

NX Nastran User's Guide

Contents

Proprietary & Restricted Rights Notice	15
Overview of NX Nastran	1-1
Overview and General Capabilities of NX Nastran	1-2
Modular Structure and DMAP	1-2
Pre-defined Solution Sequences	1-3
Using DMAP for Programmability	1-3
Overview of the NX Nastran Analysis Process	1-3
Creating the Input File	1-3
Submitting the Input File for Analysis	1-4
Reviewing the Results	1-4
Structure and Syntax for Input Data	2-1
Overview of Specifying Input Data	2-2
General Input File Syntax Rules	2-2
Understanding Types of Input Data	2-2
Using Consistent Unit Systems	2-2
Specifying Valid Input File Characters	2-4
Understanding NX Nastran Input Records	2-5
Understanding Free Field Format	2-6
Understanding Small Field Format	2-7
Understanding Large Field Format	2-7
Specifying Continuations for Bulk Data	2-8
Manually Specifying a Continuation with Identifiers	2-8
Automatically Specifying a Continuation	2-9
Replicating Bulk Data Entries	2-10
Replicating Continuation Entries	2-11
Replication with MSGMESH	2-11
Inserting External Files with INCLUDE	2-13
Using INCLUDE in Different Input File Sections	2-13
INCLUDE Statement Limitations	2-13
Including Comments in the Input File	2-14
Understanding the Input Data File	3-1
Overview of the NX Nastran Input File Sections	3-2
Example: Using NX Nastran Input Files to Model a Truss	3-2
The NASTRAN Statement (Optional)	3-4
File Management Section (Optional)	3-5
Executive Control Section (Required)	3-5
Executive Control Format	3-6
Order Dependence for Certain DMAP-related Statements	3-6
Commonly Used Executive Control Statements	3-6

Case Control Section (Required)	3-9
Case Control Format	3-9
Using the SET Command to Limit Output	3-14
Bulk Data Section	3-16
Bulk Data Section Format Rules	3-17
Sorting Bulk Data	3-17
Storing Bulk Data in the Database for Restarts	3-17
Understanding the Relationship Between Bulk Data Entries and Case Control	
Commands	3-18
Using Parameters	3-18
Parameters in the Superelement Solution Sequences	3-19
NX Nastran Output Files	4-1
Overview of the NX Nastran Output Files	4-2
Understanding the .f06 File	4-3
Controlling Printed Output	4-3
Understanding the General Analysis Output	4-6
Understanding Analysis Type Specific Output	4-7
Understanding the .f04 File	4-10
General Information	4-11
Summary of Physical File Information	4-11
Memory Allocation Map	4-12
Day Log	4-13
Understanding the I/O MB Column	4-13
User Information Messages 4157 and 6439	4-14
Memory and Disk Usage Statistics	4-15
Database Usage Statistics	4-15
Summary of Physical File I/O Activity	4-16
Other Optional Diagnostic Information	4-17
Comparing Different F04 Files with F04REPR	4-18
Understanding the .log File	4-18
Understanding the .plt and .pch Files	4-19
Output Listing Examples	4-19
Understanding the .DBALL and .MASTER Files	4-43
Specifying Whether to Save .DBALL and .MASTER Files	4-43
Manually Deleting .DBALL and .MASTER Files	4-43
Understanding the .xdb and .op2 Files	4-44
Defining Grid Points and Scalar Points	5-1
Overview of Grid Points and Scalar Points	5-2
Defining Grid Points	5-2
Format of the GRID Entry	5-3
Understanding the GRDSET Entry	5-7
Specifying Grid Point Properties	5-8
Defining Mass, Damping, and Stiffness through Direct Input Matrices	5-8
Defining Mass through the Concentrated Mass Element	5-8
Defining Temperatures at Grid Points	5-9
Assigning a Singularity Normal to a Degree-of-Freedom	5-9
Understanding Scalar Points	5-9

Comparing Grid Points to Scalar Points	5-9
Defining Scalar Points	5-10
Using Scalar Points as Extra Points for Dynamics	5-10
Scalar Point Output Format	5-10
Understanding Coordinate Systems	6-1
Overview of Coordinate Systems in NX Nastran	6-2
Understanding the Basic Coordinate System	6-2
ID of the Basic Coordinate System	6-2
Comparing the Basic and Global Coordinate Systems	6-2
Defining a Local Coordinate System	6-4
Bulk Data Entries for Defining Coordinate Systems	6-5
Defining a Rectangular Coordinate System	6-7
Defining Cylindrical Coordinate Systems	6-10
Example: Using Cylindrical and Spherical Coordinate Systems	6-16
Understanding the Element and Material Coordinate System	6-17
Special Considerations for Grid Points on the Polar Axis	6-18
Defining Element Offsets	6-18
Other Special Coordinate System Entries	6-18
CORD3G	6-18
GMCORD	6-18
Material Properties	7-1
Introduction	7-2
Element, Property, and Material Bulk Entries	7-5
Material Properties for Linear Structural Analysis	7-14
Isotropic Materials	7-15
Orthotropic Materials	7-19
Anisotropic Materials	7-30
Material Properties for Nonlinear Analysis	7-39
Understanding MATG	7-40
Understanding MATHP	7-40
Material Properties for Heat Transfer	7-41
Understanding MAT4 and MATT4	7-42
Understanding MAT5 and MATT5	7-43
Understanding RADM and RADMT	7-43
Material Properties for Fluid Analysis	7-43
Understanding MAT10	7-44
Understanding MFLUID	7-44
Applied Loads	8-1
Overview of Load Sets	8-2
Defining and Combining Subcases for Loading	8-2
Example: Creating Two Load Subcases for the Truss	8-2
Using REPCASE	8-4
Using SUBCOM to Combine Subcases	8-5
Defining Static Loads	8-7
Point Loads	8-8

FORCE, FORCE1, FORCE2	8-8
Distributed Loads	8-14
PLOAD	8-15
PLOAD1	8-16
PLOAD2	8-23
PLOAD4	8-24
PRESAX, PLOADE1, and PLOADX1	8-29
Inertia Loads (Acceleration Loads)	8-29
Using GRAV	8-30
Using RFORCE	8-33
Defining Mass in Your Model	8-35
Thermal Loads	8-38
Defining Temperatures for Grid Points	8-39
Defining Temperatures for Elements	8-39
Understanding Coefficient of Thermal Expansion (CTE)	8-39
Rigid Element Thermal Strain	8-42
Using Temperature Loads with Subcases	8-42
Enforced Element Deformation Loads	8-48
DEFORM Example	8-49
Loads Due to Enforced Motion	8-53
Statics	8-53
Dynamics	8-54
Using Residual Vectors (Modal Augmentation) in Modal Dynamic Response Analysis	8-55
Frequency-dependent Loads	8-56
Time-dependent Loads	8-57
Using Static Loading Data in Dynamic Analysis	8-59
Defining Loads for Cyclic Symmetry Analysis	8-60
Combining Loads	8-60
Using LOAD	8-61
Bolt Preload	8-65
Bolt Preload Procedure	8-66
Bolt Preload Example — Line Element Model	8-69
Bolt Preload Formulation	8-73
External Force Fields	8-77
EFPOINTS and EFFORCE Matrices	8-77
NX Nastran Input Requirements	8-78
Using the EFLOAD Case Control Command	8-81
Creating an EFPOINTS Matrix	8-82
Creating an EFFORCE Matrix	8-82
Linear Static and Normal Modes Example	8-83
Constraints	9-1
Introduction to Constraints	9-2
Single-point Constraints	9-2
Defining Single point Constraints in NX Nastran	9-3
Using GRID Entries	9-4
Using SPC and SPC1	9-5
Automatically Applying Single-point Constraints	9-7
Grid Point Singularity Processor	9-8

Understanding the Grid Point Singularity Table	9-8
Enforced Displacements at Grid Points (SPCD, SPC)	9-11
Multipoint Constraints	9-14
Uses of Multi-point Constraints	9-14
Defining Multi-point Constraints in NX Nastran	9-15
Rules for Using Multi-point Constraints	9-18
Rigid Body Supports	9-19
Linear Contact Using Constraints	9-21
Introduction	9-21
Input	9-21
Output	9-22
Limitations	9-22
EXAMPLE 1(cd_0.dat)	9-23
EXAMPLE 2(cd_1.dat)	9-26
Understanding Sets and Matrix Operations	10-1
Overview of Sets and Matrix Operations	10-2
Understanding the Global Displacement Set and General Solution Process	10-2
Understanding Basic Partitioning Operations	10-4
Static Condensation (Guyan Reduction)	10-5
Using Static Condensation	10-6
Sets for Dynamic Reduction	10-13
Sets for Aerodynamics	10-14
Output Selection Via Set Specification	10-14
Direct Matrix Input	11-1
Overview of Direct Matrix Input	11-2
Using DMIG	11-3
DMIG Bulk Data User Interface	11-3
DMIG Case Control User Interface	11-5
DMIG Examples	11-6
Stiffness Matrix Using DMIG	11-6
Mass Matrix Using DMIG	11-12
Viscous Damping Matrix Using DMIG	11-12
Structural Damping Matrix Using DMIG	11-13
Load Matrix Using DMIG	11-13
Element Data Recovery Resolved at Grid Points	12-1
Introduction to Element Data Recovery	12-2
Stress Recovery at Grid Points	12-3
Description of Method	12-3
Topological Method – Plate Elements and Solid Elements	12-4
Geometric Method – Surface Elements	12-7
Treatment of Exception Points – “BRANCHing”	12-9
User Information	12-11
Grid Point Stress Output Description	12-12
General Remarks	12-13
Mesh Stress Discontinuities at Grid Points	12-13

Error Estimates for Grid Point Stress Data	12-13
Element and Grid Point Stress Discontinuity Output Requests	12-15
Discussion of Error Measures	12-15
Grid Point Stresses and Mesh Stress Discontinuities	12-16
Solution Sequences	13-1
Understanding Solution Sequences	13-2
Obtaining DMAP Listings for Solution Sequences	13-4
Using Alters to Modify Solution Sequences	13-5
Understanding Scheduled and Unscheduled Exits	13-5
Restart Procedures	13-5
Data Processing and Matrix Operations by Functional Module	13-6
Geometry Processing in SubDMAP PHASEO	13-7
Matrix Assembly Operations in SubDMAP SEMG	13-9
Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB	13-11
Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB	13-14
Constraint and Mechanism Problem Identification in SubDMAP SEKR	13-16
Static Condensation in SubDMAPs SEKR and SEMR2	13-19
Static Solutions in SubDMAP SEKRRS	13-22
Static and Dynamic Load Generation	13-26
Dynamic Reduction and Component Mode Synthesis in SubDMAP SEMR3	13-30
Formulation of Dynamic Equations in SubDMAP GMA	13-35
Data Recovery Operations in SubDMAP SEDISP	13-39
Data Recovery Operations in SubDMAP SEDRCVR	13-46
Summary of Solution Sequence Operations	13-47
Static Solutions	13-47
Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS	13-55
Complex Eigenvalue Analysis in SubDMAP CEIGRS	13-69
Frequency Response and Random Analysis in SubDMAP FREQRS	13-69
Transient Response in SubDMAPs SEDTRAN and SEMTRAN	13-70
Buckling Analysis in SubDMAP MODERS	13-70
Superelement Analysis	13-72
Nonlinear Heat Transfer Analysis	13-84
Modeling Guidelines	14-1
Introduction to Modeling Guidelines	14-2
Choosing the Right Element	14-2
General Guidelines for Element Selection	14-2
Zero-dimensional Elements	14-3
One-dimensional Elements	14-3
Two-dimensional Elements	14-3
Three-dimensional Elements	14-7
R-type Elements	14-7
Determining Mesh Density	14-7
h-elements	14-14
p-elements	14-14
Creating Mesh Transitions	14-15
Transition from a Coarse Mesh to a Fine Mesh	14-15
Mesh Transition Between Dissimilar Element Types	14-25

Shell-to-solid Transition Element (RSSCON)	14-27
Reviewing Grid Point Stresses	14-41
Defining Consistent Loading	14-43
Symmetry	14-46
User Interface	14-49
Model Verification	15-1
Introduction to Model Verification	15-2
Preprocessor Checks	15-2
Creating a Shrink Plot of the Elements	15-2
Hidden Line/Shaded Plots	15-3
Free Edge/Face	15-4
Zipper Effect	15-6
CBAR/CBEAM Orientation and Offset Check	15-7
Duplicate Grid Points/Elements	15-7
Properties/Material Plots	15-7
Using Consistent Normals	15-7
Strain Energy Output	15-10
Diagnostic Tools	15-15
Element Summary Output	15-16
Element Geometry Checks	15-17
Weight Center of Gravity and Moment of Inertia Check	15-31
Mechanisms and Singularities	15-33
Applied Loads Check	15-35
Verifying Reaction Loads	15-37
1g Load	15-38
Output of Maximum, Minimum Grid Point-based Responses	15-39
Performing an Unconstrained Equilibrium Check	15-43
Automatic Identification of Unintentional Grounding	15-44
Thermal Equilibrium Check	15-45
Grid Point Force Balance	15-46
Stress Error Estimators	15-51
Grid Point Stress Discontinuities	15-53
Element Stress Discontinuities	15-57
Discussion of Error Measures	15-59
User Interface	15-59
Postprocessor Checks	15-61
Examining Displacements	15-61
Examining Stress Contours	15-62
Creating Grid Point Stress and Stress Discontinuity Plots	15-66
Viewing Element Strain Energy Plots	15-66
Restarts	16-1
Introduction to Restarts	16-2
Types of Restarts	16-3
Understanding Restart Terminology	16-3
Read-write Restarts	16-4
NASTRAN Statement Section	16-4
File Management Section (FMS)	16-4

Executive Control Section	16-5
Case Control Section	16-5
Bulk Data Section	16-5
Read-only Restarts	16-10
Mini-database Restarts	16-11
Determining the Version for a Restart	16-12
Database Concepts	17-1
Introduction to Database Concepts	17-2
Understanding DBsets	17-3
Delivery Database	17-5
Deleting DBsets	17-5
Database Autoassignment	17-6
Database Project and Version	17-6
Migrating Databases	17-7
Database Archival, Compression, and Transfer	17-7
Database Archival and Retrieval	17-8
Database Compression	17-9
Transferring Databases Across Different Platforms	17-9
Using the File Management Section with Databases	17-10
Using the INIT Statement	17-10
Using the ASSIGN Statement	17-11
Using the EXPAND Statement	17-13
Using the RESTART Statement	17-14
Using the DBCLEAN Statement	17-15
Using the DBDIR Statement	17-15
Using the INCLUDE Statement	17-16
Guidelines for Large Problems	17-16
Example I	17-17
Example J	17-17
Example K	17-18
Example L	17-19
BUFSIZE	17-19
Inertia Relief in Linear Static Analysis	18-1
Introduction to Inertia Relief	18-2
Description of Inertia Relief Using PARAM,INREL,-1	18-2
Implementation of Inertia Relief Using PARAM,INREL,-1	18-3
Automatic Inertia Relief	18-15
Limitations for the Automatic Inertia Relief Method	18-17
Contact	19-1
Contact Regions	19-2
Contact Region Parameters	19-3
Contact Pairs	19-4
Contact Control Parameters	19-6
Penalty Factor Adjustment and Contact Stiffness	19-8
Contact Output Request	19-11

Contact Stiffness Output	19-12
Contact Preview Output	19-12
Including CGAP Elements in a Contact Solution	19-14
Contact Conditions in Dynamic Solutions (SOLs 103, 111 and 112)	19-15
Contact Conditions in Linear Buckling (SOL 105)	19-16
Non-converged Contact Option	19-16
Contact Conditions with the Element Iterative Solver	19-17
Contact Conditions with Inertia Relief	19-17
Contact with Static Condensation	19-18
SOL 101 Contact Algorithm	19-19
Kinematic Equations	19-19
Normal Contact Constraints	19-20
Coulomb Friction Contact Constraints	19-21
Finite Element Kinematic Equations	19-21
Global Solution Strategy	19-23
Summary of Significant Aspects of Contact	19-24
Gluing Elements	20-1
Glue Regions	20-4
Defining and Selecting Glue Pairs	20-5
Glue Control Parameters	20-7
Glue Conditions in Acoustics Analysis	20-10
Glue Output Request	20-11
Glue stiffness output	20-11
Glue preview	20-12
Edge-to-Surface Glue Stiffness	20-13
Plotting	21-1
Overview of Plotting in NX Nastran	21-2
X-Y Plots	21-2
Flutter Analysis Plots	21-2
Superelement Plotting	21-2
Post Processors	21-7
Linear Buckling	22-1
Introduction to Linear Buckling	22-2
Finite Element Approach	22-2
Eigenvalue Extraction Methods	22-8
Enhanced Inverse Power Method (SINV)	22-8
Lanczos Method	22-8
Comparison of Methods	22-8
User Interface	22-9
Linear Buckling Assumptions and Limitations	22-10
Buckling Examples	22-11
Example 1 – Classical Euler Beam Buckling	22-11
Example 2 – Lateral Buckling	22-17
Example 3 – Planar Frame Buckling	22-22
Example 4 – Buckling of a Stiffened Panel with Transverse Shear Flexibility	22-25

