDAT , UXDIFV / SYMXZ / ISENS $
```

### Input Data Block:

MPAERP	Total elastic restrained loads on aerodynamic monitor points at trim due to static applied loads.
--------	---

## DDRMM Module

OPTION parameter default is changed from 'ABS' to ' ' (blank).

### Format:

```
DDRMM      CASECC , UH , OL , IUG , IQG , IES , IEF , XYCDB / OUG , OQG , OES , OEF ,
            UNUSED5 , OVG , OAG / OPTION / NOCOMP / PEXIST / ACOUSTIC / ACOUT /
            PREFDB / SEID / DVAFLAG $
```

### Parameters:

OPTION     Input-character-default=' '. Response summation method for scaled response spectra analysis only. Possible values are:

'ABS'   absolute

'SRSS'   square root of the sum of the squares

'NRL'    Naval Research Laboratory (new)

'NRLO'   Naval Research Laboratory (old)

## DCMP Module

The mechanism flag, ERR, is now set to -1 only if the factor to diagonal ratio is greater than MAXRATIO.

### Format:

```
DCMP      USET,SIL,EQEXIN,A,PARTVEC,EQMAP/
          LD,U,LSEQ/
          S,N,KSYS/CHOLSKY/BAILOUT/MAXRATIO/SETNAME/F1/DECOMP/
          DEBUG/THRESH/S,N,MINDIAG/S,N,DET/S,N,POWER/S,N,SING/
          S,N,NBRCHG/s,n,err/LMTROWS $
```

### Parameter:

ERR	Output-integer-default=-1. If BAILOUT=-1, this parameter always remains at zero. If BAILOUT=0 and greater than MAXRATIO, ERR is reset to -1.
-----	--

## DPD Module

The tenth parameter has been re-typed and re-used.

### Format:

```
DPD          DYNAMIC , GPL , SIL , USET , UNUSED5 , PG , PKYG , PBYG , PMYG , YG /  
             GPLD , SILD , USETD , TFPOOL , DLT , PSDL , RCROSSL , NLFT , TRL ,  
             EED , EQDYN / LUSET / S , N , LUSETD / S , N , NOTFL / S , N , NODLT /  
             S , N , NOPSDL / DATAREC /  
             S , N , NONLFT / S , N , NOTRL / S , N , NOEED / SORTNLFT / S , N , NOUE /  
             UNUSED12 / SEID $
```

### Parameter:

**SORTNLFT** Input-logical-default=FALSE. Sort flag for NLFT records.

## DSADJ Module

Central differencing is now done in a single call to DSADJ. DSADJ also includes pseudo-restart logic for efficiency.

### Format:

```
DSADJ      XDICTDS , XELMDS , BGPDT , CSTM , XDICTX , XELMX , UGX , ADJG ,  
           DRDUTB , DSPT1 , VGDM , IMATG , XDICTB , XELMB , UGT0 , ADJGT0 ,  
           DRUT0 / ADELX , UGT , ADJGT , DRUT / NOK4GG / WTMASS / XTYPE / CDIF /  
           COUPMASS / SHAPEOPT / SPDM / ADJMETH  $
```

### Input Data Blocks:

IMATG	Pesudo identity g-set matrix.
XDICTB	Backward perturbed element matrix dictionary if CDIF='YES'.
XELMB	Backward perturbed if CDIF='YES'
UGT0	UGT computed in a prior call to DSADJ and specified on second pass through DSADJ.
ADJGT0	ADJGT computed in a prior call to DSADJ and specified on second pass through DSADJ.
DRUT0	DRUT computed in a prior call to DSADJ and specified on second pass through DSADJ.

### Output Data Blocks:

UGT	Transposed regular solution matrix specified on first pass through DSADJ.
ADJGT	Transposed adjoint solution matrix specified on first pass through DSADJ.
DRUT	Processed version of DRDUTB specified on first pass through DSADJ.

## DSAL Module

ADELX output data block is replaced by ADELUS and ADELUF. The new input data blocks, OGPFDSN, OELOPDSN, DELGS, and DELGM, have been inserted at the 17th, 18th, 27th, and 28th positions.

### Format:

```

DSAL      DRSTBL,DELWS,DELVS,DELB1,DELF1,
          COGRID,COELEM, OUGDSN, OESDSN, OSTRDSN,
          OEFDSN, OEFITDSN, OESCDSN, OSTRCDSN,
          OQGDSN, ONRGYDSN, OGPFDSN, OELOPDSN, R1VALR, TABDEQ, OL,
          DSDIV, DELX, DELS, DELFL, DELCE, DELGS, DELGM, FRQRPR,
          DELBSH, DRDUTB, ADELUS, ADELUF, R1TABR, DRMSVL, MODRPR,
          OUGDSNM, OESDSNM, OSTRDSNM, OEFDSNM, ONRGDSNM/DSCM/
          NDVTOT/DELTAB/EIGNFREQ/ADJFLG/SEID/S,N,ADELRS/
          S,N,ADELRF  $

```

### Input Data Blocks:

- OGPFDSN    Table of grid point forces for the perturbed configuration.
- OELOPDSN    Table of element forces on adjacent elements
- DELGS       Matrix of delta generalized stiffnesses for all design variables
- DELGM       Matrix of delta generalized masses for all design variables
- ADELUS       Matrix of adjoint sensitivities for static analysis.
- ADELUF       matrix of adjoint sensitivities for frequency response.



## DSVG1 Module

LAMA is replaced by LFTAB.

### Format:

```
DSVG1      XDICTDS , XELMDS , BGPDT , SIL , CSTM , XDICT , XELM , {  UGX  } ,
                                     {  AGX  } ,
          VG , LFTAB , DSPT1 , VGDM , XDICTB , XELMB / EGX / NOK4GG / WTMASS /
          IAPP / DSVGSF / NOPSLG / COUPMASS / CDIF / SPDM / SHAPEOPT $
```

### Input Data Blocks:

- LFTAB Table of eigenvalues and generalized masses for retained normal mode eigenvalue responses.
- XDICTB Backward perturbed element matrix dictionary if CDIF='YES'.
- XELMB Backward perturbed if CDIF='YES'

### Parameters:

- CDIF Input-character-default='NO'. Finite difference scheme. 'YES'=Central and Forward='NO'.

## EFFMASS Module

The first parameter is now unused. In MSC.Nastran 2001, the value of this parameter represented the punch file line counter and is incremented by one for each line written to the punch file and is also written into columns 73-80 of each line in the punch file. In MSC.Nastran 2004, this value is now stored in system cell 397.

### Format:

```
EFFMASS    CASECC , MAA , PHA , LAMA , USET , BGPDT , UNUSED , CSTM , VGQ /
            EMF , EMM , DMA , MEMF , MPFEM , MEM , MEW /
            SEID / WTMASS / unused / SETNAM / UNUSED $
```

### Parameters:

UNUSED     Input/output-integer-default=0. Unused and may be unspecified.

## INPUTT2 Module

A tape label is no longer required on the Fortran unit designated by IUNIT when ITAPE<0.

### Format:

```
INPUTT2      /DB1,DB2,DB3,DB4,DB5 /
              itape/IUNIT/LABL/
              S,N,HNAME1/S,N,HNAME2/S,N,HNAME3/S,N,HNAME4/
              S,N,HNAME5 $
```

### Parameter:

**ITAPE**        Input-integer-default = 0. ITAPE controls the status of the unit before INPUTT2 attempts to extract any data blocks. The following controls are available.

ITAPE Value	Action
+n	Skip forward n data blocks before reading.
0	Data blocks are read starting at the current position.
-1	Rewind before reading and position tape past label (LABL), if any.
-3	Print data block names, rewind before reading, and position tape past label, if any.

# MODACC Module

## Format:

MODACC        CASECC,OL,U,P1,P2,P3/  
              OL1,U1,P11,P21,P31/APP/**S,N,MDCEQV** \$

## Parameter:

MDCEQV        Output-integer-default=-1. MODACC equivalence flag. If MDCEQV=-1 on output then no output truncation occurred and a subsequent EQUIVX statement may be used to equivalence the inputs to the outputs. Applicable only when APP='REIG'.

## Remarks:

1. Under APP='REIG', if there is no OMODES or OFREQ Case Control request and the number of subcase equals the number of modes (columns in U) or the MODES Case Control command generates a number of subcases equal to the number of modes then OL1 and U1 will not be created and MODACC must be followed subsequent EQUIVX statement; e.g.