Example 5 – Buckling of a Cylinder Under Uniform Axial Load	22-29
Theoretical Results for Example 5	22-33
Example 6 – Multiple Buckling Analyses in a Single Run	22-34
Follower Stiffness	23-1
Introduction	23-2
Benefits	23-2
Input	23-3
Output	23-3
Linear Dynamic Analysis with Follower Stiffness (SOLs 108, 109, 111, and 112)	23-3
Laminates	24-1
Overview of Laminated Composite Materials	24-2
Modeling Laminates in NX Nastran	24-3
Understanding Classical Lamination Theory for Shell Composites (PCOMP,PCOMPG)	24-8
Laminate Failure Indices and Strength Ratios	24-22
Failure Theories for Shell Composites (PCOMP, PCOMPG)	24-22
Failure Theories for Solid Composites (PCOMPS)	24-27
Bonding Material Failure Indices	24-32
Ply Stress and Strain Calculation for Shell Composites (PCOMP,PCOMPG)	24-32
Calculating Interlaminar Shear Stress and Strains on Shell Composites (PCOMP,PCOMPG)	24-33
Examples: Defining Laminate Material Properties	24-34
Example: Using PCOMP to Model a Honeycomb Plate	24-34
Example: Using PSHELL to Model a Honeycomb Section	24-41
Fluid-Structure Interaction	25-1
Overview of Fluid-Structure Interaction	25-2
Performing a Coupled Acoustic-Structural Analysis	25-2
Defining the Fluid	25-3
Acoustic Loading	25-4
Single-Point Constraints	25-5
Acoustic Output Requests	25-5
Superelement Analysis for Acoustic-Structure Analysis	25-7
Normal Modes Analysis for Acoustic-Structure Models	25-7
Acoustic-Structural Coupling Punch Output	25-8
SFE AKUSMOD Acoustic-Structural Coupling Interface	25-8
Acoustic-Structure Coupling	25-8
Acoustic Example	25-11
NX Nastran Acoustic Equations	25-16
Axisymmetric Acoustic Cavity Modeling	25-28
Assumptions and Limitations	25-29
Cyclic Symmetry	26-1
Understanding Cyclic Symmetry	26-2
Supported Solution Sequences	26-2
Comparing Cyclic Symmetry and Traditional Reflective Symmetry Techniques	26-2
Understanding Types of Symmetry	26-4
Cyclic Symmetry Theory	26-7

Symmetrical Components	26-7
Understanding Rotational Symmetry	26-8
Dihedral Symmetry	26-23
Reflective Symmetry	26-32
Axisymmetry	26-37
Using Cyclic Symmetry	26-39
Overview of Performing a Cyclic Symmetry Analysis	26-39
Bulk Data Entries Specific to Cyclic Symmetry	26-42
Case Control Commands Specific to Cyclic Symmetry	26-49
Performing Axisymmetric Analysis	26-53
Solving Axisymmetric Problems	26-54
Axisymmetric Example	26-55
p-Elements	27-1
Introduction to p-Elements	27-2
p-Elements in NX Nastran	27-2
p-Version Capabilities	27-3
Solution Sequences Supported	27-3
Geometry	27-3
Elements	27-4
Materials	27-4
Defining p-Element Loads	27-5
Line and Pressure Loads in p-Element Analysis	27-5
Thermal Loads	27-5
Gravity Loads	27-5
Pressure Loads	27-5
Point Loads	27-5
Defining p-Element Constraints	27-5
Boundary Conditions	27-6
Resolution of Conflicting Zero Constraints	27-6
p-Version Adaptivity	27-7
Data Recovery	27-8
p-Element Example Problems	27-10
Square Plate with a Circular Hole	27-11
Spherical Structure Under Internal Pressure	27-16
Motor Blade Normal Modes	27-21
Torsion of Elliptical Bar	27-23
Circular Solid Model	27-35
Clamped Rhombic Plate	27-47
Helix Modeled with CBEAM p-Elements	27-49
Plate with Stiffener	27-51
Solid p-Elements in Linear Dynamic Solution Sequences	27-54
User Interface in NX Nastran	27-54
Limitations and Recommendations	27-54
p-Element Thermal Analysis	27-56
Introduction	27-56
Geometry	27-56
Elements	27-56
Materials	27-57

Loads and Boundary Conditions 27-57

p-Version Adaptivity 27-57

Data Recovery 27-57

Thermal Stress Analysis 27-57

Limitations and Recommendations 27-57

Proprietary & Restricted Rights Notice

© 2014 Siemens Product Lifecycle Management Software Inc. All Rights Reserved.

This software and related documentation are proprietary to Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. NX is a trademark or registered trademark of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries.

NASTRAN is a registered trademark of the National Aeronautics and Space Administration. NX Nastran is an enhanced proprietary version developed and maintained by Siemens Product Lifecycle Management Software Inc.

MSC is a registered trademark of MSC.Software Corporation. MSC.Nastran and MSC.Patran are trademarks of MSC.Software Corporation.

All other trademarks are the property of their respective owners.

TAUCS Copyright and License

TAUCS Version 2.0, November 29, 2001. Copyright (c) 2001, 2002, 2003 by Sivan Toledo, Tel-Aviv University, stoledo@tau.ac.il. All Rights Reserved.

TAUCS License:

Your use or distribution of TAUCS or any derivative code implies that you agree to this License.

THIS MATERIAL IS PROVIDED AS IS, WITH ABSOLUTELY NO WARRANTY EXPRESSED OR IMPLIED. ANY USE IS AT YOUR OWN RISK.

Permission is hereby granted to use or copy this program, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any derivative code must cite the Copyright, this License, the Availability note, and "Used by permission." If this code or any derivative code is accessible from within MATLAB, then typing "help taucs" must cite the Copyright, and "type taucs" must also cite this License and the Availability note. Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. This software is provided to you free of charge.

Availability (TAUCS)

As of version 2.1, we distribute the code in 4 formats: zip and tarred-gzipped (tgz), with or without binaries for external libraries. The bundled external libraries should allow you to build the test programs on Linux, Windows, and MacOS X without installing additional software. We recommend that you download the full distributions, and then perhaps replace the bundled libraries by higher performance ones (e.g., with a BLAS library that is specifically optimized for your machine). If you want to conserve bandwidth and you want to install the required libraries yourself, download the lean distributions. The zip and tgz files are identical, except that on Linux, Unix, and MacOS, unpacking the tgz file ensures that the configure script is marked as executable (unpack with `tar xzvpf`), otherwise you will have to change its permissions manually.

Chapter 1: Overview of NX Nastran

- *Overview and General Capabilities of NX Nastran*
- *Overview of the NX Nastran Analysis Process*

1.1 Overview and General Capabilities of NX Nastran

NX Nastran is a general purpose finite element analysis computer program that lets you solve a wide range of engineering problems. NX Nastran is written primarily in FORTRAN and is optimized to run efficiently and provide identical results on a wide variety of computers and operating systems.

NX Nastran contains the following analysis capabilities:

- Linear statics (including inertia relief)
- Normal modes and buckling
- Heat transfer (steady-state and transient)
- Transient response
- Frequency response
- Response spectrum and random response
- Geometric and material nonlinear static and transient response
- Design optimization and sensitivity (including dynamic and shape optimization)
- Composite materials
- Acoustic response
- Aeroelasticity
- Superelements
- Complex eigenanalysis
- Axisymmetric analysis
- Cyclic symmetry
- p-elements

Modular Structure and DMAP

The NX Nastran software is composed of a large number of building blocks called modules. Each module is a collection of FORTRAN subroutines which is designed to perform a specific task, such as processing model geometry, assembling matrices, applying constraints, solving matrix problems, and calculating output quantities.

Within NX Nastran, the modules are controlled by an internal language called the Direct Matrix Abstraction Program (DMAP), which is a high-level programming language with its own compiler and grammatical rules. A DMAP statement is like a subroutine call statement within FORTRAN, with both input and output information.

Pre-defined Solution Sequences

Each type of analysis available in NX Nastran is called a “solution sequence.” Each solution sequence is a pre-defined collection of hundreds or thousands of DMAP commands. Once you select a solution sequence, its particular set of DMAP commands sends instructions to the modules that are needed to perform the requested solution.

NX Nastran also includes DMAP (Direct Matrix Abstraction Programming), a high-level programming language that allows you to build custom solution sequences or modify existing ones.

Using DMAP for Programmability

Although NX Nastran contains a number of pre-defined solutions, you can use special DMAP “alters” to either modify existing solution sequences or create new ones.

See also

- *NX Nastran DMAP Programmer's Guide*

1.2 Overview of the NX Nastran Analysis Process

To solve a finite element model in NX Nastran, you must:

- Create the input file
- Use the nastran command to submit the input file for batch processing
- Examine your results

Creating the Input File

To perform an analysis with NX Nastran, you must generate an input file that contains model data (information about the finite element model, including geometry, elements, materials, and loads) as well as analysis data (information about the type of analysis you want to perform, such as the analysis type and the type of data you want output).

In NX Nastran, the input file is an ASCII text file that you can create:

- Automatically using a preprocessor, such as NX Advanced Simulation
- Manually using a text editor

Input File Sections

An NX Nastran input file contains the following five separate sections:

- NASTRAN statement
- File Management statement (FMS)
- Executive Control statement
- Case Control command

- Bulk Data entry

Documentation for NX Nastran Input Options

Each section in the NX Nastran input file is described in more detail later in this guide. Comprehensive documentation for each File Management and Executive Control statement, Case Control command, and Bulk Data entry is located in the *NX Nastran Quick Reference Guide*.

Input File Naming

In general, Nastran input files are given a .dat extension. However, NX Nastran tries to process any file you submit, regardless of its file extensions. Other common NX Nastran input file extensions include .bdf or .blk.

Submitting the Input File for Analysis

Once you've created the input file, you submit the file for execution as a batch process in NX Nastran. Because NX Nastran is not an interactive program, once you submit the input file, you will not have any additional interaction with the software until the job completes, unless you need to terminate the job.

To execute NX Nastran, you typically use the nastran system command (in some cases, your system administrator may assign a different name to the command) followed by the name of the input file. For example:

```
NASTRAN MODEL1A
```

The nastran command also allows you to specify certain keywords for controlling the job execution. The format of the nastran command is:

```
nastran keyword1 = value1 keyword2 = value2 ...
```

See also

- “Using the nastran Command,” in the *NX Nastran Installation and Operations Guide*
- “nastran Command and NASTRAN Statement,” in the *NX Nastran Quick Reference Guide*

Reviewing the Results

When NX Nastran completes the analysis, you can review your results. NX Nastran automatically creates a number of different types of files, which are summarized in the following table:

File type	Description
.DBALL	Contains permanent data for database runs for restart analyses
.f04	Contains database file information and a module execution summary, such as the start and stop time for each module as well as the size of the database file (if any)
.f06	Contains the results of your analysis, such as displacements and stresses, as well as any diagnostic messages
.LOG	Contains system information and any system level error messages
.MASTER	Contains permanent data for database runs for restart analyses

In addition to these automatically generated files, you can manually request that NX Nastran create the following files:

File type	Description
.pch	Punch file
.plt	Plot file
.op2	Output2 file
.xdb	XDB file

NX Nastran provides for direct support of interfaces to other products, such as NX Nastran Access and I-DEAS Master FEM.

For MSC.Patran, the support is provided through the DBC module, which creates a “graphics” database. The interface to the DBC module is described in “POST,” in the *NX Nastran Quick Reference Guide*.

The graphics database created by the DBC module can also be read by the NX Nastran Access library of object routines. This object library can be linked with a user-created program that extracts data from the database. This is an indirect method for outside vendors to obtain NX Nastran model and results information. See the *NX Nastran Access Reference Manual* for more information.

Communication with other software packages, such as I-DEAS Master FEM, is through the NX Nastran OUTPUT2 module, which creates a FORTRAN readable file that is converted by these outside programs to their own particular data formats. The user interface and the supported data for these outside programs are described in “POST,” in the *NX Nastran Quick Reference Guide* (-1, -2, -4, and -5).

Chapter 2: Structure and Syntax for Input Data

- *Overview of Specifying Input Data*
- *General Input File Syntax Rules*
- *Understanding NX Nastran Input Records*
- *Specifying Continuations for Bulk Data*
- *Replicating Bulk Data Entries*
- *Inserting External Files with INCLUDE*
- *Including Comments in the Input File*

2.1 Overview of Specifying Input Data

In NX Nastran, you submit all your data to the solver through the input file, which is sometimes referred to an input “deck.” In most cases, you will automatically generate your NX Nastran input file by exporting an existing finite element model from a graphical pre-processing program. However, because you may need to manually edit the contents of that input file, it is important that you understand the rules and conventions of entering data into an NX Nastran input file.

This chapter describes:

- The general syntax rules for input data in NX Nastran
- NX Nastran input records, including the different input data formats (free field, small field, and large field)
- How to use the INCLUDE option to include external files in your NX Nastran input file
- How to add comments to your NX Nastran input file

The *NX Nastran Quick Reference Guide* describes the input requirements and default values for all NX Nastran data file input.

2.2 General Input File Syntax Rules

The following sections provide the general rules and conventions for inputting data in NX Nastran.

Understanding Types of Input Data

In Nastran, you can input integer or real numbers, as well as character data.

- Integer numbers cannot contain a decimal point.
- Real numbers, including zero, must contain a decimal point.

You can enter real numbers in a variety of formats. For example, the following are all acceptable versions of the real number, seven:

7.0	.7E1	0.7+1
.70+1	7.E+0	70.-1

- Character data (sometimes called literal, or BCD—binary coded decimal data) can be alphanumeric, but it must always begin with an alpha character and contain eight or fewer total characters.

Importantly, the different types of data are not interchangeable. If a particular field requires an integer and you enter a real number instead, NX Nastran issues an error message.

Using Consistent Unit Systems

NX Nastran is unitless. Accordingly, the units for physical quantities defining the geometry, material properties, and boundary conditions of an NX Nastran model must be consistent. Because NX

Nastran cannot detect inconsistent units, a warning message is not issued when inconsistent units are used. The software simply calculates erroneous results.

As an example, suppose you want to construct a structural model with gram, millimeter, and second as the base units. What units should you use in the NX Nastran input file for force, density, and elastic modulus?

Table 2-1 demonstrates how to determine the units for force, density, and elastic modulus that are consistent with the base units of mm (millimeter), g (gram), and s (second).

Table 2-1. Determining Consistent Units		
Physical Quantity	Dimensions	Consistent Unit
Force	$M \cdot L/T^2$	$g \cdot mm/s^2$
Density	M/L^3	g/mm^3
Elastic modulus	$M/L \cdot T^2$	$g/mm \cdot s^2$
M, L, and T refer to mass, length, and time, respectively.		

Now suppose the model includes an applied force of 5000 N, and a material having a density of 8000 kg/m³ and an elastic modulus of 200 GPa. What numerical values should you use in the NX Nastran input file?

You must convert N, kg/m³, and GPa to g·mm/s², g/mm³, and g/mm·s², respectively. Table 2-2 shows how to calculate the appropriate conversion factors.