```
MODACC            CASEM,LAMA,PHA,, , /  
                  LAMA2,PHA1,, , /APP/S,N,MDCEQV $  
  
EQUIVX            LAMA/LAMA2/MDCEQV $  
  
EQUIVX            PHA/PHA1/MDCEQV $
```

## MODEPOUT Module

The FMODE and FMODEF parameters have been added to the 7th and 8th positions and they have no default.

### Format:

```
MODEPOUT  LAMAF , LAMAS , CASECC , FMPF , SMPF ,  
           PMPF , LMPF , GMPF , MPFMAP ,  
           MODELSTS , MODELSTF / OFMPF2E , OFMPF2M , OSMPF2E ,  
           OSMPF2M , OPMPF2E , OPMPF2M , OLMPF2E , OLMPF2M ,  
           OGMPF2E , OGMPF2M , UNUSED1 , UNUSED2 , UNUSED3 , UNUSED4 ,  
           UNUSED5 / OUTFMP / OUTSMP / FMPFEPS / SMPFEPS / MPFSORT /  
           NOMPF2E / FMODE / FMODEF  $
```

### Parameters:

FMODE	Input-integer-no default. The lowest structure mode number resulting from user parameter input of LMODES, LFREQ, and HFREQ and also MODESELECT(STRUCTURE) Case Control input.
FMODEF	Input-integer-no default. The lowest fluid mode number resulting from user parameter input of LMODESFL, LFREQFL, and HFREQFL and also MODESELECT(FLUID) Case Control input.

## MTRXIN Module

MTRXIN can now process up to five matrices depending on the form and therefore several existing parameters have been shifted.

### Format:

#### Form 1:

$$\begin{aligned} \text{MTRXIN} \quad & , , \text{MATPOOL}, \left\{ \begin{array}{c} \text{EQEXIN} \\ \text{EQDYN} \end{array} \right\} , , / \\ & \text{NAME1}, \text{NAME2}, \text{NAME3}, \mathbf{NAME4}, \mathbf{NAME5} / \\ & \left\{ \begin{array}{c} \text{LUSET} \\ \text{LUSETD} \end{array} \right\} / \text{S}, \text{N}, \text{NONAME1} / \text{S}, \text{N}, \text{NONAME2} / \text{S}, \text{N}, \text{NONAME3} / \\ & \mathbf{S}, \mathbf{N}, \mathbf{NONAME4} / \mathbf{S}, \mathbf{N}, \mathbf{NONAME5} \quad \$ \end{aligned}$$

#### Form 2:

$$\begin{aligned} \text{MTRXIN} \quad & \text{CASECC}, \text{MATPOOL}, \left\{ \begin{array}{c} \text{EQEXIN} \\ \text{EQDYN} \end{array} \right\} , , \left\{ \begin{array}{c} \text{TFPOOL} \end{array} \right\} / \\ & \left\{ \begin{array}{c} \text{K2GG}, \text{M2GG}, \text{B2GG}, \mathbf{K42GG}, \mathbf{A2GG} \\ \text{K2PP}, \text{M2PP}, \text{B2PP} \end{array} \right\} \\ & \left\{ \begin{array}{c} \text{LUSET} \\ \text{LUSETD} \end{array} \right\} / \text{S}, \text{N}, \text{NOK2} / \text{S}, \text{N}, \text{NOM2} / \text{S}, \text{N}, \text{NOB2} / \mathbf{S}, \mathbf{N}, \mathbf{NOK42} / \\ & \mathbf{S}, \mathbf{N}, \mathbf{NOA2} / \left\{ \begin{array}{c} 1 \\ 0 \end{array} \right\} \quad \$ \end{aligned}$$

## Form 5:

$$\begin{aligned}
 & \text{MTRXIN} \quad , , \text{MATPOOL} , \left\{ \begin{array}{c} \text{EQDYN} \\ \text{EQEXIN} \\ \text{EQEXIN} \end{array} \right\} , , \left\{ \begin{array}{c} \text{TFPOOL} \\ \\ \end{array} \right\} / \\
 & \left\{ \begin{array}{c} \text{MATP1} \\ \text{MATG1} \\ \text{RMATG} \end{array} \right\} , \left\{ \begin{array}{c} \text{MATP2} \\ \text{MATG2} \end{array} \right\} , \left\{ \begin{array}{c} \text{MATP3} \\ \text{MATG3} \end{array} \right\} , \left\{ \begin{array}{c} \text{MATP4} \\ \text{MATG4} \end{array} \right\} , \left\{ \begin{array}{c} \text{MATP5} \\ \text{MATG5} \end{array} \right\} / \\
 & \left\{ \begin{array}{c} \text{LUSETD} \\ \text{LUSET} \\ \text{LUSET} \end{array} \right\} / \text{S}, \text{N}, \text{NOMAT1} / \text{S}, \text{N}, \text{NOMAT2} / \text{S}, \text{N}, \text{NOMAT3} / \text{S}, \text{N}, \text{NOMAT4} / \\
 & \text{S}, \text{N} / \text{NOMAT5} / \left\{ \begin{array}{c} 10 \\ 11 \\ 12 \end{array} \right\} / \\
 & \text{MATNAM1} / \text{MATNAM2} / \text{MATNAM3} / \text{MATNAM4} / \text{MATNAM5} / \left\{ \begin{array}{c} \text{TFLID} \end{array} \right\} / \text{NFEXIT} \$ \\
 & \text{MATID1} / \text{MATID2} / \text{MATID3} / \text{MATID4} / \text{MATID5} \$
 \end{aligned}$$

## Parameters:

**MATIDXi**     Input-integer-default=-1. Index to the current name in the SET containing the DMIG names.

## OFP Module

The first parameter is now unused. In MSC.Nastran 2001, the value of this parameter represented the punch file line counter and is incremented by one for each line written to the punch file and is also written into columns 73-80 of each line in the punch file. In MSC.Nastran 2004, this value is now stored in system cell 397.

### Format:

```
OFP      OFP1 , OFP2 , OFP3 , OFP4 , OFP5 , OFP6 ,
          CSTM , BGPDTVU , ERROR1 , DEQATN , DEQIND , DIT , UNUSED //
          unused / ODCODE / PVALID / OCID $
```

### Parameters:

UNUSED     Input/output-integer-default=0. Unused and may be unspecified.



## OUTPRT Module

The CASEDS input data block has been inserted at the 10th position.

### Format:

```
OUTPRT      CASECC , ECT , BGPDT , SIL , XYCDB , DYNAMIC , MATPOOL , PG , VGFD ,  
            CASEDS , TABEVP , TABEVS , MMCDB , OINT , PELSET / PVGRID , PVSPC ,  
            PVMPC , PVLOAD , PVMPC / S , N , SDRMETH / NOSE / SDROVR / SDRDENS /  
            ADPTINDX / ADPTEXIT $
```

### Input Data Block:

CASEDS      Case control table for the data recovery of design responses.

## ORTHOG Module

The ORTHOG module cannot orthogonalize complex matrices. If a complex matrix is input then a fatal message is issued. SubDMAP ONORM can orthogonalize complex matrices. See "General Enhancements for Orthogonalizing Complex Vectors" in this Guide.

## OUTPUT2 Module

Starting with MSC.Nastran 2004 the OUTPUT2 file transport format has been enhanced to better delineate the content to allow external applications to identify version and stride values for IFP class datablocks. For more details see "Modifications to OUTPUT2 (INPUTT2) Transport Formats" in this guide and the *MSC.Nastran 2004 DMAP Programmer's Guide*.

## SDR2 Module

EDT can no longer be specified in the 8th input position and must be specified in the 24th input position. OES and OSTR and OQG and OQMG may now be output in a single call to SDR2. The OFP outputs may be in SORT1 or SORT2 format depending on the value of the 2nd parameter, SORTFLAG. The number of values that can be specified for NOCOMP are now reduced.

### Format:

```
SDR2      CASECC , CSTM , MPT , DIT , EQEXIN ,
          SIL , ETT , OL , BGPDT , PG ,
          QG , UG , EST , XYCDB , OINT ,
          PELSET , VIEWTB , GPSNT , DEQATN , DEQIND ,
          DITID , PCOMPT , GPKE , EDT , VG
          AG , QMG , MMCDB /
          OPG , OQG , UG , OES , OEF ,
          PUG , OVG , OAG , OQMG , OGPKE1 /
          OSTR , OMM1 /
          APP / S , N , SORTFLAG / NOCOMP / ACOUSTIC / DVAFLAG /
          ISOFLG / GPF / ACOUT / PREFDB / TABS /
          SIGMA / ADPTINDX / ADPTEXTIT / BSKIP / W3 /
          W4 / LANGLE / OMID / G / S , N , OCID $
```

### Output Data Blocks:

OPG	Table of applied loads in SORT1 or SORT2 format.
OQG	Table of single forces-of-constraint in SORT1 or SORT2 format.
UG	Table of displacements in SORT1 or SORT2 format.
OES	Table of element stresses in SORT1 or SORT2 format.
OEF	Table of element forces in SORT1 or SORT2 format.
OVG	Table of velocities in SORT1 or SORT2 format.
OAG	Table of accelerations in SORT1 or SORT2 format.
OQMG	Table of multipoint forces-of-constraint in SORT1 or SORT2 format.
OSTR	Table of element stresses in SORT1 or SORT2 format.

**Parameters:**

**SORTFLAG** Input/Output; Default=1. SORT1/SORT2 format flag.

If 1 on input then:

The columns of UG, QG, and QMG correspond to frequencies, time steps, or eigenvalues, etc. depending on APP and output will be in SORT2 format

Then on output:

set to 1 if (1) SORT2 format is requested or XYCDB exists, or (2) APP='TRANRESP' with no SORT1 requests; -1 otherwise.

If 2 on input then:

The rows of UG, QG, and QMG correspond to frequencies, time steps, or eigenvalues, etc. depending on APP and output will be in SORT2 format

Then on output:

set to -1 if XYCDB does not exist and (1) SORT1 format is requested or (2) APP<>'TRANRESP' with no SORT2 requests; 1 otherwise.

**NOCOMP** Input-Integer; Default=1. Composite stress/strain flag.

- 0 Compute stresses for all elements
- 1 Compute stresses for non-composite elements only
- 2 Compute element forces and strain curvatures ofr composite elements in the set referenced by STRESS=sid
- 3 Compute element forces and strain curvatures ofr composite elements using the set referenced by STRAIN=sid

**Remarks:**

- 2. If SORT2 format is desired for the outputs then the parameter SORTFLAG must be set to 2 and the input data blocks UG, QG, and QMG must be in their transposed form.

## XYTRAN Module

The fourth parameter is now unused. In MSC.Nastran 2001, the value of this parameter represented the punch file line counter and is incremented by one for each line written to the punch file and is also written into columns 73-80 of each line in the punch file. In MSC.Nastran 2004, this value is now stored in system cell 397.

### Format:

```

XYTRAN      XYCDB, {
                PSDF
                OFP12
                OVG
                OXRESP
            }, {
                AUTO
                OFP22
            }, OFP32, OFP42, OFP52/