Table 2-2. Determining Conversion Factors	
Physical Quantity	Conversion Factor
Force	$1 \text{ g} \cdot \text{mm}/\text{s}^2 = (10^{-3} \text{ kg})(10^{-3} \text{ m})/\text{s}^2 = 10^{-6} \text{ kg} \cdot \text{m}/\text{s}^2 = 10^{-6} \text{ N}$
Density	$1 \text{ g}/\text{mm}^3 = (10^{-3} \text{ kg})/(10^{-3} \text{ m})^3 = 10^6 \text{ kg}/\text{m}^3$
Elastic modulus	$1 \text{ g}/\text{mm} \cdot \text{s}^2 = (10^{-3} \text{ kg})/(10^{-3} \text{ m})\text{s}^2 = 1 \text{ kg}/\text{m} \cdot \text{s}^2 = 1 \text{ N}/\text{m}^2 = 1 \text{ Pa} = 10^{-9} \text{ GPa}$

Using the conversion factors in Table 2-2 you can calculate the numerical values to use in the NX Nastran input file as follows:

- $5000 \text{ N} = 5000 \text{ N} (1 \text{ g} \cdot \text{mm}/\text{s}^2/10^{-6} \text{ N}) = 5 \times 10^9 \text{ g} \cdot \text{mm}/\text{s}^2 \rightarrow \text{Use } 5.E9 \text{ in the input file.}$
- $8000 \text{ kg}/\text{m}^3 = 8000 \text{ kg}/\text{m}^3 (1 \text{ g}/\text{mm}^3/10^6 \text{ kg}/\text{m}^3) = 8 \times 10^{-3} \text{ g}/\text{mm}^3 \rightarrow \text{Use } 8.E-3 \text{ in the input file.}$
- $200 \text{ GPa} = 200 \text{ GPa} (1 \text{ g}/\text{mm} \cdot \text{s}^2/10^{-9} \text{ GPa}) = 2 \times 10^{11} \text{ g}/\text{mm} \cdot \text{s}^2 \rightarrow \text{Use } 2.E11 \text{ in the input file.}$

Similarly, the results that NX Nastran produces are in units consistent with the specified set of base units. For example, suppose that NX Nastran outputs the stress at a particular grid to be 1.E8 in the above example. You would interpret this result to be $1 \times 10^8 \text{ g}/\text{mm} \cdot \text{s}^2$.

When solving mechanics problems, mass, length, and time are usually selected as the base units. However, when solving mechanics problems using the finite element method, you typically use force, length, and time as the base units. Force, length, and time are convenient to use because you typically know:

- The physical dimensions of the component you are modeling.
- The forces that act on the component you are modeling.

Table 2-3 lists five sets of consistent units that are commonly used in structural analysis.

For information regarding consistent units for thermal analysis, see *Consistent Units for Thermal Analysis* in the *Thermal Analysis User's Guide*.

Table 2-3. Consistent Sets of Units for Structural Analysis – Base Units of Force, Length, and Time						
Physical Quantity	Dimen-sions	English lbf-in-s	English lbf-ft-s	SI mN-mm-s	SI N-mm-s	SI N-m-s
<i>Base units</i>						
Force	F	lbf	lbf	mN	N	N
Length	L	in	ft	mm	mm	m
Time	T	s	s	s	s	s
<i>Consistent units for typical inputs</i>						
Acceleration	L/T^2	in/s^2	ft/s^2	mm/s^2	mm/s^2	m/s^2
Angular acceleration	$1/T^2$	rad/s^2	rad/s^2	rad/s^2	rad/s^2	rad/s^2
Angular velocity	$1/T$	rad/s	rad/s	rad/s	rad/s	rad/s
Coordinate	L	in	ft	mm	mm	m
Density	$F \cdot T^2/L^4$	$\text{lbf} \cdot \text{s}^2/\text{in}^4$	slug/ft^3	kg/mm^3	tonne/mm^3	kg/m^3
Displacement	L	in	ft	mm	mm	m
Elastic modulus	F/L^2	psi	lbf/ft^2	kPa	MPa	Pa
Force	F	lbf	lbf	mN	N	N
Mass	$F \cdot T^2/L$	$\text{lbf} \cdot \text{s}^2/\text{in}$	slug	kg	tonne	kg
Moment	$F \cdot L$	$\text{lbf} \cdot \text{in}$	$\text{lbf} \cdot \text{ft}$	$\text{mN} \cdot \text{mm}$	$\text{N} \cdot \text{mm}$	$\text{N} \cdot \text{m}$
Pressure	F/L^2	psi	lbf/ft^2	kPa	MPa	Pa
Velocity	L/T	in/s	ft/s	mm/s	mm/s	m/s
<i>Consistent units for typical results</i>						
Displacement	L	in	ft	mm	mm	m
Force	F	lbf	lbf	mN	N	N
Strain energy	$F \cdot L$	$\text{lbf} \cdot \text{in}$	$\text{lbf} \cdot \text{ft}$	μJ	mJ	J
Stress	F/L^2	psi	lbf/ft^2	kPa	MPa	Pa
1 slug = 1 $\text{lbf} \cdot \text{s}^2/\text{ft}$; 1 tonne = 1000 kg; 1 psi = 1 lbf/in^2 ; 1 Pa = 1 N/m^2 ; 1 J = 1 $\text{N} \cdot \text{m}$						
F, L, and T refer to force, length, and time, respectively.						

Specifying Valid Input File Characters

Table 2-4 lists the valid characters for NX Nastran.

Table 2-4. ASCII Subset for NX Nastran			
Character	ASCII (HEX)	Character	ASCII (HEX)
HT (Horizontal Tabulation)	09	@	40
		A	41
SP (Space “blank”)	20	B	42
		C	43

Table 2-4. ASCII Subset for NX Nastran

Character	ASCII (HEX)	Character	ASCII (HEX)
!	21	D	44
"	22	E	45
#	23	F	46
\$	24	G	47
%	25	H	48
&	26	I	49
'	27	J	4A
(28	K	4B
)	29	L	4C
*	2A	M	4D
+	2B	N	4E
,	2C	O	4F
-	2D	P	50
.	2E	Q	51
/	2F	R	52
0	30	S	53
1	31	T	54
2	32	U	55
3	33	V	56
4	34	W	57
5	35	X	58
6	36	Y	59
7	37	Z	5A
8	38	[5B
9	39	\	5C
:	3A]	5D
;	3B	^	5E
<	3C	-	5F
=	3D		
>	3E		
?	3F		

2.3 Understanding NX Nastran Input Records

Input data records are 80 characters (or columns) in length on each line. Those records are then divided into fields. How those records are divided depends upon the data format.

There are two basic categories of input data formats in NX Nastran:

- “Free” format data, in which the data fields are simply separated by commas. This type of data is known as free field data.

- “Fixed” format data, in which your data must be aligned in columns of specific width. There are two subcategories of fixed format data that differ based on the size of the fixed column width:
 - o Small field format, in which a single line of data is divided into 10 fields that can contain eight characters each.
 - o Large field format, in which a single line of input is expanded into two lines. The first and last fields on each line are eight columns wide, while the intermediate fields are sixteen columns wide. The large field format is useful when you need greater numerical accuracy.

Which type of format data you use depends upon the section of the input file that you are working with.

- You must use the free field format for the NASTRAN statement and the File Management, Executive Control, and Case Control sections.
- You can use the free field, small field, or large field formats with the Bulk Data section.

The rules for entering free field, small field, and large field format data are described in more detail below.

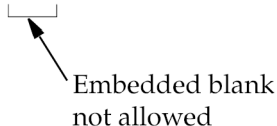
Understanding Free Field Format

In free field format, you separate data fields by commas.

Free Field Format Rules

- You must begin free field data entries in column 1.
- You can use commas to skip fields.
 - o To skip one field, use two commas in succession.
 - o To skip two fields, use three commas in succession (and so on).
- Integer or character data is limited to eight characters. If you include an integer or character longer than eight characters, NX Nastran issues a fatal error.
- With free field format data, NX Nastran rounds real numbers with more than eight characters. Therefore, you may lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If you need more significant digits, use the large field format.
- Free field data cannot contain embedded blanks. An example of a free field embedded blank is shown below:

```
GRID, 2, , 1 0, -2.0, 3.0, , 136
```



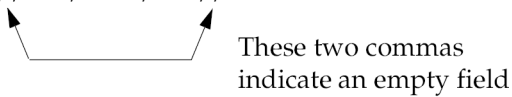
Embedded blank
not allowed

- You use a dollar sign (\$) to terminate the free field entry. You can follow the dollar sign with comments.

Free Field Format Example

The following example shows the GRID Bulk Data entry in free field format:

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



These two commas indicate an empty field

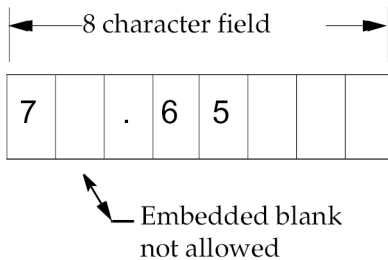
Understanding Small Field Format

With the small field format, the software separates an 80-column line of Bulk Data entry into 10 equal fields of eight characters each:

1	2	3	4	5	6	7	8	9	10
<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->

Small Field Format Rules

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right- or left-justified. However, aligning the data fields is a good practice.
- You can't include any embedded blanks in small field format input data. An example of a small field embedded blank is shown below:



8 character field							
7		.	6	5			

Embedded blank not allowed

Small Field Format Example

The following example shows the GRID entry in small field format:

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

Understanding Large Field Format

A high degree of numerical accuracy is required in some NX Nastran applications. You can use large field format when small field format doesn't provide enough significant digits. For example, you can use large field format when you need to input integers larger than eight characters, or you need to enter real numbers with more significant digits than you can fit in an eight-character field.

Note: A minus sign, decimal point, and the “E” in scientific notation count as characters.

Large Field Format Rules

Large field format requires (at least) two lines for each entry:

- The first and last field of each line contains eight columns.
- The fields in between contain 16 columns.

Large field entries are denoted by an asterisk (*). The asterisk must:

- Immediately follow the character string in field 1A of the first line of the entry
- Immediately precede the character string in field 1B of the second line of the entry

Large Field Format Example

The following is an example of the GRID Bulk Data entry in large field format:

First Line: (Left half of single field)

Field	1A	2	3	4	5	6
	GRID*	2		1.0	-2.0	*GRID10
	← 8 →	← 16 →	← 16 →	← 16 →	← 16 →	← 8 →
	columns					

Second Line: (Right half of single field)

Field	1B	6	7	8	9	10B
	.GRID10	3.0		136		
	← 8 →	← 16 →	← 16 →	← 16 →	← 16 →	← 8 →
	columns					

2.4 Specifying Continuations for Bulk Data

Some Bulk Data entries require more than eight fields (72 columns) of data. In those cases, you need to specify a continuation to continue the remainder of the data on one or more subsequent lines. To do this, you follow the parent entry (the first line) with one or more continuation entries on subsequent lines.

There are two ways to specify a continuation:

- Manually
- Automatically

Manually Specifying a Continuation with Identifiers

You can manually specify a continuation by using a continuation identifier. A continuation identifier is a special character (+ or *) that indicates that the data continues on another line. Once you've

specified the continuation identifier, you then follow the parent entry (the first line) with one or more continuation entries on subsequent lines.

Rules for Manually Specifying a Continuation

- You must use a + or * in column 1, field 1 of a continuation entry.
 - The alphanumeric character strings you use as the continuation identifiers cannot contain the symbols *, =, or \$.
- Note: The + symbol used in column 1 of field 10 is for clarity.
- Field 10 contains the continuation identifier that must be unique with respect to all other continuation identifiers.
 - NX Nastran always ignores column 1 of field 10. It is not considered to be part of the identifier.
 - The remaining contents in columns 2-8, field 1 of a continuation entry must be identical to the entry in field 10 (columns 2-8) of either the parent entry or the preceding continuation entry.
 - You can use small and large field continuation entries together to define a single data item entry.

Manual Continuation Example

For example, consider the following PBAR simple beam property entry:

Standard PBAR Entry

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

Continuation Example

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

In this example:

- +PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line of the entry.
- +PB2 in field 10 of the second line points to the third line.

Automatically Specifying a Continuation

The automatic continuation method allows you to avoid the continuation identifiers.

To have NX Nastran automatically generate continuations:

- Leave both field 10 of the parent entry and field 1 of the continuation line blank.

- Place the continuation line directly following the parent entry.

The blank field indicates that it is a small-field continuation of the previous line.

You can mix small-field and large-field format continuations. However, it is generally not recommended because it becomes difficult to discern the locations of the fields.

Rules for Automatically Generating Continuations

To have NX Nastran automatically generate a continuation:

- The continuation line(s) must immediately follow the parent Bulk Data entry.
- You must leave fields 1 and 10 of the continuation line (or lines) blank.
- Your bulk data must be in sorted order.

Note: Setting SYSTEM CELL 357 to 1 tells NX Nastran to ignore the continuation field. This eliminates the concern of duplicate continuation identifiers.

Automatic Continuation Example

This process is illustrated in the following example:

CHEXA,	1,	10,	3,	5,	7,	1,	15,	17,
,	19,	13,	4,	6,	8,	2,	10,	11,
,	12,	9,	16,	18,	20,	14		

Input (.DAT) file

GEAR TOOTH EXAMPLE										JULY 10, 2003 NX NASTRAN 07/10/2003										PAGE 3	
										S O R T E D B U L K D A T A E C H O											
ENTRY COUNT		1		2		3		4		5		6		7		8		9		10	
1-		CHEXA		1		10		3		5		7		1		15		17		+000001	
2-		+000000119				13		4		6		8		2		10		11		+000002	
3-		+000000212				9		16		18		20		14						+000003	

Output (.f06) file

Note that column 1 of field 10 contains a + symbol. Since NX Nastran ignores this column, the choice of the + is arbitrary. However, using the + symbol improves readability and serves as a reminder for the format of the continuation line.

2.5 Replicating Bulk Data Entries

To avoid having to input each bulk data entry individually, you can replicate the repetitive fields from a single entry definition. You can use replication with bulk data that is in fixed-field or free-field format for small field (that is, 8 character) input. Large field (that is, 16 character) input is not supported for either fixed-field or free-field format, except in the case of bulk entries that have only the first four fields specified. An example of this is GRID entries with the X3 and subsequent fields left blank.

- To replicate selected fields from the preceding entry, use the symbol =.
- To replicate all the remaining entries from the preceding entry, use the symbol ==.

- To generate an incremented value from the previous entry, use *x or *(x), where x is the value of the increment. "x" should be a real number for real fields or an integer for integer fields.
- To repeatedly replicate a field, use =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.

Replicating Continuation Entries

If you are manually creating continuation entries, you can replicate the continuation fields (fields one and ten) using the following conventions:

- You can use only letters of the alphabet and integers. They are coded into a base 36 number. That is, the sequence of numbers is 0,1, 2, ..., 8, 9, A, B, ...
- The software increments the continuation field by +1 regardless of the value you specify.
- The software does not increment the first character in fields 1 or 10.
- The software will not ever increase the number of characters in an incremented field. For example, if the first field is "0", the thirty-seventh field will also be "0", resulting in an illegal entry. A method to solve this problem is to start with a first field of "00". This provides thirty-six squared unique fields.
- At least one field in fields 2 through 8 of continuation entries must contain data.

Replication with MSGMESH

Replication is a limited data generation capability. You can use replication with bulk data that is either fixed (large or small field) or free field format.

- The MSGMESH capability includes the capabilities described here, plus the following capabilities as long as NASTRAN MESH is specified in the File Management Section.
 - o Continuation entry fields may be incremented or decremented.
 - o Repeated replication is indicated by coding =(n) in field 1, where n is number of entry images to be generated using the values of increments from the current or preceding replication entry.

Entered entries:

```
GRID,101,17,1.0,10.5,,17,3456
=(4),*(1),=,*(0.2),==$
```

Generated entries:

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.5		17	3456		
GRID	103	17	1.4	10.5		17	3456		
GRID	104	17	1.6	10.5		17	3456		
GRID	105	17	1.8	10.5		17	3456		

- o A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

```
BSET1,123,1,2,3,4,5,6,7
,,*7,*7,*7,*7,*7,*7,*7
=(3)
```

Generated entries:

BSET1	123	1	2	3	4	5	6	7	+00001
++00001		8	9	10	11	12	13	14	+00002
++00002		15	16	17	18	19	20	21	+00003
++00003		22	23	24	25	26	27	28	+00004
++00004		29	30	31	32	33	34	35	+00005

- o A “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of nine entries may be replicated as a group. The default continuation entry increment is 10. Example:

Entered entries:

```
CTRIA3,10,1,1,10,11/+C1
=(D),*(1),=,=,*(1),*(1)/*(20)
+C1,,2.0,1.0,1.0
=(2),==
```

Generated entries:

CTRIA3	10	1	1	10	11				+C1
+C1			2.0	1.0	1.0				
CTRIA3	11	1	1	11	12				+C21
+C21			2.0	1.0	1.0				
CTRIA3	12	1	1	12	13				+C41
+C41			2.0	1.0	1.0				

- o Parentheses are optional on replication entries and an equal sign may replace an asterisk.