            XYPLOT/
            APP/EXSET/S,N,PLTNUM/unused/S,N,NOXPLOT/
            S,N,TABID/DVAFLAG $

```

### Parameters:

**UNUSED**     Input/output-integer-default=0. Unused and may be unspecified.

## 14.4 Summary of Data Block Changes from MSC.Nastran 2001 to MSC.Nastran 2004

The following material describes changes in data blocks for only those records that existed in MSC.Nastran 2001. All new records added in MSC.Nastran 2004 are described in the *MSC.Nastran 2004 DMAP Programmer's Guide*.

## DYNAMIC - Table of Bulk Data Entry Images Related to Dynamics

1. Added real-valued words for time delay and/or phase lead to the ACSRCE, RLOAD1, RLOAD2, TLOAD1, and TLOAD2 records.
2. Changed FLAG1 and FLAG2 from integer to logical values in the EIGRL record.

### ACSRCE(5307,53,379)

Added two new items: T and PH for time delay and phase lead.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	DAREA	I	DAREA Bulk Data entry identification number
3	DPHASE	I	DPHASE Bulk Data entry identification number
4	DELAY	I	DELAY Bulk Data entry identification number
5	TC	I	TABLEDi Bulk Data entry identification number for C(f)
6	RHO	RS	Density of the fluid
7	B	RS	Bulk modulus of the fluid
8	T	RS	Time delay
9	PH	RS	Phase lead

### EIGRL(308,8,348)

FLAG1 and FLAG2 are changed from integer to logical values.

Word	Name	Type	Description
1	SID	I	Set identification number
2	V1	RS	Lower bound of frequency range of interest
3	V2	RS	Upper bound of frequency range of interest
4	ND	I	Number of desired eigenvectors



Word	Name	Type	Description
5	MSGVLV	I	Diagnostic level
6	MAXSET	I	Number of vectors in block or set
7	SHFSCL	RS	Estimate of first flexible mode
8	FLAG1	LOGI	V1 specification flag - set to 1 if V1 is specified
9	FLAG2	LOGI	V2 specification flag - set to 1 if V2 is specified
10	NORM	CHAR4	Method for normalizing eigenvectors
11	ALPH	RS	Constant for quadratic frequency segment distribution
12	NUMS(C)	I	Number of frequency segments
13	FI	RS	Frequency at the upper boundary of the i-th segment

Word 13 repeats 15 times

### **RLOAD1(5107,51,131)**

Added two new items: T and PH for time delay and phase lead.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	DAREA	I	DAREA Bulk Data entry identification number
3	DPHASE	I	DPHASE Bulk Data entry identification number
4	DELAY	I	DELAY Bulk Data entry identification number
5	TC	I	TABLEDi Bulk Data entry identification number for C(f)
6	TD	I	TABLEDi Bulk Data entry identification number for D(f)
7	TYPE	I	Nature of the dynamic excitation

Word	Name	Type	Description
8	T	RS	Time delay
9	PH	RS	Phase lead

### RLOAD2(5207,52,132)

Added two new items: T and PH for time delay and phase lead.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	DAREA	I	DAREA Bulk Data entry identification number
3	DPHASE	I	DPHASE Bulk Data entry identification number
4	DELAY	I	DELAY Bulk Data entry identification number
5	TB	I	TABLEDi Bulk Data entry identification number for B(f)
6	TP	I	TABLEDi Bulk Data entry identification number for Phi(f)
7	TYPE	I	Nature of the dynamic excitation
8	T	RS	Time delay
9	PH	RS	Phase lead

### TLOAD1(7107,71,138)

Added one new item: T for time delay.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	DAREA	I	DAREA Bulk Data entry identification number
3	DELAY	I	DELAY Bulk Data entry identification number
4	TYPE	I	Nature of the dynamic excitation

Word	Name	Type	Description
5	TID	I	Identification number of TABLEDi entry that gives F(t)
6	T	RS	Time delay

**TLOAD2(7207,72,139)**

Added one new item: T for time delay.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	DAREA	I	DAREA Bulk Data entry identification number
3	DELAY	I	DELAY Bulk Data entry identification number
4	TYPE	I	Nature of the dynamic excitation
5	T1	RS	Time constant 1
6	T2	RS	Time constant 2
7	F	RS	Frequency
8	P	RS	Phase angle
9	C	RS	Exponential coefficient
10	B	RS	Growth coefficient
11	T	RS	Time delay

## EPT - Element Property Table

For both the PBUSH and PBUSHT records there are now six values for the structural damping coefficient where there used to be one. By default, for negative values of user PARAMeter POST, these records are automatically restored to their MSC.Nastran 2001 formats on the resulting op2 files. See the “[Modifications to OUTPUT2 \(INPUTT2\) Transport Formats](#)” on page 570.

### PBUSH(1402,14,37)

There are now six values for GE where there used to be one.

Word	Name	Type	Description
1	PID	I	Property identification number
2	K(6)	RS	Nominal stiffness values
8	B(6)	RS	Nominal damping coefficient
14	GE(6)	RS	Nominal structural damping constant
20	SA	RS	Stress recovery coefficient in the translational component
21	ST	RS	Stress recovery coefficient in the rotational component
22	EA	RS	Strain recovery coefficient in the translational component
23	ET	RS	Strain recovery coefficient in the rotational component

### PBUSHT(702,7,38)

There are now six values for TGEID where there used to be one.

Word	Name	Type	Description
1	PID	I	Property identification number
2	TKID(6)	I	TABLEDi entry identification numbers for stiffness
8	TBID(6)	I	TABLEDi entry identification numbers for viscous damping

Word	Name	Type	Description
14	TGEID(6)	I	TABLEDi entry identification number for structural damping
20	TKNID(6)	I	TABLEDi entry IDs for force vs. deflection

## GEOM2 - Table of Bulk Data entries related to element connectivity

1. The relative thickness flag, TFLAG, has been added to the CQUAD8 and CTRIA6 records.
2. A previously undefined item in the CQUAD4, CQUADP, CTRIA3, CTRIAP, CQUADR, and CTRIAR records is replaced with the relative thickness flag, TFLAG.
3. The orientation flag, F, in the CBAR and CBEAM record is replaced with a bit pattern which must be logically added to 65539 to get the values defined in V2001.
4. CWELD record has a totally new format.

By default for negative values of user PARAMeter POST, all the changes above are automatically restored to the MSC.Nastran 2001 formats on the resulting op2 files. See the “[Modifications to OUTPUT2 \(INPUTT2\) Transport Formats](#)” on page 570.

### CBAR(2408,24,180)

The orientation flag, F, is replaced with a bit pattern which must be logically added to 65536 to get the values defined in MSC.Nastran 2001.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	GA	I	Grid point End A identification number
4	GB	I	Grid point End B identification number
F,65539 =0		Z	
5	X1	RS	T1 component of orientation vector from GA
6	X2	RS	T2 component of orientation vector from GA
7	X3	RS	T3 component of orientation vector from GA
8	F	I	Orientation vector flag = 1
F,65539 =1		XYZ option - global coordinate system	
5	X1	RS	T1 component of orientation vector from GA

Word	Name	Type	Description
6	X2	RS	T2 component of orientation vector from GA
7	X3	RS	T3 component of orientation vector from GA
8	F	I	Orientation vector flag = 1
F,65539 =2		Grid option	
5	GO	I	Grid point ID at end of orientation vector
6	UNDEF(2)	none	
8	F	I	
End F,65539			Orientation vector flag = 2
9	PA	I	Pin flags for end A
10	PB	I	Pin flags for end B
11	W1A	RS	T1 component of offset vector from GA
12	W2A	RS	T2 component of offset vector from GA
13	W3A	RS	T3 component of offset vector from GA
14	W1B	RS	T1 component of offset vector from GB
15	W2B	RS	T2 component of offset vector from GB
16	W3B	RS	T3 component of offset vector from GB

**CBEAM(5408,54,261)**

The orientation flag, F, is replaced with a bit pattern which must be logically added to 65536 to get the values defined in MSC.Nastran 2001.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	GA	I	Grid point End A identification number
4	GB	I	Grid point End B identification number
5	SA	I	Scalar or grid point End A identification number for warping

Word	Name	Type	Description
6	SB	I	Scalar or grid point End B identification number for warping
<b>F,65539 =0</b>		Y	
7	X1	RS	T1 component of orientation vector from GA
8	X2	RS	T2 component of orientation vector from GA
9	X3	RS	T3 component of orientation vector from GA
10	F	I	Orientation vector flag = 0
<b>F,65539 =1</b>		XYZ option - global coordinate system	
7	X1	RS	T1 component of orientation vector from GA
8	X2	RS	T2 component of orientation vector from GA
9	X3	RS	T3 component of orientation vector from GA
10	F	I	Orientation vector flag =1
<b>F,65539 =2</b>		Grid option	
7	GO	I	Grid point ID at end of orientation vector
8	UNDEF(2)	none	
10	F	I	
<b>End F,65539</b>			
11	PA	I	Pin flags for end A
12	PB	I	Pin flags for end B
13	W1A	RS	T1 component of offset vector from GA
14	W2A	RS	T2 component of offset vector from GA
15	W3A	RS	T3 component of offset vector from GA
16	W1B	RS	T1 component of offset vector from GB



Word	Name	Type	Description
17	W2B	RS	T2 component of offset vector from GB
18	W3B	RS	T3 component of offset vector from GB

**CQUAD4(2958,51,177)**

The second undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(4)	I	Grid point identification numbers of connection points
7	THETA	RS	Material property orientation angle or coordinate system ID
8	ZOFFS	RS	Offset from the surface of grid points reference plane
9	UNDEF	none	
10	TFLAG	I	Flag signifying meaning of T(4) values
11	T(4)	RS	Membrane thickness of element at grid points

**CQUAD8(4701,47,326)**

The relative thickness flag, TFLAG, has been added.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(8)	I	Grid point identification numbers of connection points
11	T(4)	RS	Membrane thickness of element at grid points

Word	Name	Type	Description
15	THETA	RS	Material property orientation angle or coordinate system ID
16	ZOFFS	RS	Offset from the surface of grid points reference plane
17	TFLAG	I	Flag signifying meaning of T(4) values

### CTRIA3(5959,59,282)

The third undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(3)	I	Grid point identification numbers of connection points
6	THETA	RS	Material property orientation angle or coordinate system ID
7	ZOFFS	RS	Offset from the surface of grid points reference plane
8	UNDEF(2)	none	
10	TFLAG	I	Flag signifying meaning of T(3) values
11	T(3)	RS	Membrane thickness of element at grid points

### CQUADP(11101,111,9014)

The second undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(17)	I	Internal indices of connection points

Word	Name	Type	Description
20	UNDEF(7)	none	
27	INORM	I	Flag for normals
28	THETA	RS	Material property orientation angle or coordinate system ID
29	ZOFFS	RS	Offset from the surface of grid points reference plane
30	UNDEF	none	
31	TFLAG	I	Flag signifying meaning of T(4) values
32	T(4)	RS	Membrane thickness of element at grid points

**CQUADR(8009,80,367)**

The second undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(4)	I	Grid point identification numbers of connection points
7	THETA	RS	Material property orientation angle or coordinate system ID
8	ZOFFS	RS	Offset from the surface of grid points reference plane
9	UNDEF	none	
10	TFLAG	I	Flag signifying meaning of T(4) values
11	T(4)	RS	Membrane thickness of element at grid points

**CTRIA6(4801,48,327)**

The relative thickness flag, TFLAG, has been added.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(6)	I	Grid point identification numbers of connection points
9	THETA	RS	Material property orientation angle or coordinate system ID
10	ZOFFS	RS	Offset from the surface of grid points reference plane
11	T(3)	RS	Membrane thickness of element at grid points
14	TFLAG	I	Flag signifying meaning of T(3) values

**CTRIAP(11301,113,9015)**

The third undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(11)	I	Internal indices of grid points
14	UNDEF(3)	none	
17	THETA	RS	Material property orientation angle or coordinate system ID
18	ZOFFS	RS	Offset from the surface of grid points reference plane
19	UNDEF(2)	none	
21	TFLAG	I	Flag signifying meaning of T(3) values
22	T(3)	RS	Membrane thickness of element at grid points

**CTRIAR(9200,92,385)**

The third undefined item in MSC.Nastran 2001 is now replaced by new item TFLAG, for relative thickness.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(3)	I	Grid point identification numbers of connection points
6	THETA	RS	Material property orientation angle or coordinate system ID
7	ZOFFS	RS	Offset from the surface of grid points reference plane
8	UNDEF(2)	none	
10	TFLAG	I	Flag signifying meaning of T(3) values
11	T(3)	RS	Membrane thickness of element at grid points

**CWELD(11701,117,559)**

This record has a totally new format.

Word	Name	Type	Description
1	EID	I	Element ID
2	PID	I	Property ID
3	GS	I	Spot weld master node ID GS
4	FORMAT(C)	I	Connection format (0=gridid)
5	GA	I	ID of GA
6	GB	I	ID of GB
7	TYPE	I	Types of upper and lower elements for FORM="GRIDID"
8	CID	I	C
9	GUPPER(8)	I	Grid IDs of the upper shell

Word	Name	Type	Description
17	GLOWER(8)	I	Grid IDs of the lower shell
25	TAVG	RS	Average shell thickness
26	UNDEF(2 )	none	
28	TMIN	RS	Minimum shell thickness

## GEOM3 - Table of Bulk Data Entry Images Related to Static and Thermal Loads

Added four new words: SORL(2) and LDIR(2) to PLOAD4 record By default for negative values of user PARAMeter POST, this record is automatically restored to its V2001 format on the resulting op2 files. See the “[Modifications to OUTPUT2 \(INPUTT2\) Transport Formats](#)” on page 570.

### PLOAD4(7209,72,299)

Added four new words: SORL(2) and LDIR(2)

Word	Name	Type	Description
1	SID	I	Load set identification number
2	EID	I	Element identification number
3	P(4)	RS	Pressures
7	G1	I	Grid point identification number at a corner of the face
8	G34	I	Grid point ID at a diagonal from G1 or CTETRA corner
9	CID	I	Coordinate system identification number
10	N(3)	RS	Components of a vector coordinate system defined by CID
13	SORL(2)	CHAR4	Load act on element SURF or LINE
15	LDIR(2)	CHAR4	Load direction

## GEOM4 - Table of Bulk Data Entry Images Related to Constraints, Degree-of-freedom Membership and Rigid Element Connectivity

Added new item, ALPHA, for thermal expansion in the RBAR, RBE1, RBE2, RBE3, RROD, and RTRPLT records. By default, for negative values of user PARAMeter POST, these records are automatically restored to their MSC.Nastran 2001 formats on the resulting op2 files. See the “[Modifications to OUTPUT2 \(INPUTT2\) Transport Formats](#)” on page 570.

**RBAR(6601,66,292)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	GA	I	Grid point A identification number
3	GB	I	Grid point B identification number
4	CNA	I	Component numbers of independent DOFs at end A
5	CNB	I	Component numbers of independent DOFs at end B
6	CMA	I	Component numbers of dependent DOFs at end A
7	CMB	I	Component numbers of dependent DOFs at end B
8	ALPHA	RS	Thermal expansion coefficient

**RBE1(6801,68,294)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	GN	I	Grid point identification number for independent DOFs
3	CN	I	Component numbers of independent DOFs