The following is an example of the use of replication, automatic continuation field generation, and the free field format:

```
GRID,101,17,1.0,10.5,,17,3456
=,*1,=,*0.2,*0.1),==$ COMMENTS MAY APPEAR AFTER $
=3
EIGR,13,LAN
CBAR,1,1,101,102,0.,0.,1.,+0
=,*1,=,*1,*1====*1
+0,56
*1,=$
```

The above free-field entries generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

Note: A “,” should always be used after the “*1” for the continuation increment even if fixed field format is being used.

CBAR	1	1	101	102	0.	0.	1.		+0
+0	56								
CBAR	2	1	102	103	0.	0.	1.		+1
+1	56								
EIGR	13	LAN							

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.6		17	3456		
GRID	103	17	1.4	10.7		17	3456		
GRID	104	17	1.6	10.8		17	3456		
GRID	105	17	1.8	10.9		17	3456		

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user's responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

2.6 Inserting External Files with INCLUDE

You can use the INCLUDE statement to insert external files into your NX Nastran input file. You can specify INCLUDE in any part of the input file as long as the data referenced by the INCLUDE statement is appropriate for that section of the input file. This allows you to avoid repetitious input in the different sections of the input file.

Using INCLUDE in Different Input File Sections

You can use INCLUDE in a variety of ways in the different sections of your input file.

- In the File Management section, you can use the INCLUDE statement to assign multiple database files with one statement.
- In the Executive section, you can use the INCLUDE statement to include alters to a solution sequence.
- In the Case Control section, the INCLUDE file can be useful if you want to use the same subcase structure and/or the same output requests for different models.
- In the Bulk Data section, the INCLUDE statement is very useful for working with superelement data. You can use it to place each superelement of a superelement model in a different physical file. You can then share the files with other users and analyze them separately. Since superelement models are usually very large and complex, using the INCLUDE statement can greatly reduce the complexity of the input file.

INCLUDE Statement Limitations

- Although you can nest INCLUDE statements within the external file, the nested depth level must not be greater than 10.
- Each line has a 72 character limit. Multiple lines can be used when file names are long. The entire string must begin and end with the single quote character.

For example,

D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat

can be defined on multiple lines:

```
INCLUDE 'D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat'
```

The following input format is also supported:

```
INCLUDE 'D:\folder1\folder2\folder3\
      folder4\folder5\
      folder6\folder7\
      folder8\folder9\mydata.dat'
```

Example: Using the INCLUDE Statement

Below is an example of the use of the INCLUDE statement. Note that the syntax for the filenames is machine dependent. The format of lowercase filenames within single quotes works on all computer operating systems that support NX Nastran.

```
INCLUDE 'nastran.dat'
INCLUDE 'database.dat'
SOL 101
INCLUDE 'sol101.alter'
CEND
TITLE = Complete Model
SUBTITLE = Three load cases
INCLUDE 'output.dat'
SUBCASE 1
  LOAD = 10
BEGIN BULK
INCLUDE 'se0.dat'
INCLUDE 'se10.dat'
INCLUDE 'se20.dat'
ENDDATA
```

Each of the INCLUDE files in the example contains different information. For this example, “nastran.dat” contains NASTRAN system cells, “database.dat” contains FMS statements, “sol101.alter” contains a DMAP Alter for SOL 101, “output.dat” contains Case Control commands, and “se*.dat” contains modeling information for superelements 0, 10, and 20.

See also

- “**INCLUDE**,” in the *NX Nastran Quick Reference Guide*

2.7 Including Comments in the Input File

You can insert comments in any part input file to help organize your input data. Comment entries begin with a dollar (\$) sign in column one followed by any characters out to column 80. Comments appear only in the unsorted echo of the input file listing.

Chapter 3: Understanding the Input Data File

- *Overview of the NX Nastran Input File Sections*
- *The NASTRAN Statement (Optional)*
- *File Management Section (Optional)*
- *Executive Control Section (Required)*
- *Case Control Section (Required)*
- *Bulk Data Section*
- *Using Parameters*

3.1 Overview of the NX Nastran Input File Sections

The input file is comprised of five distinct sections and three required one-line delimiter which must follow the sequence shown in [Figure 3-1](#). These delimiters and sections are described below.

Note: If any of the required delimiters are missing from your input file, NX Nastran issues a fatal error message when you submit your job.

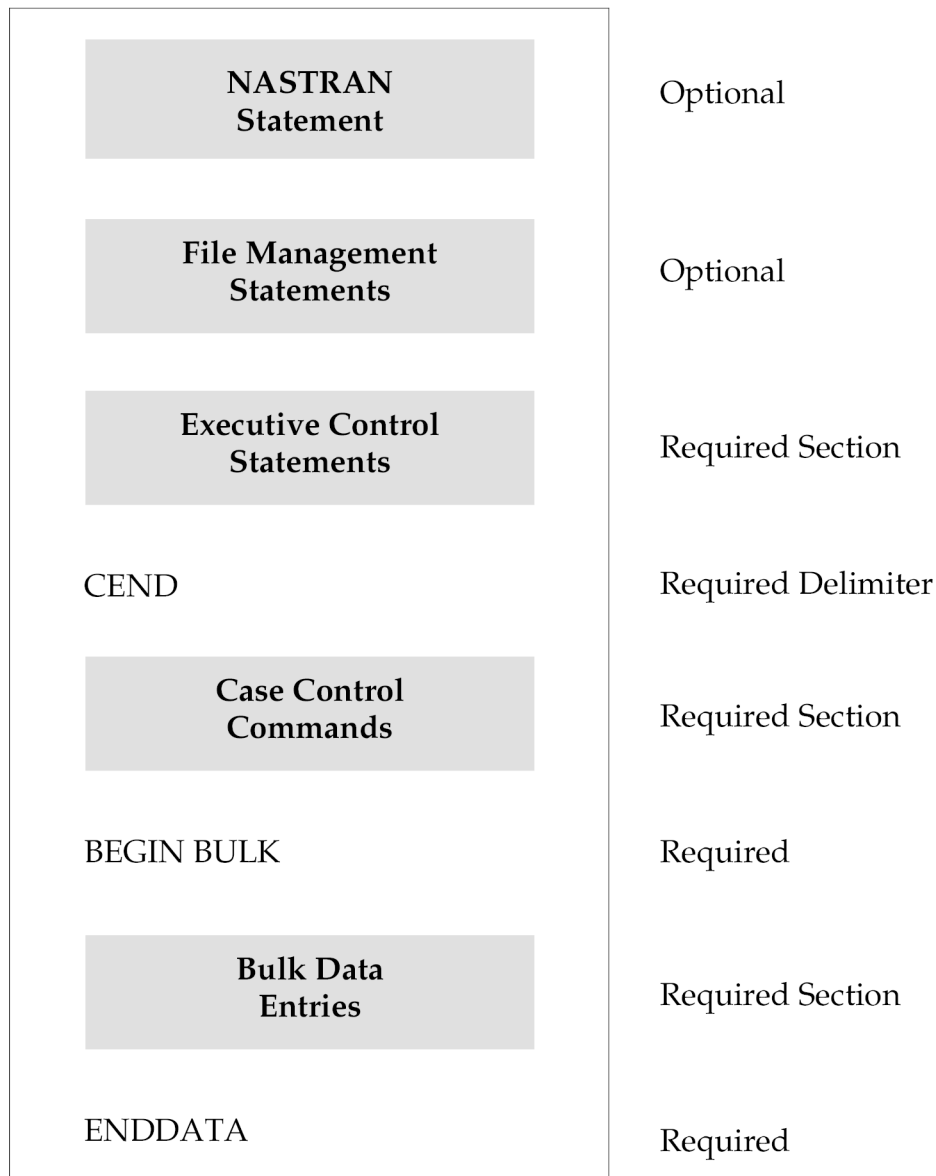
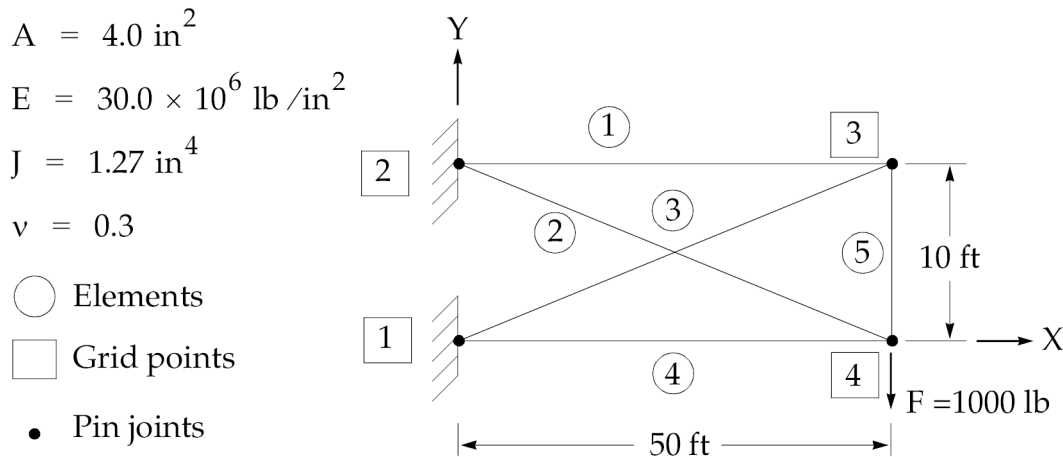


Figure 3-1. NX Nastran Input File Structure

Example: Using NX Nastran Input Files to Model a Truss

Throughout this chapter, a simple truss structure, shown in [Figure 3-2](#), is used to illustrate different aspects of input files.

**Figure 3-2. Truss Example**

The truss model consists of four grid points that represent the joints of the structure. The locations of these grid points are given in the default coordinate system for NX Nastran, which is known as the “basic coordinate system.” In this example, the origin is located at grid point 1.

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

Although the dimensions given in **Figure 3-2** are in feet, the dimensions used for the grid point locations in the input file (**Listing 3-1**) are in inches. For consistency, the units for the cross-sectional area, torsional stiffness, and Young's modulus in the input file are also provided in inches.

Note: You must always ensure that you enter data in a consistent system of units.

Listing 3-1 shows the input file for the truss structure. Although this input file represents a fairly simple structure, it is typical of all NX Nastran input files.

<pre> \$ FILENAME - TRUSS1.DAT \$ ID LINEAR, TRUSS1 SOL 101 TIME 2 CEND TITLE = LINEAR STATICS USER'S SAMPLE INPUT FILE SUBTITLE = TRUSS STRUCTURE LABEL = POINT LOAD AT GRID POINT 4 LOAD = 10 SPC = 11 DISPLACEMENT = ALL ELFORCE = ALL ELSTRESS = ALL </pre>	Executive Control section
	Case Control section

<pre> BEGIN BULK \$ \$ THE GRID POINTS LOCATIONS \$ DESCRIBE THE GEOMETRY \$ GRID 1 0. 0. 0. 3456 GRID 2 0. 120. 0. 3456 GRID 3 600. 120. 0. 3456 GRID 4 600. 0. 0. 3456 \$ \$ MEMBERS ARE MODELED USING \$ ROD ELEMENTS \$ CROD 1 21 2 3 CROD 2 21 2 4 CROD 3 21 1 3 CROD 4 21 1 4 CROD 5 21 3 4 \$ \$ PROPERTIES OF ROD ELEMENTS \$ PROD 21 22 4. 1.27 \$ \$ MATERIAL PROPERTIES \$ MAT1 22 30.E6 .3 \$ \$ POINT LOAD \$ FORCE 10 4 1000. 0. -1. 0. \$ SPC1 11 123456 1 2 \$ ENDDATA </pre>	Bulk Data section
--	-------------------

Listing 3-1. Truss Example Input File

3.2 The NASTRAN Statement (Optional)

The NASTRAN statement lets you modify the default values for certain operational parameters (also called system cells) at runtime to control internal solution processing or provide specific diagnostics. For example, you can use the NASTRAN statement to control aspects of working memory, datablock size, and datablock parameters.

- You can include more than one NASTRAN statement in a single input file.
- You can combine the NASTRAN statement with the File Management section.
- You can specify the NASTRAN statement in the Runtime Configuration (RC) files at the system, user, and job level.

See also

- “Using the NASTRAN Statement” in the *NX Nastran Installation and Operations Guide*
- “nastran Command and NASTRAN Statement” in the *NX Nastran Quick Reference Guide*
- “**BUFFSIZE**” in the *NX Nastran User's Guide*

3.3 File Management Section (Optional)

The File Management section (FMS) of the input file is optional and typically follows the NASTRAN statement(s). It ends with the specification of an Executive Control statement. For example, the File Management section lets you:

- Assign files
- Attach or initialize the NX Nastran database sets (DBsets) and FORTRAN files
- Perform restart analyses

The initialization of a database includes specification of its maximum size, member names, and physical filenames. The initialization of a FORTRAN file includes the specification of its filename, FORTRAN unit numbers, and FORTRAN attributes.

The File Management section is commonly used in input files for larger analysis problems. For many NX Nastran problems, no File Management statements are required because a default File Management section is executed by the software at the beginning of every run. For example, since the truss model shown in [Listing 3-1](#) is small and the desired analysis is not a restart, a File Management section isn't included in the input file.

See also

- “[File Management Statements](#)” in the *NX Nastran Quick Reference Guide*
- “[Database Concepts](#)” in the *NX Nastran User's Guide*
- “[Restarts](#)” in the *NX Nastran User's Guide*

3.4 Executive Control Section (Required)

The Executive Control section is the first required section in the NX Nastran input file and provides the overall control for the solution. For example, statements in the Executive Control section let you:

- Specify an optional ID statement to identify the job
- Select the type of analysis (solution sequence) you want to perform
- Specify an optional TIME statement to set limits on the maximum allowable CPU time for the run
- Request various diagnostics
- Include complete DMAP sequences, if you want to use DMAP in the analysis

For example, the Executive Control section for the truss example ([Listing 3-1](#)) identifies the job, requests the static Structured Solution Sequence 101, and sets the maximum time limit for the job to two minutes.

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

See also

- “**Executive Control Statements**” in the *NX Nastran Quick Reference Guide*

Executive Control Format

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

You must use the CEND delimiter to mark the end of the Executive Control section.

Order Dependence for Certain DMAP-related Statements

Most statements within the Executive Control section are order independent. However, the following statements, all related to the use of DMAP, are order dependent:

- COMPILE
- COMPILER
- ALTER
- ENDALTER
- LINK

Notably:

- The LINK statement must appear after all COMPILE statements.
- The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. This statement sets the default print control for subsequent COMPILE statements.

Commonly Used Executive Control Statements

Some commonly used Executive Control statements are:

- ID
- SOL
- ECHOON and ECHOOFF
- TIME
- DIAG
- CEND

The ID Statement (Optional)

You can use the ID statement to identify your job. Its format is:

```
ID      i1, i2
```

where i1 and i2 are character strings. i1 may be one to eight characters in length. i2 may be of any length. The first character of each string must be alphabetic.

You can use only one ID statement, although you can specify it anywhere in the Executive Control section. The format of the ID statement is the keyword “ID” followed by a delimiter and a comment. The comment may be any ASCII characters. For example:

```
ID THIS IS RUN 2
```

See also

- “ID” in the *NX Nastran Quick Reference Guide*

The SOL Statement (Required)

The SOL statement specifies the type of analysis (solution sequence) you want to perform. The format of the SOL statement is:

```
SOL      n
```

where n is a positive integer identifying the solution type or the character name of the solution procedure.

See also

- “Understanding Solution Sequences” in the *NX Nastran User’s Guide*
- “SOL” in the *NX Nastran Quick Reference Guide*
- “SOL 601,N or SOL 701” in the *NX Nastran Quick Reference Guide*

ECHO Statement (Optional)

The ECHO statement lets you control whether the software prints (“echoes”) the Executive Control section to the output file. ECHOON is the default.

- Use ECHOON (default) to print the section to the output file.
- Use ECHOOFF to prevent the software from printing the section to the output file.

In general, using ECHOOFF hinders the usability of your output file, especially if you need to review the output file several months after your original analysis. However, the ECHOOFF statement can be useful for security. For example, you can use ECHOOFF, to prevent NX Nastran from printing proprietary portions of the Executive Control section (such as user-written DMAP) to the output file.