Words 2 through 3 repeat until (-2,-2) occurs			



Word	Name	Type	Description
4	GM	I	Grid point identification number for dependent DOFs
5	CM	I	Component numbers of dependent DOFs
<b>Words 4 through 5 repeat until (-1,-1) occurs</b>			
6	ALPHA	RS	Thermal expansion coefficient
7	UNDEF	none	

**RBE2(6901,69,295)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	GN	I	Grid point identification number for independent DOFs
3	CM	I	Component numbers of dependent DOFs
4	GM	I	Grid point identification number for dependent DOFs
<b>Word 4 repeats until End of Record</b>			
5	ALPHA	RS	Thermal expansion coefficient

**RBE3(7101,71,187)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	REFG	I	Reference grid point identification number
3	REFC	I	Component numbers at the reference grid point
4	WT1	RS	Weighting factor for components of motion at G

Word	Name	Type	Description
5	C	I	Component numbers
6	G	I	Grid point identification number
<b>Word 6 repeats until End of Record</b>			
<b>Words 4 through 6 repeat until End of Record</b>			
7	GM	I	Grid point identification number for dependent DOFs
8	CM	I	Component numbers of dependent DOFs
<b>Words 7 through 8 repeat until End of Record</b>			
9	ALPHA	RS	Thermal expansion coefficient

**RROD(6501,65,291)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	GA	I	Grid point A identification number
3	GB	I	Grid point B identification number
4	CMA	I	Component numbers of dependent DOFs at end A
5	CMB	I	Component numbers of dependent DOFs at end B
6	ALPHA	RS	Thermal expansion coefficient

**RTRPLT(6701,67,293)**

Added new item, ALPHA, for thermal expansion.

Word	Name	Type	Description
1	EID	I	Element identification number
2	GA	I	Grid point A identification number
3	GB	I	Grid point B identification number
4	GC	I	Grid point C identification number

Word	Name	Type	Description
5	CNA	I	Component numbers for independent DOFs at vertex A
6	CNB	I	Component numbers for independent DOFs at vertex B
7	CNC	I	Component numbers for independent DOFs at vertex C
8	UNDEF	none	
9	CMA	I	Component numbers for dependent DOFs at vertex A
10	CMB	I	Component numbers for dependent DOFs at vertex B
11	CMC	I	Component numbers for dependent DOFs at vertex C
12	ALPHA	RS	Thermal expansion coefficient

## 14.5 Removal of Old Features

The DYNRED Case Control command and Bulk Data entry are no longer documented in the *MSC.Nastran Quick Reference Guide*. Furthermore, SOLs 1 through 16 are also no longer documented. These features will also be removed from the code in a future version.

## 14.6 MSC.Nastran Error List

MSC.Nastran errors are included on the delivery CD. We have corrected 263 errors in MSC.Nastran 2004 as compared to MSC.Nastran 2001. In addition, all unresolved errors are listed. The error lists can also be obtained from the MSC Software website:

[http://www.mscsoftware.com/support/prod\\_support/nastran/errorlist/index.cfm](http://www.mscsoftware.com/support/prod_support/nastran/errorlist/index.cfm)



## APPENDIX

# A

---

# Statements, Commands, Entries, and Parameters

- New and Modified Executive Control Statements for MSC.Nastran 2004
- New and Modified Case Control Commands for MSC.Nastran 2004
- New and Modified Parameters for MSC.Nastran 2004
- New and Modified Bulk Data Entries for MSC.Nastran 2004

## A.1 New and Modified Executive Control Statements for MSC.Nastran 2004

### New Executive Control Statements for MSC.Nastran 2004

The following Executive Control statements are new for MSC.Nastran 2004. The statements are detailed in “[Executive Control Statements](#)” on page 99 of the *MSC.Nastran Quick Reference Guide*.

### New Executive Control Statement for MSC.Nastran Implicit Nonlinear (SOL 600) in MSC.Nastran 2004

SOL 600,ID

### Executive Control Statements for MSC.Nastran 2004 with Major Modifications

The following Executive Control statements have been greatly modified for MSC.Nastran 2004. The statements are included in this section as well as in “[Executive Control Statements](#)” on page 99 of the *MSC.Nastran Quick Reference Guide*.

DOMAINSOLVER

### Executive Control Statements for MSC.Nastran 2004 with Minor Modifications

The following Executive Control statements are modified for MSC.Nastran 2004. To see the complete statements, go to “[Executive Control Statements](#)” on page 99 of the *MSC.Nastran Quick Reference Guide*.

SOL



## A.2 New and Modified Case Control Commands for MSC.Nastran 2004

### New Case Control Commands for MSC.Nastran 2004

The following Case Control commands are new for MSC.Nastran 2004. The commands are detailed in “[Case Control Commands](#)” on page 155 of the *MSC.Nastran Quick Reference Guide*. Case Control commands denoted with an “#” sign are available only when executing MSC.Marc using MSC.Nastran Implicit Nonlinear (SOL 600).

A2GG	MAXMIN
ADAMSMNF	MCFRACTION
AUTOSPC	MODALKE
BCONTACT#	MODALSE
DESVAR	MODESELECT
DRSPAN	NLRESTART
EQUILIBRIUM	NSM
EXTSEOUT	POST
FLSFSEL	RCROSS
FLSPOUT	RESVEC
FLSTCNT	RIGID
K42GG	STEP
MAXMIN(DEF)	

### Case Control Commands for MSC.Nastran 2004 with Major Modifications

The following Case Control commands have been greatly modified for MSC.Nastran 2004. The commands are included in this section as well as in “[Case Control Commands](#)” on page 155 of the *MSC.Nastran Quick Reference Guide*.

B2GG	K2PP
B2PP	M2GG
ELSUM	M2PP
IC	P2G
K2GG	

# **Case Control Commands for MSC.Nastran 2004 with Minor Modifications**

The following Case Control commands are modified for MSC.Nastran 2004. To see the complete commands, go to “[Case Control Commands](#)” on page 155 of the *MSC.Nastran Quick Reference Guide*.

ACCELERATION	PARTN
ANALYSIS	SET
DISPLACEMENT	SPCFORCES
FORCE	STRAIN
MPCFORCES	STRESS
NLSTRESS	TEMPERATURE
OLOAD	VELOCITY

## A.3 New and Modified Parameters for MSC.Nastran 2004

### New Parameters for MSC.Nastran 2004

The following Parameters are new for MSC.Nastran 2004. The Parameters are detailed in “[Parameter Descriptions](#)” on page 574 of the *MSC.Nastran Quick Reference Guide*. Parameters with a “#” sign next to them are only available for MSC.Nastran Implicit Nonlinear (SOL 600). See the individual parameter for further details.

ADJMETH	MARCDIS2#	MARCSCLR#	MRTABLS1#
ARF	MARCDIS3#	MARCSINC#	MRTABLS2#
ARS	MARCDIS4#	MARCSIZ3#	MRTSHEAR#
AUTOMSET	MARCEKND#	MARCSIZ4#	NLMAX
AUTOSPRT	MARCFRIC#	MARCSIZ5#	NLMIN
BEAMBEA#	MARCGAUS#	MARCSIZ6#	NODCMP
BEIGRED	MARCIAMN#	MARCSLHT#	PARTMEM#
DBEXT	MARCLUMP	MARCSOLV#	PENFN, PH2OUT
DPHFLG	MARCMEM#	MARCSUMY#	RCONTACT#
ENFMOTN	MARCNOER#	MARCT16#	RMSINT
FZERO	MARCONLY#	MARCT19#	SHLDAMP
GYROAVG	MARCOPT#	MARCTABL#	SKINOUT
LGSTRN#	MARCOTIM#	MARCTOL#	SOLADJC
LMFACT	MARCPOS#	MARCTVL#	SPARSEDM
MRALIAS#	MARCPOST#	MARCVERS#	SPARSEDR
MALIAS03, etc.#	MARCPRN#	MARUPDAT#	SPDMRAT
MARC3D#	MARCPH#	MCONTACT#	SPDRRAT
MARCAUTO#	MARCRBAL#	MHOUBOLT#	SRCOMPS
MARCAUTD#	MARCRBAR#	MMFIL	STRESS#
MARCBUG#	MARCRBE2#	MDUMLOAD#	WRBEAMB#
MARCCENT#	MARCRBE3#	MRCONRES#	
MARCCON2#	MARCREVR#	MRMEMSUM#	
MARCCON3#	MARCRIGD#	MRRELNOD#	
MARCCPY#	MARCRUN#	MRSRING#	

## Parameters for MSC.Nastran 2004 with Major Modifications

The following Parameters are modified for MSC.Nastran 2004. The complete Parameter can be viewed in “[Parameter Descriptions](#)” on page 574 of the *MSC.Nastran Quick Reference Guide*.

ERROR

K6ROT

OSWELM

OSWPPT

STIME