You can use multiple ECHOON and ECHOOFF statements in an single input file.

See also

- “ECHO” in the *NX Nastran Quick Reference Guide*

Time Statement (Optional)

The TIME statement allows you to set the maximum CPU time and I/O time for the job before NX Nastran terminates the run. Its format is

```
TIME      t1, t2
```

where:

- t1 is the maximum allowed execution time in CPU minutes (real or integer; default = one minute, which is adequate only for very small jobs).
- t2 is the maximum allowable I/O time in seconds (default is infinity).

Importantly, the time limits you can set with the TIME statement are different from the system time limit imposed by your system administrator. In general, you should use the TIME statement to set the NX Nastran time limit to a value less than the system time limit. If your job exceeds the system time limit during execution, the system aborts the job, which may corrupt the database. Once the database is corrupted, it is difficult, if not impossible, to restart the job.

However if the required execution time exceeds the time specified on the NX Nastran TIME statement, NX Nastran cleanly terminates the job. This allows you to restart the job later. Also, NX Nastran estimates the CPU time required to perform certain operations during execution. If the time remaining is not sufficient to complete a particular operation, the job terminates without wasting any additional computer resources.

See also

- “**TIME**” in the *NX Nastran Quick Reference Guide*

DIAG Statement (Optional)

You can use the DIAG statement to request a variety of additional diagnostic information or output and/or to modify the solution.

See also

- “**DIAG**” in the *NX Nastran Quick Reference Guide*
- “Other Optional Diagnostic Information” in the *NX Nastran User's Guide*

The CEND Statement (Required)

The CEND statement is a required statement that designates the end of the Executive Control section (and the beginning of the Case Control section). Its format is simply:

```
CEND
```

For example:

```
ID  SIMPLE,MODEL
SOL 101
TIME 5
CEND
```

See also

- “**CEND**” in the *NX Nastran Quick Reference Guide*

3.5 Case Control Section (Required)

The Case Control section contains commands. It always follows the Executive Control section and precedes the Bulk Data section. You use the Case Control section to:

- Define/select analysis subcases (multiple loadings in a single job execution)
- Define SETS to specify and control the type of analysis output produced (e.g., forces, stresses, and displacements)
- Select output requests for printing, punching and plotting
- Define titles, subtitles and labels for documenting the analysis

For the truss example, a title, subtitle, and label are defined. These labels are printed on each page of the .f06 output file. The LOAD = 10 command instructs NX Nastran to apply the loading defined by the FORCE entry with an ID of 10 in the Bulk Data section. The SPC = 11 command instructs NX Nastran to apply the constraints defined by the SPC1 entry with an ID of 11 in the Bulk Data section. Printed displacements for all the grid points and the forces and stresses within each member are requested with the DISPLACEMENT = ALL, FORCE = ALL, and STRESS = ALL commands. By default, these output requests are printed to the .f06 output file.

See also

- “[Case Control Commands](#)” in the *NX Nastran Quick Reference Guide*.

Case Control Format

The Case Control section must end with the BEGIN BULK delimiter.

Defining SUBCASES

The SUBCASE case control command defines each subcase within your analysis. You define a subcase with the SUBCASE command together with an integer to identify the subcase number, such as:

```
SUBCASE i
```

In general, a separate subcase is defined for each loading condition and/or each set of constraints. Subcases may also be used for output requests.

Case control commands placed above the subcase level (before the first subcase) are used as the default for all subcases, unless overridden within an individual subcase.

For example:

```

1 { CEND
    TITLE=Equipment Rack Analysis
    ECHO=BOTH
    SPC=20
    SET 1=1 THRU 50
    DISP=1
    SUBCASE 1
        SUBTITLE=Dead Load
        LOAD=10
        DISP=All
    SUBCASE 2
        SUBTITLE=NW Wind Load
        LOAD=20
        SET 10=2,4,6
        DISP=10
    SUBCASE 3
        SUBTITLE=SW Wind Load
        LOAD=30

```

1	Case Control requests above the first subcase level apply to all subcases unless overridden by requests within a subcase.
2	Case Control requests defined within a subcase are limited to that subcase only.

Defining Output

By default, NX Nastran doesn't output any results. With NX Nastran, you must use Case Control commands to explicitly request the results you want. NX Nastran output is requested in the Case Control Section.

Echoing Bulk Data in the Output

An NX Nastran model resides in a .DAT text file. The .DAT file is submitted to NX Nastran and an .f06 results file is produced.

The following Case Control command produces sorted and unsorted model file listings, called echoes, at the beginning of the .f06 file:

```
ECHO=BOTH
```

The unsorted input file is an exact copy of the Executive Control, Case Control, and Bulk Data Sections of the input (.DAT) file, including comment (\$) entries. The sorted input file is a listing of the Bulk Data Section with entries rearranged in alphabetical order and with comments removed. In addition, the sorted Bulk Data is expanded to ten fields, each eight columns wide. Therefore, if the .DAT file is entered in free field format, it appears in small field format in the sorted Bulk Data listing. This small field format listing is especially helpful when reviewing the model's Bulk Data Section.

Examples of unsorted and sorted input listings are shown in [Listing 3-2](#) and [Listing 3-3](#).

N A S T R A N E X E C U T I V E C O N T R O L D E C K E C H O ID MPM,EXAMPLE1 SOL 101 TIME 100 CEND		JULY 10, 2003 NX NASTRAN 7/10/2003 PAGE 1
SIMPLY SUPPORTED BEAM WITH CONCENTRATED FORCE		JULY 10, 2003 NX NASTRAN 07/10/2003 PAGE 2
CASE CONTROL DECK ECHO CARD COUNT 1 ECHO=BOTH 2 DISP=ALL 3 STRESS=ALL 4 FORCE=ALL 5 SPCF=ALL 6 SPC=100 7 LOAD=10 8 TITLE=SIMPLY SUPPORTED BEAM 9 SUBTITLE=WITH CONCENTRATED FORCE 10 \$ 11 BEGIN BULK		
SIMPLY SUPPORTED BEAM WITH CONCENTRATED FORCE		JULY 10, 2001 NX NASTRAN 7/10/2003 PAGE 3
INPUT BULK DATA DECK ECHO 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 \$ DEFINE GRID POINTS GRID,1,,0.,0.,0. GRID,2,,10.,0.,0. GRID,3,,20.,0.,0. GRID,4,,30.,0.,0. \$ \$ DEFINE CBAR ELEMENTS CBAR,1,101,1,2,0.,1.,0. CBAR,2,101,2,3,0.,1.,0. CBAR,3,101,3,4,0.,1.,0. \$ \$ DEFINE CBAR ELEMENT CROSS SECTIONAL PROPERTIES PBAR,101,201,2.,.667,.1667,.458,,,+PB1 +PB1,1.,.5,-1.,.5 \$ \$ DEFINE MATERIAL PROPERTIES MAT1,201,30.E6,,.3 \$ \$ DEFINE SPC CONSTRAINT SET SPC1,100,12345,1 SPC1,100,12345,4 \$ \$ DEFINE CONCENTRATED FORCE FORCE,10,3,, -100.,0.,1.,0. \$ ENDDATA INPUT BULK DATA CARD COUNT = 26		

Listing 3-2. Unsorted Listings

SIMPLY SUPPORTED BEAM WITH CONCENTRATED FORCE				JULY 10, 2003 NX NASTRAN 7/10/2003 PAGE 4									
				S O R T E D B U L K D A T A E C H O									
CARD	1	2	3	4	5	6	7	8	9	10			
COUNT													
1-	CBAR	1	101	1	2	0.	1.	0.					
2-	CBAR	2	101	2	3	0.	1.	0.					
3-	CBAR	3	101	3	4	0.	1.	0.					
4-	FORCE	10	3		-100.	0.	1.	0.					
5-	GRID	1		0.	0.	0.							
6-	GRID	2		10.	0.	0.							
7-	GRID	3		20.	0.	0.							
8-	GRID	4		30.	0.	0.							
9-	MAT1	201	30.E6		.3								
10-	PBAR	101	201	2.	.667	.1667	.458			+PB1			
11-	+PB1	1.	.5	-1.	.5								
12-	SPC1	100	12345	1									
13-	SPC1	100	12345	4									
ENDDATA													
TOTAL COUNT= 14													

Listing 3-3. Sorted Listing

Other options for the ECHO command include:

ECHO = SORT

ECHO = UNSORT

Prints only sorted Bulk Data (this is the default)
Prints only unsorted Bulk Data

ECHO = NONE	Turns off the Bulk Data listing
ECHO = PUNCH	Prints a sorted echo of the Bulk Data to a separate file

While learning to use NX Nastran by using small models, you should set ECHO to BOTH. The resulting listings in the .f06 file does not occupy very much space and having the model information available can be useful. However, when the models become very large it may be best to use ECHO = PUNCH or even ECHO = NONE.

Printing Bulk Data Using the ECHO Case Control Command

You can use the ECHO Case Control command to control how the Bulk Data section is printed to the output file. For example, you can use ECHO to specify whether you want NX Nastran to print the Bulk Data in sorted or unsorted format. For example, to echo the Bulk Data in the unsorted form, you use the following ECHO command:

```
ECHO = UNSORT
```

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

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

See also

- “ECHO” in the *NX Nastran Quick Reference Guide*

Types of Output

There are two main types of output that you can request in NX Nastran:

- Grid point output
- Element output

In a small or medium-sized model, it may be best to simply request output for all grid points or elements. However, in a large model, you may want to be more selective.

Grid Point Output (DISP, SPCF, OLOAD, GPFORCE)

Requests for output quantities that occur at grid points include the following (n is a SET ID, and ALL requests that all quantities be printed):

DISP	Requests displacements for grid points. In general, you should use DISP=ALL as displacement output takes up very little space.
SPCFORCE	Requests forces of single point constraint (SPCFORCES). In general, you should use SPCF = ALL so you can examine reaction forces during the model validation process.

LOAD	Requests the set of applied loads in static analysis.
GPFORCE	Generates a table containing a grid point force balance at the selected grid points. This is useful for determining load paths, contributions of applied loads to element response, and the effects of initial thermal strain. Contributors to the grid point force balance table include applied loads, SPC forces, and element elastic forces.

Element Output (STRESS, FORCE, STRAIN, ESE)

Requests for element-based output quantities include the following:

STRESS	Requests element stresses for a set of structural elements
FORCE	Requests element forces to be calculated for a set of structural elements
STRAIN	Requests the strains for a set of plate elements
ESE	Requests the strain energy for a set of elements

CQUAD4 stresses, strains, and forces are available at corner grid points, with output at the element center the default. CQUAD4 center and corner output is obtained using the STRESS, STRAIN, and FORCE Case Control commands as follows:

```
STRESS(CORNER) = {ALL or n}
STRAIN(CORNER) = {ALL or n}
FORCE(CORNER) = {ALL or n}
```

Where { } indicates that a choice of ALL or n is mandatory, but the braces are not included.

Only one type of element output (center or corner) is supported per run..

Selecting the Destination File for the Output

For each type of results you request, you can specify where you want NX Nastran to print the results.

- The PRINT option provides printed results in the .f06 output file. It is the default option for most result types.
- The PUNCH option prints the results in the ASCII format .pch file using a “punch” format (80 column width per line). Writing results to the .pch file is useful if you want to export your results to other programs.
- The PLOT option instructs NX Nastran to calculate the requested results, but not print them. In general, you use this option when you want to view the results as plots or examine them with a post processing program. This option is also useful when you're working with large models where the quantity of printed output would be excessive but you still need NX Nastran to recover the specified data for post processing.

Formatting Printed Output

You can include up to three title lines per page of NX Nastran output with the TITLE, SUBTITLE, and LABEL Case Control commands. You can use any character string. Titles are optional but highly recommended as they can help you distinguish between your jobs.

Example

These Case Control commands

```
TITLE=SIMPLY SUPPORTED BEAM
SUBTITLE=WITH CONCENTRATED FORCE
LABEL=NX NASTRAN USER'S GUIDE
```

produce the following headings on each page of output:

```
SIMPLY SUPPORTED BEAM              JULY 10, 2001 NX NASTRAN 7/10/2003 PAGE 12
WITH CONCENTRATED FORCE
NX NASTRAN GETTING STARTED USER'S GUIDE
```

POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R			
			T2	T3		R1	R2	R3	
1	G	0.0	0.0	0.0	0.0	0.0	0.0	-2.221112E-04	
2	G	0.0	-1.943473E-03	0.0	0.0	0.0	0.0	-1.388195E-04	
3	G	0.0	-2.221112E-03	0.0	0.0	0.0	0.0	1.110556E-04	
4	G	0.0	0.0	0.0	0.0	0.0	0.0	2.776390E-04	

Using the SET Command to Limit Output

In many cases, particularly in larger models, you may want to limit the output to a particular set of grid points or elements. In NX Nastran, you can use the SET Case Control command to identify a group of grid points and/or elements as a specific set ID.

For example, suppose you want to output all of the grid point displacements for Subcase 1 and the grid point displacements for grid points 3 and 4 for Subcase 2. In addition, you want the element force output for element 3 for Subcase 1 and elements 3 and 4 for Subcase 2. A Case Control section that meets these requirements is shown in [Listing 3-4](#).

```

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

```

Listing 3-4. Controlling Output with Set Commands

A Case Control set is a collection of grid point IDs or element IDs for use in output requests. Case Control sets are used to obtain output for a selected portion of the model. Case Control sets are defined with the SET command according to the following formats:

```
SET n = ALL
```

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

where n is the set identification number and i₁, i₂, i₃, etc., are entity identification numbers, e.g. grid point numbers.

Example

Consider the Case Control Section shown below:

```

CEND
TITLE=OUTPUT SELECTION EXAMPLE
SUBTITLE=ILLUSTRATES USE OF SETS
LOAD=15
SET 1=3,4,7,9,11
SET 5=2,9,15 THRU 21,23
DISP=1
FORCE=1
STRESS=5
BEGIN BULK

```

DISP is a grid point output quantity, so displacements for grid points 3, 4, 7, 9, and 11 will be printed. FORCE is an element output quantity, so forces for elements 3, 4, 7, 9, and 11 will be printed. Note that grid quantities and element quantities can share the same set, since a set is simply a list of numbers. STRESS is an element quantity, so the stresses in elements 2, 9, 15 through 21 (inclusive), and 23 will be calculated and printed.

3.6 Bulk Data Section

The Bulk Data section always follows the Case Control section and begins with the BEGIN BULK delimiter. In NX Nastran, Bulk Data entries contain all the information necessary to define the finite element model, such as:

- Geometry
- Coordinate systems
- Finite elements
- Element properties
- Loads
- Boundary conditions
- Material properties

The Bulk Data section generally constitutes the majority of the content of the input file. The order of Bulk Data entries is not critical (continuations without continuation identifiers are an exception to this rule and are discussed later).

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

If you leave a data field blank, the software uses the appropriate default. Any applicable defaults for a particular field are listed in the *NX Nastran Quick Reference Guide*. There are certain situations that allow either an integer or a real number in a data field. However, in these situations, whether you specify an integer or a real number in a data field can affect the problem's solution.

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

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

Figure 3-3. A Bulk Data Example: the GRID Entry

See also

- [“Bulk Data Entries”](#) in the *NX Nastran Quick Reference Guide*

Bulk Data Section Format Rules

You must begin the Bulk Data section with the BEGIN BULK delimiter. The last entry must be the ENDDATA delimiter. The BEGIN BULK entry is in free field format. The ENDDATA delimiter must begin in Column 1 or Column 2. The ENDDATA delimiter also signifies the end of the NX Nastran input file.

Note: You must include both BEGIN BULK and ENDDATA in your input file, even if you are not introducing any new Bulk Data is being introduced into the problem or if all of the Bulk Data is coming from an alternate source, such as user-generated input.

NX Nastran Bulk Data contains 10 fields per input data entry:

- The first field contains the character name of the Bulk Data item, such as GRID, CBAR, MAT1.
- The second through ninth fields contain data input information for the Bulk Data entry.
- The tenth field never contains data. It is reserved for entry continuation information, if applicable.

Sorting Bulk Data

You can enter Bulk Data entries in any order since NX Nastran automatically sorts them (using the XSORT module) prior to the execution of the Input File Processor (IFP) module.

However, for very large problems, this sorting process can be CPU intensive. If you anticipate submitting the same input file multiple times, you should use the ECHO=PUNCH Case Control command to generate a sorted Bulk Data file in the first analysis. Then, you can use the sorted Bulk Data input for subsequent analyses.