## A.4 New and Modified Bulk Data Entries for MSC.Nastran 2004

### New Bulk Data Entries for MSC.Nastran 2004

The following Bulk Data entries are new for MSC.Nastran 2004. The entries are detailed in “[Bulk Data Entries](#)” on page 783 of the *MSC.Nastran Quick Reference Guide*. Entries denoted with an "#" are available only when executing MSC.Marc using MSC.Nastran Implicit Nonlinear (SOL 600). See individual entry description for further details.

BCBODY#	IPSTRAIN#	MATVE#	ROTORG
BCBOX#	ISTRESS#	MATVP#	RSPINR
BCHANGE#	MARCIN#	NLAUTO#	RSPINT
BCMATL#	MARCOUT#	NLDAMP#	RTRPLT1
BCMOVE#	MATEP#	NLSTRAT#	RVDOF
BCPARA#	MATF#	NSM	RVDOF1
BCPROP#	MATG#	NSM1	SWLDPRM
BCTABLE#	MATHE#	NSMADD	UNBALNC
BNDFIX	MATHED#	NSML	
BNDFIX1	MATORT#	NSML1	
BNDFREE	MATTEP#	NTHICK#	
BNDFREE1	MATTG#	PARAMARC#	
BSURF#	MATTHE#	RCROSS	
CONTRLT	MATTORT#	RESTART#	
DRESP3	MATTVE#	RGYRO	
GMNURB#	MATT8	RJOINT	

## Bulk Data Entries for MSC.Nastran 2004 with Major Modifications

The following Bulk Data entries have been greatly modified for MSC.Nastran 2004. The entries are included in this section as well as in “[Bulk Data Entries](#)” on page 783 of the *MSC.Nastran Quick Reference Guide*.

ACMODL	NLPARM	RTRPLT
CQUADR	PLOAD4	
CTRIAR	PWELD	
CWELD	RBAR	
DOPTPRM	RBE1	
DRESP1	RBE2	
DRESP2	RBE3	
DSCREEN	RROD	

## Bulk Data Entries for MSC.Nastran 2004 with Minor Modifications

The following Bulk Data entries have been modified for MSC.Nastran 2004. To see the complete entries, go to “[Bulk Data Entries](#)” on page 783 of the *MSC.Nastran Quick Reference Guide*.

ACSRCE	CTRIAR	PCOMP
AXIC	DCONSTR	PSHELL
BAROR	DTABLE	RADCAV
BEAMOR	DVPREL1	RLOAD1
CBAR	DVPREL2	RLOAD2
CBEAM	PBAR	SET1
CQUAD4	PBARL	TIC
CQUAD8	PBEAM	TLOAD1
CQUADR	PBEAML	TLOAD2
CTRIA3	PBUSH	VIEW3D
CTRIA6	PBUSHT	

APPENDIX

B

References

■ References

## B.1 References

1. Sung, S.H., and Nefske, D.J., **A Coupled Structural-Acoustic Finite Element Model for Vehicle Interior Noise Analysis**, Design and Production Engineering Technical Conference, Dearborn, Mich., September 11-14, 1983, The American Society of Mechanical Engineers.
2. Kinsler, Frey, Coppens, and Sanders, **Fundamentals of Acoustics**, John Wiley, 1982.
3. Nelson, R.B., **Simplified Calculation of Eigenvector Derivatives**, AIAA Journal, Vol. 4, No 9, pp. 1201-1205, 1976.
4. Coppolino, B., **Robust Modal Derivative Computation**, UAI/Nastran Document, 10/28/1998.
5. Vaidyanathan, S., **MSC.Nastran Geometry Evaluator Developer's Guide**, Jan. 1994.
6. Wilder, S., **Software Requirement Specification for Beam-Cross-Section Enhancement**, May 1995.
7. Johnson, E.H., **Innovative Uses of Synthetic Responses in Design Optimization**, The MSC World User's Conference Proceedings, May 1995, Universal City, California.
8. "Random Analysis with Coupled Excitations" on page 238 of the **MSC.Nastran Advanced Dynamic Analysis User's Guide**.
9. Kirsch, U., **On Singular Topologies in Optimum Structural Design**, Structural Optimization, Vol. 2, p.133, 1990.
10. Rodden, W.P., and Johnson, E.H., **MSC.Nastran Aeroelastic Analysis User's Guide**, Version 68, Section 8.10, MSC.Software, October 1994.
11. Tsai, **Composites Design**, 4th Ed., Think Composites, Dayton, Ohio, May 1988.
12. Hopkins, P., **Strength Ratio Calculations**, Memo, Astrium Ltd.
13. Reissner, E., **A note on variational principles in elasticity**, Int. J. Solids Struct., 1, pp. 93-95, 1965.
14. Allman, D.J., **A compatible triangle element including vertex rotations for plane elasticity problems**, Comp. Struct., 19, 1-8, 1984.
15. MacNeal, R.H., and Harder, R.L., **A proposed standard set of problems to test finite element accuracy**, Finite Elements in Analysis and Design 1, pp.3-20, 1985



16. Cook, D., **On the Allman triangle and a related quadrilateral element**, Computers & Structures 22, pp.1065-1067, 1986
17. MacNeal, R.H., and Harder, R.L., **A refined four noded membrane element with rotational degree of freedom**, comp. Struct., 18, 75-84, 1988.
18. Hughes, T.J.R., and F. Brezzi, **On drilling degrees of freedom**, Comp. Methods Appl. Mech. Eng., 72, 105-121, 1989.
19. Ibrahimbegavic, A., Taylor, R. L., and Wilson, E. L., **A robust quadrilateral membrane finite element with drilling degrees of freedom**, Int. J. Numer. Methods Eng., 30, 445-457, 1990.
20. Hoff, C.C., Harder, R.L., Campbell, G., MacNeal, R.H., and Wilson, C.T., **Analysis of shell structures using MSC/Nastran's shell elements with surface normals**, Proceedings of the 1995 MSC World Users' Conference, Paper #26, Universal City, CA, 1995
21. MacNeal, R.H., Wilson, C.T., Harder, R.L., and Hoff, C.C., **The treatment of shell normals in finite element analysis**, Finite Elements in Analysis and Design 30, pp.235-242, 1998.
22. UAI/NASTRAN 20.1 User's Reference Manual, Universal Analytics Inc, Sixth Edition, March 1999.
23. *MSC.Nastran Quick Reference Guide*, MSC.Software Corporation, Copyright 2003.
24. H.Chang, R.L.Harder, C.C.Hoff, C.T.Wilson, **Practical Finite Element Modeling of Assembled Shell Structures**, " Proceedings of the NAFEMS World Congress 2003, Orlando, FL.



# I N D E X

## MSC.Nastran Release Guide

## Numerics

3D Contact, 346

### A

ACMODL, 308  
ACMS, 411, 412, 414  
ACSRCE, 92  
ADAMSMNF, 446  
adaptive bisection, 381, 385  
adaptive time steps, 7  
adjoint load method, 25, 28  
adjoint method, 2  
ALTER, 524  
anisotropic, 346  
API, 62  
Arruda-Boyce rubber, 346  
ASET, 297  
ASET1, 297  
ASMBULK, 297  
ASSIGN statement  
    RECL, 9  
    SIZE, 9  
Augmented Lagrange Multiplier Method,  
    249  
Auto-correlation Function (AUTO), 106, 108  
automatic time stepping, 378  
auto-PSDs, 108  
AUTOSPC, 263

### B

B2GG, 552  
BCBODY, 353, 358  
BCBOX, 353

BCHANGE, 353  
BCMATL, 353  
BCMOVE, 353  
BCONTACT, 353, 357  
BCPARA, 353  
BCPROP, 353  
BCTABLE, 353, 358  
BEND, 231  
BNDFIX, 297  
BNDFREE, 297  
BNDFREE1, 297  
BSET, 297  
BSET1, 297  
BSURF, 353, 358  
buckling analysis, 387

### C

CBAR, 202, 205  
CBEAM, 202, 205  
CBUSH, 118  
CHACAB, 333  
CHEXA, 333  
Cholesky, 151, 152  
CLAN, 5  
CMETHOD, 392  
complex eigenvalue analysis, 5, 123, 155  
complex source strength, 340  
Composite Rotations, 247  
CONNECT, 61  
CONTRLT, 435, 436, 440, 443  
convection, 440  
COPYR, 348  
CQUAD4, 235, 292, 333  
crack, 347  
Craig-Bampton, 447

creep, 347, 363  
 cross-correlation functions, 4  
 cross-power spectral density, 4  
 cross-PSDs, 108  
 CSET, 297  
 CSET1, 297  
 CTRIA3, 292  
 CWELD, 279, 280

## D

DCMP, 421  
 DECOMP, 29, 30  
 decompositions, 158  
 Degree-of-Freedom domain decomposition, 400  
 degrees-of-freedom (DOF) domain parallel  
   Lanczos method, 2  
 DELAY field, 3, 92  
 DESVAR, 59  
 DFREQ, 30  
 differential stiffness, 387, 388  
 direct transient analysis, 97  
 Distributed Memory Parallel  
   DMP, 404  
 distributed memory parallel processing  
   DMP, 31, 35  
 DMIGPCH, 298  
 DMP, 400, 411, 412, 415  
 DNDFIX1, 297  
 DNODE, 59  
 DOF based Domain Decomposition, 403  
 Domain Decomposition, 7  
 domain decomposition, 347  
 DOMAINSOLVER, 401, 405, 406, 409, 415, 417, 419  
   ACMS, 407  
   FREQ, 407  
   MODES, 33, 34, 407  
   STAT, 407

DOPTPRM, 3, 51, 53  
   P2CALL, 51  
   P2CBL, 51  
   P2CC, 51  
   P2CDDV, 51  
   P2CM, 51  
   P2CP, 51  
   P2CR, 51  
   P2RSET, 51  
 DPHASE field, 3, 92  
 DRESP1, 14, 59  
 DRESP2, 40, 55, 59  
 DRESP3, 3, 56, 57, 59, 61, 62, 67, 70  
 DTABLE, 59  
 DVCREL1, 59  
 DVCREL2, 59  
 DVMREL1, 59  
 DVMREL2, 59  
 DVPREL1, 59, 235  