Note: Alternatively, the sorted Bulk Data is stored on the database and is available using the Restart feature.

See also

- “XSORT” in the *NX Nastran DMAP Programmer's Guide*
- “IFP” in the *NX Nastran DMAP Programmer's Guide*

Storing Bulk Data in the Database for Restarts

To minimize the handling of a large number of entries, NX Nastran solution sequences let you store the Bulk Data in the database. That way, you can easily modify it on subsequent runs.

For any initial analysis (also referred to as a “cold start”), you must submit the entire Bulk Data section of the input file. For subsequent analyses, if you save the database from any previous run, the Bulk Data exists in the database in sorted form where you can modify it and reuse it in a restart analysis.

When you perform a restart, the software adds the Bulk Data entries contained in the input file to the Bulk Data stored in the database. You can use a “/” entry to remove entries from the database.

See also

- “Restarts” in the *NX Nastran User's Guide*

Understanding the Relationship Between Bulk Data Entries and Case Control Commands

Most of the entries in the Bulk Data section do not need to be called out by a Case Control command to be included in the model. However, for loads and constraints, the Bulk Data and Case Control sections of the input file are carefully interrelated.

- In the Bulk Data section, you specify actual load and constraint entries.
- In the Case Control section, you specify which of the load and constraint entries you defined in the Bulk Data section to use in a given analysis.

In other words, NX Nastran includes only the load and constraint entries you define in the Bulk Data section if they are specified in the Case Control section. Therefore, you can have load and constraint entries in the Bulk Data section that are not specified in the Case Control. However, those entries will not be used during the analysis.

3.7 Using Parameters

Each NX Nastran solution sequence is designed with a specific set of default values and solution paths. In the majority of the cases, the default values are the ones best suited for your model. There are occasions, however, when you may want to deviate from the default values. Several methods are available in NX Nastran to perform this task: DMAP (Direct Matrix Abstraction Program) modules, NASTRAN statements, and parameters.

Using parameters is the most straightforward method to change defaults in NX Nastran. Depending on the parameter, you can specify it either as a PARAM Bulk Data entry and/or a PARAM Case Control command. The format of each is as follows:

```
PARAM,xname,xvalue
```

where xname and xvalue are the name and value of the PARAMeter, respectively.

```
PARAM,WTMASS,0.00259
```

For example, with PARAM,WTMASS, the software multiplies the terms of the structural mass matrix by the value of this parameter.

In NX Nastran, parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. You can specify values for parameters using either the PARAM Bulk Data entry or the PARAM Case Control command.

See also

- “**PARAM**” in the *NX Nastran Quick Reference Guide* (for the PARAM Case Control command)
- “**PARAM**” in the *NX Nastran Quick Reference Guide* (for the PARAM bulk data entry)
- “**Parameters**” in the *NX Nastran Quick Reference Guide* (for a complete description of all parameters)
- *NX Nastran DMAP Programmer's Guide* (for a description of the manner in which NX Nastran processes parameters)

Parameters in the Superelement Solution Sequences

The specification of PARAMeters differs between the rigid formats and the superelement solution sequences. These differences require different user action in placing the PARAM entries in the NX Nastran section, and in changing PARAMeter values on restarts.

The program module reads the Case Control section for PARAM entries. If PARAM entries exist, the VPS table is updated with the values found in the Case Control section. The PARAM values are updated at the beginning of the generation, assembly, reduction, and the data recovery loops of the superelement solution sequences. This allows the values of some user-supplied parameters to change for each superelement during the course of one run.

In the nonlinear solution sequences the PARAM values are updated at the beginning of the iteration loop. This allows the values of some user-set parameters to vary between subcases. These parameters may appear in either the Case Control and/or Bulk Data section. Other PARAMeters must appear in the Bulk Data section. If they appear in the Case Control section, they will not cause a preface error, but will be ignored. Other parameters may be reset to default values during the course of the run. This type should be set only in the subcase of the Case Control section.

The PARAM Case Control command is treated in the same manner as constraint and output commands. Parameters set above the subcase level are effective in every subcase. If there is also a parameter value set in the subcase, it takes precedence. Only one value of a parameter may be current at any time during the execution of the run. These rules imply an order of precedence if several forms are used in the same run. The rules are illustrated in the following example.

A partial subcase structure is listed on the following page. The subcases are listed in ascending superelement number order. PARAM, GRDPNT is varied between them.

See also

- The *NX Nastran Quick Reference Guide* (for parameter applicability)

Subcase Structure, Sorted by SEID

```
SUBCASE 1
SUPER = 1
...
SUBCASE 2
SUPER = 2
PARAM,GRDPNT,100
...
SUBCASE 3
SUPER = 3
PARAM,GRDPNT,200
...
SUBCASE 4
SUPER = 4
...
SUBCASE 100 $ RESIDUAL STRUCTURE
PARAM,GRDPNT,-1
...
BEGIN BULK
PARAM,GRDPNT,0
```

Processing order	Value of GRDPNT
SEID	
1	0

Processing order	Value of GRDPNT
2	100
3	200
4	0
0	-1

The Bulk Data entries are processed at the beginning of the run. Any PARAM entries in the Bulk Data section set the value of that parameter until it is changed by a Case Control command.

- Superelement 1 is processed first. Since it has no PARAM,GRDPNT entry the value of 0, as set by the Bulk Data entry PARAM,GRDPNT,0 is used.
- The value for superelement 2 is set at 100 by a Case Control command.
- Superelement 4 is processed next. Its subcase contains no PARAM entry, so that the value of the parameter that remains is set by the bulk data.
- The values for superelement 3 and the residual structure are set by Case Control commands.

An alternate method of parameter specification is to set the value most used above the subcase level, and exceptional values within the subcase. For example, if PARAM,GRDPNT,0 were placed above the subcase level and the values of 100, 200, and -1 placed in subcases 2, 3 and 100, respectively, the value used in each superelement will be the same as described above, regardless of processing order.

For the unstructured solution sequences, NX Nastran does not store the VPS table in the database. Therefore, it is not recovered on restarts. Therefore, parameters set by the user must be in the NX Nastran section if the default values are not wanted.

Chapter 4: NX Nastran Output Files

- *Overview of the NX Nastran Output Files*
- *Understanding the .f06 File*
- *Understanding the .f04 File*
- *Understanding the .log File*
- *Understanding the .plt and .pch Files*
- *Understanding the .DBALL and .MASTER Files*
- *Understanding the .xdb and .op2 Files*

4.1 Overview of the NX Nastran Output Files

When you submit an input file to NX Nastran for execution, the software generates a number of different output files. Some files are generated automatically, while other files are generated only by specific requests you make in the input file or on the command line when you submit the job. The following table shows the different types of NX Nastran output files.

Table 4-1. Types of Files Output by NX Nastran			
File Extension	File Format	How Created	Brief Description
.f06	ASCII	automatically	Main output file with printed output, such as displacements and stresses.
.f04	ASCII	automatically	Provides history of assigned files, disk space usage, and modules used during the analysis.
.log	ASCII	automatically	Gives a summary of the command line options used and the execution links
.DBALL	binary	automatically	A database containing the input file, assembled matrices, and solutions. Used also for restarting the run for additional analysis or output.
.MASTER	binary	automatically	Contains the master directory of the files used by the run and the physical location of those files on your system. Needed for a restart.
.plt	binary	by request	Contains the plot information requested by the plotter commands specified in input file.
.pch	ASCII	by request	Contains the punch output as requested in the input file.
.op2	binary	by request	Graphics database file used for postprocessing.
.xdb	binary	by request	Graphics database file used for postprocessing.

In addition to the files listed above, the software also generates several temporary (scratch) files during the analysis which are automatically deleted when the solve completes.

Note: By default, the software writes the .f06, .f04, and .log files separately. However, you can use the “append” keyword (append=yes) on the command line when you run your job to have the software combine them into a single file with a .out extension.

The following sections describe the different output file types in more detail.

See also

- “Automatic Deletion of Scratch Data Blocks” in the *NX Nastran DMAP Programmer's Guide*
- “Append” in the *NX Nastran Installation and Operations Guide*

4.2 Understanding the .f06 File

The output file that you will use most frequently is the .f06 file. The software writes this file to FORTRAN unit 6. The .f06 file contains all the requested results from your analysis, such as the displacements, stresses, and element forces, as well as any diagnostic messages. The information in the .f06 file is critical for model checkout and debugging.

The output from data recovery and plot modules is all optional and its selection is controlled by commands in the Case Control section.

Although most of the solution sequence output is optional and requested in the Case Control Section, some of the printer output is automatic or under control of DIAG statements in the Executive Control Section.

See also

- “Flushing .f04 and .f06 Output to Disk” in the *NX Nastran Installation and Operations Guide*

Controlling Printed Output

In NX Nastran, there are two different formats in which the software prints results: SORT1 and SORT2.

Understanding the SORT1 Format

With the SORT1 format, the software presents the analysis outputs as a tabular listing of the selected output for each grid point in each individual subcase. With SORT1, the software prints the output for each loading condition on any page. This is the default for static analysis, frequency response, steady state heat transfer, real and complex eigenvalue analysis, flutter analysis, and buckling analysis.

If you select SORT1 in a transient solution for one or more of the commands listed below, then the software outputs all remaining commands in the input file in SORT1 format:

ACCE, DISP, ENTH, FORC, HDOT, MPCF, OLOA, SPCF, STRA, STRE, VELO

Understanding the SORT2 Format

With the SORT2 format, the software presents the analysis output as a tabular listing of the subcases for each selected output option. With SORT2, the software prints the output for each grid point or element on a new page. Consequently, the SORT2 option can produce a large number of output pages. Therefore, SORT2 should generally be used only when you're performing dynamic solutions.

The SORT2 format is the default output format in transient response analysis (structural and heat transfer). The SORT2 option isn't available when you're performing real eigenvalue (including buckling) analyses, complex eigenvalue analyses, or flutter analyses.

If you select SORT2 in a frequency response solution for one or more of commands listed below, then the software outputs all remaining commands in the input file in SORT2 format:

ACCE, DISP, FORC, MPCF, OLOA, SPCF, STRA, STRE, VELO

Controlling General Output Options

The printer output is designed for 132 characters per line, with the lines per page controlled by the LINE command in the Case Control Section. The default is LINE = 50 for 11-inch paper.

Optional titles are printed at the top of each page from information in the Case Control Section. You can define these titles at the subcase level. All pages of the output are automatically dated and numbered.

A few printer output items are under the control of PARAM Bulk Data entries.

The following output is either automatically or optionally provided during execution:

- NX Nastran title page – automatic
- File Management Section echo – automatic if statements are present
- Executive Control Section echo – automatic
- Case Control Section echo – automatic
- Unsorted Bulk Data Section echo – optional, selected in the Case Control Section
- Sorted Bulk Data Section echo – automatic, unless suppressed in the Case Control Section

By default, the output file contains an echo (listing) of the input file with the Bulk Data sorted alphabetically as shown in [Figure 4-1](#).


```

ID LINEAR,TRUSS1
  SOL 101
  TIME 2
  CEND
LINEAR STATICS USER'S GUIDE                JUNE 18, 2004 NX NASTRAN 6/13/04 PAGE 3
  TRUSS STRUCTURE
  POINT LOAD AT GRID POINT 4

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

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

NX NASTRAN USER'S GUIDE                JUNE 18, 2004 NX NASTRAN 6/13/04 PAGE 4
  TRUSS STRUCTURE
  POINT LOAD AT GRID POINT 4

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

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

NX NASTRAN USER'S GUIDE                JUNE 18, 2004 NX NASTRAN 6/13/04 PAGE 5
  TRUSS STRUCTURE
  POINT LOAD AT GRID POINT 4
USER INFORMATION MESSAGE
ORIGIN OF ASSEMBLY BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
RESULTANTS ABOUT ORIGIN OF ASSEMBLY BASIC COORDINATE SYSTEM IN ASSEMBLY BASIC SYSTEM COORDINATES.
                                OLOAD   RESULTANT
                                T1      T2      T3      R1      R2      R3
  1      0.0000000E+00 -1.0000000E+03  0.0000000E+00  0.0000000E+00  0.0000000E+00 -6.0000000E+05

LINEAR STATICS USER'S GUIDE                JUNE 18, 2004 NX NASTRAN 6/13/04 PAGE 6
  TRUSS STRUCTURE
  POINT LOAD AT GRID POINT 4
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS

  1      -5.2155330E-15      6.4518387E+01

```

Figure 4-1. truss1.f06 Output File (Continued)

Figure 4-1. truss1.f06 Output File

LINEAR STATICS USER'S GUIDE				JUNE 18, 2004 NX NASTRAN				6/13/04 PAGE		7
TRUSS STRUCTURE										
POINT LOAD AT GRID POINT 4				D I S P L A C E M E N T V E C T O R						
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3			
1	G	.0	.0	.0	.0	.0	.0			
2	G	.0	.0	.0	.0	.0	.0			
3	G	1.247578E-02	-1.285377E-01	.0	.0	.0	.0			
4	G	-1.252422E-02	-1.290368E-01	.0	.0	.0	.0			
NX NASTRAN USER'S GUIDE				JUNE 18, 2004 NX NASTRAN				6/13/04 PAGE		8
TRUSS STRUCTURE										
POINT LOAD AT GRID POINT 4				F O R C E S I N R O D E L E M E N T S (C R O D)						
ELEMENT ID.	AXIAL FORCE	TORQUE	ELEMENT ID.	AXIAL FORCE	TORQUE					
1	2.495156E+03	.0	2	2.554449E+03	.0					
3	-2.544570E+03	.0	4	-2.504844E+03	.0					
5	4.990313E+02	.0								
NX NASTRAN USER'S GUIDE				JUNE 18, 2004 NX NASTRAN				6/13/04 PAGE		9
TRUSS STRUCTURE										
POINT LOAD AT GRID POINT 4				S T R E S S E S I N R O D E L E M E N T S (C R O D)						
ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	
1	6.237891E+02		.0		2	6.386123E+02		.0		
3	-6.361426E+02		.0		4	-6.262109E+02		.0		
5	1.247578E+02		.0							

See also

- “Parameters” in the *NX Nastran Quick Reference Guide*

Understanding the General Analysis Output**OLOAD Resultant**

After the input file echo, the software prints the “OLOAD RESULTANT.” This shows the resultant of the loads about the origin of the basic coordinate system in this case.

User Information Message 5293

Next is the USER INFORMATION MESSAGE (UIM) 5293 which shows the epsilon and external work for the run. The epsilon is a measure of the numeric conditioning of the model, while the external work is the work due to the applied loads. Both of these values are useful for model checkout.

In the truss example, output is requested for displacements, element forces, and element stresses. The displacement request is the only grid point output. Element forces and stresses are element output.

The last summary is the maximum of the applied loads.

After the summaries, the displacement for all of the grid points, the element forces, and the element stresses is printed as requested in the Case Control Section.

For static analysis of the truss model, just one applied load is used. You can, however, specify multiple loading conditions to be analyzed in a single input file if desired. If you do specify multiple loading conditions, the OLOAD RESULTANT and EXTERNAL WORK are shown for each selected loading conditions. The external work is calculated as $1/2 \{P\}^T [U]$. In the case of enforced

displacement, the equivalent load is calculated as the constraint force required to impose the enforced displacement. The external work printout in this case is meaningless.

See also

- “[Model Verification](#)” in the *NX Nastran User's Guide*

Accuracy Diagnostic Information for Global Iterative Solver

When you're using NX Nastran's global iterative solver, the .f06 file also contains important accuracy diagnostic information.

See also

- “Accuracy Diagnostics” in the *NX Nastran Numerical Methods User's Guide*

Grid Point Singularity Table

If singularities remain in the stiffness matrix at the grid point level, the software automatically outputs a Grid Point Singularity Table (GPST) after the Grid Point Singularity Processor (GPSP) executes. The table lists singular degrees-of-freedom in the global coordinate system. You can have the software automatically constrain singular degrees-of-freedom using the PARAM,AUTOSPC,YES option.

Note: You can print any of the matrices or tables that are prepared by the functional modules using selected utility modules described in the *NX Nastran DMAP Programmer's Guide*. These utility modules can be scheduled at any point in a solution sequence by using the alter feature. In general, they should be scheduled immediately after the functional module that generates the table or matrix to be printed. However, you should check the calling sequence for the utility module, in order to be certain that all required inputs have been generated prior to this point.

See also

- “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#)” in the *NX Nastran User's Guide*
- “[Understanding the Grid Point Singularity Table](#)” in the *NX Nastran User's Guide*
- “AUTOSPC” in the *NX Nastran Quick Reference Guide*

Understanding Analysis Type Specific Output

The following descriptions relate to automatic output that is associated with particular types of solution sequences.

Detailed information on the force and stress output available for each element type is given in the *NX Nastran Element Library*.

Real Eigenvalue Analysis

Each eigenvalue is identified with a mode number determined by sorting the eigenvalues by their magnitude. The following summary is also automatically printed for all methods:

- Mode number

- Extraction order (this is irrelevant under the Lanczos method)
- Eigenvalue
- Frequency (radians)
- Frequency (Hz)
- Generalized mass
- Generalized stiffness

For all methods, if the orthogonality criterion fails, then the software issues User Warning Message 3034.

For the Lanczos method, the following additional summary content is provided:

- Block size used
- Number of decompositions
- Number of roots found
- Number of (block) solves required
- One of the following termination messages:
 - o Required number of eigenvalues found
 - o Not all the roots found within the range
 - o All eigenvalues found within the range
 - o Insufficient time to find more eigenvalues

Complex Eigenvalue Analysis

Each complex eigenvalue is identified with a root number determined by sorting the complex eigenvalues by magnitude. The following summary of the complex eigenvalues extracted is automatically printed for all methods:

- Root number
- Extraction order (this is irrelevant under the Lanczos method)
- Real and imaginary parts of the eigenvalue
- The coefficients f (frequency) and g (damping coefficient) in the following representation of the eigenvalue ($p = \alpha + i\omega$):

$$p = 2\pi f(i - 1/2g)$$

Equation 4-1.

Therefore,

$$f = \left| \frac{\omega}{2\pi} \right|$$

Equation 4-2.

$$g = \frac{-2\alpha}{|\omega|} = 2 \frac{C}{C_{cr}}$$

Equation 4-3.

If you use the complex Lanczos method, the software also issues User Information Messages 5444, 5445, and 5453.

If you use the Determinant method, the software also prints:

- Number of eigenvalues extracted
- Number of passes through starting points.
- Number of criteria changes
- Number of starting point moves
- Number of triangular decompositions
- Number of failures to iterate to a root
- Number of predictions outside region
- The reason for termination is indicated by one of the following integers:

Reason Number	Description
1	The number of roots desired have been found
2	All predictions for eigenvalues are outside the regions specified
3	Insufficient time to find another root
4	Matrix is singular at the first three starting points

- Swept determinant functions for each starting point

Superelement Analysis

- Superelement mapping information that lists the membership of grid points and elements in the superelements, the order in which the superelements will be processed, and timing and space estimates for each superelement

- A page header that identifies the superelement for which the printed information on the page pertains
- Matrices wherein mechanisms are identified
- Resultants of external loads
- Maximum applied loads
- Maximum displacements
- Maximum SPC forces

For SOL 101, if PARAM,INREL,-1 or PARAM,INREL,-2, the following data is printed automatically:

```
INTERMEDIATE MATRIX...QRR
```

This is the rigid body mass matrix of the total structure, including superelements. It is presented as a 6×6 matrix, measured at the reference grid point, in its global coordinates. Masses on scalar points are ignored.

```
INTERMEDIATE MATRIX...QRL
```

This is the resultant of the applied loads, measured at the reference point. There is one column for each loading condition.

```
INTERMEDIATE MATRIX...URACCEL
```

This is the user-supplied rigid body acceleration, input on the DMIG,UACCEL entry. This output does not appear if the entry is not present.

```
INTERMEDIATE MATRIX...URA
```

This is the rigid body acceleration matrix, computed from the applied loads.

4.3 Understanding the .f04 File

The .f04 file contains information that helps you monitor and tune the performance of your analysis job. For example, the .f04 file contains:

- A summary of physical file information that describes the files used for DBsets
- A map showing memory allocation
- A “day log” that provides a DMAP execution summary
- A listing of memory and disk usage statistics
- A listing of database usage statistics that summarizes the I/O activity for the DBsets
- A summary of physical file I/O activity for each database file
- Module execution and statistical information
- Other optional diagnostic information as requested by the DIAG executive control command in the input file

If you're performing an analysis using NX Nastran's global iterative solver, the .f04 file also contains important performance diagnostic information about the solver.

The software writes the .f04 file to FORTRAN unit 4.

See also

- “Flushing .f04 and .f06 Output to Disk” in the *NX Nastran Installation and Operations Guide*
- “Performance Diagnostics” in the *NX Nastran Numerical Methods User's Guide*

General Information

The .f04 file begins with a section that lists general information about the run.

- DAY TIME indicates the time of day as the run progresses.
- ELAPSED indicates the number of elapsed “wall clock” minutes:seconds at the beginning on the job. This number may vary greatly depending on the system load in a time-sharing environment.
- I/O SEC is an internally generated measure of I/O. This measure is a count of the number of BUFFSIZE blocks transferred, divided by the value of SYSTEM(84) (the estimated rate of number of blocks moved per second). This number doesn't include I/O for loading the executable file or I/O for FORTRAN files.
- DEL I/O is the I/O seconds spent in the previous DMAP statement.
- CPU SEC shows the number of CPU seconds that had elapsed when this line was printed.

In general, you should examine the above entries and look for large jumps in the CPU SEC number. For example, the cost for “formatting” the output (modules SDR2 and OFP) may be greater than the cost of generating the numbers. In such situations, simple techniques such as reducing output requests may save more time than other more well-known techniques such as sequencing, OMITs, better matrix methods, etc. In short, you should be aware of the expensive operations on each large run.

- DEL CPU is the CPU time spent in the previous DMAP statement.

Summary of Physical File Information

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

S U M M A R Y O F P H Y S I C A L F I L E I N F O R M A T I O N			
ASSIGNED PHYSICAL FILE NAME	RECL (BYTES)	MODE	FLAGS
-----	-----	-----	-----
/tmp/65872_57.SCRATCH	8192	R/W	L
/tmp/65872_57.OBJSCR	8192	R/W	L
/tmp/65872_57.MASTER	8192	R/W	L
/tmp/65872_57.DBALL	8192	R/W	L
/tmp/65872_57.DBALL2	8192	R/W	L
/tmp/65872_57.SCR300	8192	R/W	L
/nxn			
r			
/aix/SSS.MASTERA	8192	R/O	L
/nxn			
r			

```

/aix/SSS.MSCOBJ                                8192    R/O    L
FLAG VALUES ARE --
F   FFIO INTERFACE USED TO PROCESS FILE
H   HPIO INTERFACE USED TO PROCESS FILE
L   FILE HAS BEEN LOCKED
M   FILE MAPPING USED TO PROCESS FILE
R   FILE BEING ACCESSED IN 'RAW' MODE
** PHYSICAL FILES LARGER THAN 2GB FILES ARE NOT SUPPORTED ON THIS PLATFORM

```

In this summary:

- ASSIGNED PHYSICAL FILENAME is the physical FILENAME with any symbols translated.
- RECL is the record length in bytes.
- MODE is the file access mode.
- R/W is read-write mode.
- R/O is read-only mode.
- The FLAGS column contains various letters depending on the capabilities of the platform and user requests. The text below the table indicates possible flag values for your specific platform.
- BUFFSIZE lists the buffer size of each DBset. This buffer size may be defined by the INIT command for each DBset (File Management Statement).
- CLUSTER SIZE lists the unit of allocation for GINO and executive blocks. This unit may be defined on the INIT command.
- The TIME STAMP column lists the time the file was created. The format is YYMMDDHHMMSS.

In this example, an INIT statement was used to create the DBALL DBSet with two files using the logical names DBALL and DBALL2.

Below the summary is a message that indicates whether large files are available on the platform on which you're running.

Memory Allocation Map

The next section of the file is a map of memory allocation.

```

** MASTER DIRECTORIES ARE LOADED IN MEMORY.
USER OPENCORE (HICORE)      =      3804612  WORDS
EXECUTIVE SYSTEM WORK AREA  =       78605   WORDS
MASTER(RAM)                 =       30000   WORDS
SCRATCH(MEM) AREA           =       204900  WORDS (      100 BUFFERS)
BUFFER POOL AREA (GINO/EXEC) =       76183  WORDS (       37 BUFFERS)
TOTAL NX NASTRAN MEMORY LIMIT =      4194300  WORDS

```

In this memory map:

- USER OPENCORE is the amount of memory available to the module for computation purposes.
- EXECUTIVE SYSTEM WORK AREA is the space reserved for the executive system.
- MASTER(RAM) is the space reserved to cache datablocks from the MASTER DBSet.

- SCRATCH(MEM) AREA is the space reserved to cache datablocks from the SCRATCH and SCR300 DBSets.
- BUFFER POOL AREA is the space reserved for the buffer pool.
- TOTAL NX Nastran MEMORY LIMIT is the total space allocated to NX Nastran's open core using the "memory" keyword.

See also

- "Managing Memory" in the *NX Nastran Installation and Operations Guide*

Day Log

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

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

In the Day Log:

- DAY TIME is the time of day of the entry.
- ELAPSED is the elapsed time since the start of the job.
- I/O MB is the megabytes of I/O to the databases since the start of the job
- DEL_MB is the delta I/O since the previous entry.
- CPU SEC is the total CPU seconds since the start of the job.
- DEL_CPU is the delta CPU since the previous entry.
- SUB_DMAP/DMAP_MODULE" indicates the DMAP statement being executed.
- MESSAGES are any messages issued by the module.
- BEGN and END indicate the start and end of the module or subDMAP.

The number that precedes the module name on lines that terminate with BEGN or END is the DMAP statement number. This number corresponds to the statement number in the DMAP listing for the solution sequence being used. The DMAP sequence listing will be printed as a part of the output if you include either DIAG 14 or COMPILER LIST in the Executive Control section of your input file.

Understanding the I/O MB Column

- The software computes the "I/O MB" value by multiplying SYSTEM(85), which is incremented by one for each GINO I/O, by BUFFSIZE. This value loses accuracy if the DBSets don't have the same BUFFSIZE.
- If SYSTEM(84) is set to 0, the "I/O MB" column will be the number of GINO I/Os.

- The software scales the “I/O MB” column by gigabytes. It appends a “G” after each number if the value is greater than or equal to 100 000.

User Information Messages 4157 and 6439

The UIM 4157 text provides decomposition estimates when the preface of the decomposition module completes. The format of this message varies slightly depending on whether you selected either sparse or parallel methods.

This counterpart to UIM 4157 is UIM 6439, which provides actual information from the completed decomposition process. These two messages are interspersed within the Day Log at each decomposition.

The following is an example from a sparse decomposition:

```
*** USER INFORMATION MESSAGE 4157 (DFMSA) ---PARAMETERS FOR SPARSE DECOMPOSITION OF DATA BLOCK KLL (TYPE=BDP) FOLLOW
      MATRIX SIZE =          64 ROWS          NUMBER OF NONZEROES =          260 TERMS
      NUMBER OF ZERO COLUMNS =          0          NUMBER OF ZERO DIAGONAL TERMS =          0
      CPU TIME ESTIMATE =          0 SEC          I/O TIME ESTIMATE =          0 SEC
      MINIMUM MEMORY REQUIREMENT =          20 K WORDS          MEMORY AVAILABLE =          3804 K WORDS
      MEMORY REQR'D TO AVOID SPILL =          30 K WORDS
      EST. INTEGER WORDS IN FACTOR =          1 K WORDS          EST. NONZERO TERMS =          1 K TERMS
      ESTIMATED MAXIMUM FRONT SIZE =          11 TERMS          RANK OF UPDATE =          16
*** USER INFORMATION MESSAGE 6439 (DFMSA) ---ACTUAL MEMORY AND DISK SPACE REQUIREMENTS FOR SPARSE SYM. DECOMPOSITION
      SPARSE DECOMP MEMORY USED =          30 K WORDS          MAXIMUM FRONT SIZE =          11 TERMS
      INTEGER WORDS IN FACTOR =          1 K WORDS          NONZERO TERMS IN FACTOR =          1 K TERMS
      SPARSE DECOMP SUGGESTED MEMORY =          30 K WORDS
```

In UIM 4157:

- MINIMUM MEMORY REQUIREMENT is an estimate of the open core memory that will allow the decomposition to run, but with heavy spilling to disk.
- MEMORY REQR'D TO AVOID SPILL is the estimate of memory necessary for the decomposition to run in “in core”, without spilling to disk.

Note: These two values represent memory requirement extremes. The amount of memory necessary for optimal CPU performance is somewhere between the two.

- ESTIMATED MAXIMUM FRONT SIZE is a function of the model and affects the memory estimates described above.
 - MINIMUM MEMORY REQUIREMENT is a function of the front size.
 - MEMORY REQR'D TO AVOID SPILL is a function of the square of the front size.
- NUMBER OF NONZEROES is the size of the input matrix, multiply this value by 8 to estimate the size of the input file in bytes.
- The sum of EST. INTEGER WORDS IN FACTOR and EST. NONZERO TERMS is the size of the output matrix.

Note: Multiply the integer value by 4 and the nonzero value by 8 to estimate the size of the output file in bytes.

- RANK OF UPDATE is the number of rows that will be simultaneously updated during the decomposition. You can set by either the “rank” keyword or SYSTEM(205).

Note

Setting SYSTEM(69)=64 causes NX Nastran to terminate after printing UIM 4157. This can be useful for determining a job's memory and disk space requirements.

In UIM 6439:

- SPARSE DECOMP MEMORY USED states the actual memory used in the decomposition process.
- SPARSE DECOMP SUGGESTED MEMORY is based on the execution of the module. Using this value will result in optimal throughput performance.

Memory and Disk Usage Statistics

NX Nastran writes the memory and disk usage statistic tables after the job completes. These tables indicate the maximum memory used by any sparse numerical module and the maximum disk used by any module during the job. For example:

SPARSE SOLUTION MODULES					MAXIMUM DISK USAGE				
HIWATER	DAY TIME	SUB DMAP	DMAP		HIWATER	DAY TIME	SUB DMAP	DMAP	
(WORDS)		NAME	MODULE		(MB)		NAME	MODULE	
517786	04:35:44	SEKRRS	18 DCMF		15.625	04:35:48	SESTATIC	186 EXIT	

In the sparse solution modules table:

- HIWATER WORDS is the maximum amount of open core used by certain sparse numerical modules.
- DAY_TIME is the time of day the module ran.
- SUB_DMAP NAME is the name of the SUBDmap.
- DMAP MODULE indicates the line number and module name that made the maximum request.

In the maximum disk usage table:

- HIWATER (MB) is the maximum amount of disk space used by any module.
- DAY_TIME is the time of day the module ran.
- SUB_DMAP NAME is the name of the SUBDmap.
- DMAP MODULE indicates the line number and module name that made the maximum request.

Database Usage Statistics

These statistics, provided in table format, summarize the I/O activity for the DBSets.

*** DATABASE USAGE STATISTICS ***

LOGICAL DBSETS					DBSET FILES				
DBSET	ALLOCATED (BLOCKS)	BLOCKSIZE (WORDS)	USED (BLOCKS)	USED %	FILE	ALLOCATED (BLOCKS)	HIWATER (BLOCKS)	HIWATER (MB)	I/O TRANSFERRED (GB)
MASTER	5000	2048	143	2.86	MASTER	5000	143	1.117	.010
DBALL	250000	2048	9	.00	DBALL	250000	9	.070	.000
					DBALL2	300	1	.008	.000
OBJSCR	5000	2048	121	2.42	OBJSCR	5000	121	.945	.003
SCRATCH	500100	2048	19	.00	(MEMFILE	100	81	.633	.000)
					SCRATCH	250000	1	.008	.000
					SCR300	250000	1	.008	.000
TOTAL:									.013

This statistical table contains two parallel tables:

- The LOGICAL DBSETS table lists each DBset
- The DBSET FILES tables lists the component files of the DBSet.

In these tables:

- DBSET is the name of the DBSet.
- ALLOCATED is the NX Nastran DBSet size limit in blocks.
- BLOCKSIZE is BUFFSIZE of the DBSet minus one.
- USED (BLOCKS) and USED % are the number of GINO and executive blocks used at the end of the job as well as the percent of the DBSet actually used.
- FILE is the file's logical name associated with the DBSet to the left.
- ALLOCATED is the number of blocks allocated by NX Nastran to the file.
- HIWATER (BLOCKS) and HIWATER (MB) are the maximum number of GINO and executive blocks and actually used during the execution.
- I/O TRANSFERRED is the amount of I/O to the file. The last line of this column lists the total I/O transferred.