 DVPREL2, 59, 235

## E

EDE, 527  
 EIDSUM, 214  
 EIGC, 392  
 eigenvector sensitivity, 2, 14  
 EKE, 527  
 element coordinate system, 203  
 ELEMID, 279  
 ELPAT, 279  
 ELSUM, 213, 214, 217  
 enforced motion, 104  
 EQUILIBRIUM, 218, 219, 220  
 equilibrium check, 5  
 ESE, 363  
 Euler angles, 246  
 EXTBULK, 298  
 EXTDR, 296  
 EXTDROUT, 296  
 EXTOUT, 296  
 EXTSEOUT, 296, 297

## F

F06, 351  
 f06, 348  
 F11, 351  
 failure criteria, 225, 227, 228  
     HILL, 225  
     HOFFMAN, 227  
     TSAI-Wu, 228  
 Failure Index (FI), 225  
 FBS, 421  
 FBS (forward-backward substitutions), 30  
 Flexible-Flexible surfaces, 346  
 Flexible-Rigid surfaces, 346  
 FLSFSEL, 333  
 FLSPOUT, 334  
 FLSTCNT, 333  
 fluid load participation, 321  
 fluid mode participation, 320  
 fluid-structure panel participation, 321  
 fluid-structure panel-grid participation, 321  
 FLUTTER, 33  
 foam, 346  
 follower force, 380  
 follower stiffness, 391  
 forward-backward substitutions (FBS), 158  
 Free Field Format, 555  
 frequency response, 5, 86, 123  
 FRRD1 module, 31  
 FSD, 235

## G

Gasket, 347  
 geometric domain decomposition, 403  
 Gimbal angles, 246  
 GIV, 156  
 GMNURB, 353  
 GMODES, 405  
 GPFORCE, 39, 363  
 gray-diffuse, 428  
 GRIDID, 279

## H

heat exchanger, 440  
 HFREQ parameter, 93  
 HFREQFL parameter, 93  
 Hierarchic Distributed Parallel Lanczos  
     Method, 7, 404  
 Hierarchic Method, 401  
 HOU, 156

## I

IC, 97  
 IC(MODAL), 98  
 IC(PHYSICAL), 98  
 IC(STATSUB), 98  
 IC(STATSUB,DIFFK), 98  
 Implicit Nonlinear Analysis, 346  
 INCLUDE, 524  
 initial condition, 4, 97  
 IPSTRAIN, 353  
 ISHELL, 524  
 ISTRESS, 353

## K

K2GG, 552  
 K2PP, 553  
 K42GG, 553  
 K6ROT, 263, 292

## L

Lagrange Elimination Method, 244  
 Lagrange elimination method, 251  
 Lagrange rigid element, 243  
 Lanczos method  
     (LAN), 145  
 Left Rotation, 247  
 LFREQ parameter, 93  
 LFREQFL parameters, 93  
 Linear Elimination Method, 249  
 LMODES parameter, 93  
 LMODESFL parameters, 93

Log-Log, 112  
 LOOPID, 385

## M

M2GG, 552  
 M2PP, 553  
 MacNeal/Harder test, 268  
 MALTER, 524  
 MARCIN, 353  
 MARCOUT, 353  
 massless mechanism, 145  
 MAT2, 118, 231  
 MAT8, 118  
 MATEP, 353  
 MATG, 353  
 MATHE, 353  
 MATHED, 353  
 MATORT, 353  
 MATT, 353  
 MATTE, 354  
 MATTEP, 354  
 MATTG, 354  
 MATTORT, 354  
 MATTVE, 354  
 MATV, 354  
 MATVP, 354  
 Max/Min/RMS, 134  
 maximum stress (strain), 229  
 MAXMIN(DEF), 134  
 MAXRATIO, 147  
 MCEIG, 33  
 MCFRACTION, 122, 124, 126, 127  
 MECHFIL, 147  
 MECHFIX, 146  
 MECHPRT, 146  
 MEM, 231  
 Message Passing Interface  
   MPI, 401, 405  
 METHOD, 392  
 METHOD(FLUID), 333  
 METHOD(STRUCT), 333  
 MFREQ, 29, 31, 33  
 mfreqea, 114  
 MID1, 118

MID2, 118  
 MID3, 118  
 MID4, 118  
 MMFIL, 147  
 modal contribution fraction, 122, 126  
 modal kinetic energy, 4, 114  
 modal participation factor, 4, 122  
 modal strain energy, 4  
 modal transient analysis, 97  
 MODALKE, 114, 115  
 MODALSE, 114, 115  
 MODESELECT, 3  
 Modified Method of Feasible Directions, 87  
 MSC.ADAMS, 399, 448  
 MSC.Marc, 347, 348  
 mtranea, 114  
 MTRANS, 33

## N

nast2004.kwds, 521, 522  
 nast2004.params, 521  
 nast2004.rcf, 521  
 nast2004rc, 521  
 NASTRAN FASTDFR, 157  
 NDDL  
   Nastran Data Definition Language, 528  
 Nelson's method, 14, 15  
 Newton's method, 390  
 Newton-Raphson method, 245  
 NLAUTO, 354  
 NLDAMP, 354  
 NLMAX, 147  
 NLMIN, 148  
 NLPCI, 363  
 NLRGAP, 381  
 NLSTRAT, 354  
 NOLIN, 378, 381  
 NOLIN1, 381  
 NOLIN2, 381  
 NOLIN3, 381  
 NOLIN4, 381  
 nonlinear buckling analysis, 7  
 NSM, 207, 208  
 NSM1, 207, 208

NSMADD, 207, 208  
 NSML, 207, 208  
 NSML1, 207, 208  
 NTHICK, 354  
 Number of Zero Crossing (N0), 106  
 NVH optimization, 3, 48

## O

offset vectors, 203  
 offsets, 5, 202  
 OFREQ, 86  
 Ogden rubber, 346  
 OP2, 351  
 op2, 348  
 orientation vector, 204  
 orthotropic, 346  
 OUTR, 348

## P

P2G, 552  
 PANEL, 334

PARAM, 230  
 ADJMETH, 47, 48  
 AUTOADJ, 26  
 BUCKLE, 390  
 DPHFLG, 19, 20  
 EIGD, 14  
 ENFMOTN, 104  
 FKSVMFAC, 373, 391  
 FOLLOWK, 373  
 GRDPNT, 460, 463  
 LANGLE, 253, 372  
 LGDISP, 373  
 LGSTRN, 354  
 LMFACT, 253  
 MALIASij, 356  
 MARCASUM, 355  
 MARCCONi, 355  
 MARCDILT, 355  
 MARCDISi, 355  
 MARCEKND, 355  
 MARCGAUS, 355  
 MARCIAMN, 355  
 MARCMEM, 354  
 MARCOPT, 354  
 MARCPOS, 354  
 MARCPOST, 355  
 MARCRBAL, 356  
 MARCRBAR, 355  
 MARCRBE2, 355  
 MARCRBE3, 355  
 MARCSHLT, 355  
 MARCSINC, 355  
 MARCT16, 355  
 MARCT19, 355  
 MARCVERS, 355  
 MARSIZi, 354  
 MARSUMY, 355  
 MARUPDAT, 355  
 MAXLP, 373  
 MAXRATIO, 147  
 MECHFIL, 147  
 MECHFIX, 145, 146  
 MECHPRT, 146  
 MRALIAS, 356  
 MRCONRES, 355

MRSPRING, 355  
 NDAMP, 380  
 NLAYERS, 373  
 NLTOL, 373  
 NMLOOP, 446  
 NOCOMPS, 234  
 ORIGID, 253  
 PENFN, 253  
 PH2OUT, 373  
 QRMETH, 272  
 RMSINT, 112  
 SEKD, 385  
 SHLDAMP, 119, 120  
 SOLADJC, 28, 29  
 SPARSEDM, 47  
 SPARSEDR, 47  
 SRCOMPS, YES/NO, 230  
 STIME, 96  
 STRESS, 354  
 WTMASS, 460, 463  
 PARAMARC, 354  
 parameter, 521, 526  
 parameter, see also PARAM  
 PARTFAC, 122  
 PARTOPT, 33, 34, 405  
 PARTPAT, 279  
 PBEAM, 75, 76  
 PBEAML, 74, 75, 76  
 PBUSH, 120  
 PCH, 351  
 pch, 348  
 PCOMP, 231, 235  
 PIDSUM, 214, 217  
 PLOTTEL, 298  
 Power Spectral Density Function  
     PSDF, 106  
 PSHELL, 118, 231, 235  
 punch, 527, 530

## Q

QBDY3, 435  
 QSET, 297  
 QUAD4, 263, 267  
 QUAD8, 267

QUADR, 263, 267  
 quaternion, 246  
 QVECT, 435  
 QVOL, 435  
 QZHESS method, 5, 155

## R

radiation view factors, 428  
 RANDOM, 108, 530  
 random analysis, 106  
 random response, 4  
 RANDPS, 334  
 Rayleigh Quotient Approximation, 86  
 RBAR1, 254, 255  
 RCROSS, 108  
 residual structure, 126  
 residual vector, 4, 100  
 RESTART, 354  
 Restart, 369  
 restarts, 4  
 RFORCE, 363  
 Right Rotation, 247  
 RIGID, 253, 254  
 RJOINT, 254, 256  
 RLOAD1, 92  
 RLOAD2, 92  
 Rodrigues parameters, 246  
 Root Mean Square  
     RMS, 106, 112  
 RPRINT, 113  
 RPUNCH, 113  
 RTRPL1, 254  
 RTRPLT1, 255  
 rubber, 347  
 RVDOF, 103  
 RVDOF1, 103

## S

SDAMPING, 104  
 SEBNDRY, 297  
 SEBULK, 297  
 SECONCT, 297  
 sensor, 441



Sequential Quadratic Programming  
     SQP, 87  
 SEUSETi,U6, 103  
 Shared Memory Parallel  
     SMP, 400  
 SMCORE, 231, 235  
 SMEAR, 231, 235  
 soil, 347  
 SOL 400, 257, 362  
 SOL 600, 346, 347  
 sparse Cholesky factor, 156  
 sparse data recovery  
     SDR, 46  
 SSALTER, 342  
 stability derivatives, 493, 494  
 STEP, 362, 363  
 strain energy, 114  
 strength ratio, 242  
 structural damping, 4, 118  
 structure mode participation, 320  
 SUBCASE, 362, 363  
 Subspace Iteration Method, 14, 15, 19  
 superelements, 18, 27, 126  
 symmetric, 235  
 system cell  
     210, 528  
     366, 372  
     401, 372

## T

T16, 352  
 T19, 352  
 TABRND1, 334  
 Test/Analysis Correlation, 14  
 TFLAG, 221  
 thermal control, 435  
 thermostat, 436, 441  
 thermostat control, 435, 436  
 TIC, 97  
 TLOAD1, 92, 380  
 TLOAD2, 92  
 TRIAR, 263

## U

USETi,U6, 103

## V

VECPLOT, 218, 219  
 VIEW factor shadowing, 424  
 view factor shadowing, 424  
 virtual mass (MFLUID), 413  
 Virtual Mass (VM), 342  
 Visco-elastic, 347  
 volumetric heat source, 436

## X

XDB, 351  
 xdb, 348  
 XYTRAN, 527

## Z

ZONA51, 492, 493