In this example, the MASTER and OBJSCR DBSets are each composed of one file. The DBALL DBSet is composed of two files, DBALL and DBALL2; and the SCRATCH DBSet has three components, MEMFILE, SCRATCH, and SCR300.

You can use this database statistics table to determine whether the DBSets and files are appropriately sized and to assess the amount of I/O activity associated with each file. You can obtain the best elapsed time performance can be obtained if the files with the greatest activity are on different physical devices (and better yet, separate I/O controllers or busses).

Summary of Physical File I/O Activity

This section of the .f04 file describes the physical file I/O for each database file.

*** SUMMARY OF PHYSICAL FILE I/O ACTIVITY ***

ASSIGNED PHYSICAL FILENAME	RECL (BYTES)	READ/WRITE COUNT	MAP WSIZE (NUM)	MAP COUNT
/tmp/65872_57.SCRATCH	8192	1	128KB (4)	1
/tmp/65872_57.OBJSCR	8192	378	128KB (4)	24
/tmp/65872_57.MASTER	8192	1247	128KB (4)	11
/tmp/65872_57.DBALL	8192	26	128KB (4)	1
/tmp/65872_57.SCR300	8192	1	128KB (4)	1
/nxnr/aix/SSS.MASTERA	8192	162	N/A	N/A

/nxnr/aix/SSS.MSCOBJ

8192

202

N/A

N/A

In this summary:

- ASSIGNED PHYSICAL FILENAME, RECL, and MAP WSIZE and NUM are repeated from the “Summary of Physical File Information” table.
- READ/WRITE COUNT is the number of GINO reads and writes that were performed on the file.
- MAP COUNT is the number of times the map window had to be remapped (these columns are only present on systems supporting mapped I/O).

You can use this summary to tune I/O performance. For mapped I/O systems, if the map count approaches the number of reads and writes, you should increase the map size and/or the number of maps.

- Increasing the number of maps is suggested if a module simultaneously accesses more data blocks or matrices in a file than there are windows.
- Increasing the size of the windows is suggested if a file contains very large data blocks or matrices.

You can obtain the best elapsed time performance, with or without mapping, if the files with the greatest activity are on different physical devices, and better yet, on separate I/O controllers or busses.

Other Optional Diagnostic Information

The DIAG Executive Control command has a number of different options that you can use to have NX Nastran print additional, diagnostic information to the .f04 file. Several of these options are described in detail below.

See also

- “**DIAG**” in the *NX Nastran Quick Reference Guide*

DIAG 8

If you include DIAG 8 in the Executive Control section of your input file, the software prints matrix trailers as the matrices are generated. Terms in the DIAG 8 output include:

- The number of GINO blocks in the matrix (BLOCKS);
- The average number of terms per string (STRL)
- The number of strings in a matrix data block (NBRSTR)
- The number of bank terms in a matrix data block on the average (BNDL)
- The number of banks in a matrix data block (NBRBND)
- The average first nonzero row position (ROWL)
- The average of full bank width (BNDAVG)

- The maximum full bank width of all columns of data blocks (BNDMAX)
- The number of null columns in a data block (NULCOL)

DIAG 13

If you include DIAG 13 in your input file, the software includes the message REAL CORE IS xxxx throughout the day log. It shows the value of the NASTRAN keyword REAL. The message only appears after module executions which use the value. The open core (i.e., working space) for other modules is set via HICORE

DIAG 19

If you include DIAG 19 in your input file, the software prints data for MPYAD and FBS method selection in the Execution Summary table at the top of the .f04 file.

DIAG 49

If you include DIAG 49, the software includes a detailed analysis of timing data (statistical information on module expected/actual CPU item) at the end of the .f04 file in three separate sections.

- The first section contains information about each module and submodule.
- The second section reports statistical information about the submodules (FBS, MPYAD, and DCMP) grouped by specific methods. Instances where the CPU estimate doesn't pass the following criteria are output:

$$\frac{|\text{Estimated CPU} - \text{Actual CPU}|}{\text{Actual CPU}} < 0.05$$

or

$$|\text{Estimated CPU} - \text{Actual CPU}| < 5\% \text{ of the total CPU of the job}$$

- The third section contains information about the modules listed in alphabetical order.

Comparing Different F04 Files with F04REPRT

The NX Nastran F04REPRT utility program contains a Perl script that allows you to summarize and/or compare the contents of different .f04 files. This utility can be helpful if you want to compare the relative performance of one or more jobs under various configurations.

See also

- “F04REPRT” in the *NX Nastran Installation and Operations Guide*

4.4 Understanding the .log File

The .log file contains system information, such as the name of the computer on which the analysis was run. It also contains any system errors encountered during the analysis.

4.5 Understanding the .plt and .pch Files

The .plt file is a binary file that contains the plotting information generated by the NX Nastran plotter. If the NX Nastran plotter is not used, run1.plt is deleted following the completion of the run.

The .pch file ("punch file") is an ASCII file that contains "punched" output. It is written to FORTRAN unit 7. In the Case Control section, you can specify whether you want the software to output specified results to the punch file.

4.6 Output Listing Examples

Examples of most of the printed output formats that are used in linear static analysis are contained in this section. Each example is annotated with comments and identified by title. These examples include:

Forces in Scalar Springs	Forces in CQUAD8 Elements
Stresses in Scalar Springs	Stresses in CQUAD8 Elements
Forces in CROD Elements	Strains in CQUAD8 Elements
Stresses in CROD Elements	Stresses in CHEXA Elements
Forces in CBAR Elements	Strains in CHEXA Elements
Stresses in CBAR Elements	Forces in CWELD Elements
Force Distribution in CBAR Elements (CBARAO)	Grid Point Force Balance
Stress Distribution in CBAR Elements (CBARAO)	Grid Point Stress Output
Forces in CBEAM Elements	Grid Point Stress Discontinuities Output
Stresses in CBEAM Elements	Element Stress Discontinuities Output
Forces in CBEND Elements	Element Strain Energy
Stresses in CBEND Elements	Sequence Processor Output
Forces in CBUSH Element	Grid Point Singularity Table
Forces Acting on Shear Panel Elements	User Information Message 5293
Stresses in Shear Panels	OLOAD Resultant
Forces in CQUAD4 Elements (Without Corner Option)	SPCFORCE Resultant
Stresses in CQUAD4 Elements (Without Corner Option)	Maximum SPCFORCES, DISPLACEMENT, and Applied Loads
Strains in CQUAD4 Elements (Without Corner Option)	Displacement Vectors
Forces in CQUAD4 Elements (With Corner Option)	Load Vectors
Stresses in CQUAD4 Elements (With Corner Option)	Database Summary Table
Strains in CQUAD4 Elements (With Corner Option)	Restartable Versions

Forces in Scalar Springs (C E L A S 2)

ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE
1	-6.366724E+02	2	3.633277E+02	3	-5.537512E+02	4	3.653757E+02

- Standard FORCE output, requested in Case Control Section.
- This output is typical of all scalar elements (CELAS1, CELAS2, CELAS3, CELAS4).

Stresses in Scalar Springs (C E L A S 2)

ELEMENT ID.	STRESS	ELEMENT ID.	STRESS	ELEMENT ID.	STRESS	ELEMENT ID.	STRESS
1	-6.366723E+04	2	3.633277E+04	3	-5.537512E+04	4	3.653757E+04

- Standard STRESS output, requested in Case Control Section
- Stress output is obtained by multiplying the force in the spring by the stress coefficient entered in field 9 of the CELAS2 entry or field 9 of the PELAS entry.
- This output applies only to the CELAS1 and CELAS2 Bulk Data entries.

Forces in CROD Elements

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

- See “One-dimensional Elements” for a definition of positive torque.
- Positive axial force is tension.

Stresses in CROD Elements

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



1	Axial stress and margin-of-safety due to axial stress only
2	Torsional stress and margin-of-safety due to torsion stress only

- Positive stress is tension.
- Standard STRESS output, requested in Case Control Section.

- Definition of Margin-of-Safety (M.S.): The M.S. for axial stress is: $M.S. = (ST/SA) - 1.0$ if SA is greater than zero or $M.S. = (-SC/SA) - 1.0$ if SA is less than zero, where ST and SC are the tension and compression allowables given on the MAT1 entry and SA is the axial stress given in the output. M.S. is computed and printed if and only if ST (or SC) is greater than zero and SA is not equal to zero.

The M.S. for torsional stress is: $M.S. = (SS/TAU) - 1.0$, where SS is the shear allowable given on the MAT1 entry and TAU is the torsional stress given in the output. M.S. is computed and printed if and only if SS is greater than zero and TAU is not equal to zero.

Forces in CBAR Elements

ELEMENT ID.	BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL FORCE	TORQUE
	PLANE 1	PLANE 2	PLANE 1	PLANE 2	PLANE 1	PLANE 2		
1	2.586560E+01	-1.519694E+01	1.619619E+00	4.542250E+00	2.168636E-02	-1.765535E-02	3.726698E+03	2.810185E+00
2	2.586560E+01	1.519694E+01	1.619619E+00	-4.542250E+00	2.168636E-02	1.765535E-02	3.726698E+03	-2.810185E+00
3	-5.172854E+01	.0	-3.240443E+00	.0	-4.336907E-02	.0	-7.453428E+03	.0

- Standard FORCE output, requested in the Case Control Section.
- See CBAR in the “NX Nastran Element Library Reference” for definition of planes 1 and 2 and for positive directions of bending moments, shears, axial force, and torque.

Stresses in CBAR Elements

ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
1	8.434532E-01	1.054002E+01	-1.054002E+01	-8.434532E-01	4.658372E+02	4.763772E+02	4.552972E+02	-3.2E-02
	-2.004801E+00	-1.397634E+00	1.397634E+00	2.004801E+00		4.678420E+02	4.638324E+02	-2.8E-03
2	-1.054002E+01	-8.434532E-01	8.434532E-01	1.054002E+01	4.658372E+02	4.763772E+02	4.552972E+02	
	1.397634E+00	2.004801E+00	-2.004801E+00	-1.397634E+00		4.678420E+02	4.638324E+02	
3	9.696071E+00	-9.696071E+00	9.696071E+00	-9.696071E+00	-9.316785E+02	-9.219824E+02	-9.413745E+02	6.1E-12
	6.073933E-01	-6.073933E-01	6.073933E-01	-6.073933E-01		-9.310710E+02	-9.322859E+02	6.0E-12

1	Normal stresses due to bending only at ends A and B, points C, D, E, and F Example: SA1 = stress at end A, point C. SB4 = stress at end B, point F.
2	Normal stresses due to axial loads only.
3	Normal stresses due to combined bending and axial loads. Maximum and minimum values over points C, D, E, and F at ends A and B.
4	Margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

- Positive stress is tension.
- Standard STRESS output, requested in Case Control Section.
- Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output, $S-MAX = \max (SA-MAX,$

SB-MAX). M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 card and S-MIN is the minimum stress given in the output, $S-MIN = \min(SA-MIN, SB-MIN)$. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Force Distribution in CBAR Elements (CBARAO)

ELEMENT ID.	STATION (PCT)	BEND-MOMENT		SHEAR FORCE		AXIAL FORCE	TORQUE
		PLANE 1	PLANE 2	PLANE 1	PLANE 2		
11	.000	.0	1.868856E+06	.0	-7.851453E+03	.0	.0
11	.200	.0	3.439147E+06	.0	-7.851453E+03	.0	.0
11	.400	.0	4.887104E+06	.0	-5.331454E+03	.0	.0
11	.600	.0	5.346729E+06	.0	1.028547E+03	.0	.0
11	.800	.0	4.358352E+06	.0	9.148546E+03	.0	.0
11	1.000	.0	2.528644E+06	.0	9.148546E+03	.0	.0

Diagram showing a horizontal line with points 1 and 2 marked below it. Point 1 is at the left end, and point 2 is at the right end.

1	Element ID
2	Distance from end A as a fraction of the element's length

- Optional output for the CBAR element using the CBARAO entry.
- See CBAR in the “NX Nastran Element Library Reference” for definition of planes 1 and 2.
- Force output must be requested in Case Control Section.

Stress Distribution in CBAR Elements (CBARAO)

ELEMENT ID.	STATION (PCT)	SXC	SXD	SXE	SXF	AXIAL	S-MAX	S-MIN	M.S.-T M.S.-C
11	.000	-1.868856E+04	-1.868856E+04	1.868856E+04	1.868856E+04	.0	1.868856E+04	-1.868856E+04	6.1E-01
11	.200	-3.439147E+04	-3.439147E+04	3.439147E+04	3.439147E+04	.0	3.439147E+04	-3.439147E+04	-1.3E-01
11	.400	-4.887105E+04	-4.887105E+04	4.887105E+04	4.887105E+04	.0	4.887105E+04	-4.887105E+04	-3.9E-01
11	.600	-5.346729E+04	-5.346729E+04	5.346729E+04	5.346729E+04	.0	5.346729E+04	-5.346729E+04	-4.4E-01
11	.800	-4.358352E+04	-4.358352E+04	4.358352E+04	4.358352E+04	.0	4.358352E+04	-4.358352E+04	-3.1E-01
11	1.000	-2.528644E+04	-2.528644E+04	2.528644E+04	2.528644E+04	.0	2.528644E+04	-2.528644E+04	1.9E-01

Diagram showing a horizontal line with points 1, 2, 3, 4, and 5 marked below it. Point 1 is at the left end, point 2 is at the right end, point 3 is at the center, point 4 is at the left end, and point 5 is at the right end.

1	Distance from end A as a fraction of the element's length.
2	Normal stresses due to bending point at points C, D, E, and F.
3	Normal stresses due to axial loads only.
4	Normal stresses due to combined bending and axial loads. Maximum and minimum values over points C, D, E, and F.
5	Margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

- The default position for points C, D, E and F is on the neutral axis.
- This is special output requested by including CBARA0 and/or PLOAD1 entries in the Bulk Data Section along with a STRESS request in the Case Control Section.
- Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output. A M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 entry and S-MIN is the minimum stress given in the output. An M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Forces in CBEAM Elements

ELEMENT-ID	GRID	STAT DIST/ LENGTH	- BENDING MOMENTS -		- WEB SHEARS -		AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
			PLANE 1	PLANE 2	PLANE 1	PLANE 2			
1									
	1	.000	8.653494E+03	-9.295851E+03	3.013959E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.200	6.791337E+03	-7.331033E+03	2.765999E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.400	4.929180E+03	-5.366216E+03	2.518019E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.600	3.067023E+03	-3.401399E+03	2.270049E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.800	1.204866E+03	-1.436580E+03	2.022079E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	2	1.000	-6.572902E+02	5.282371E+02	1.774109E+02	-1.995699E+02	9.060291E+01	-7.902837E+01	9.954489E-17

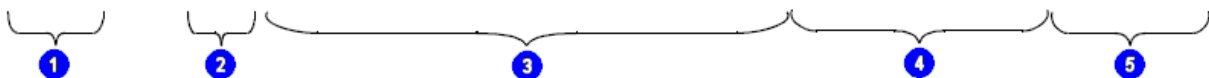


1	Element ID
2	Grid points at ends A and B
3	Distance from end A as a fraction of the element's length
4	Part of total torque due to cross-sectional warping

- See CBAR in the “NX Nastran Element Library Reference” for definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force, and torque.
- Standard FORCE output, requested in Case Control Section.

Stresses in CBEAM Elements

ELEMENT-ID	GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
1										
	1	.000	-6.118605E+02	1.554121E+03	6.269609E+02	-1.539021E+03	1.554121E+03	-1.539021E+03		
	0	.200	-6.634866E+02	1.679736E+03	6.816094E+02	-1.661615E+03	1.679736E+03	-1.661615E+03		
	0	.400	-6.934662E+02	1.748845E+03	7.173659E+02	-1.724945E+03	1.748845E+03	-1.724945E+03	-4.3E-01	-4.2E-01
	0	.600	-6.610018E+02	1.649506E+03	6.914990E+02	-1.619019E+03	1.649506E+03	-1.619019E+03		
	0	.800	-4.553899E+02	1.094180E+03	4.965609E+02	-1.053009E+03	1.094180E+03	-1.053009E+03		
	2	1.000	2.408044E+02	-7.437941E+02	-1.804025E+02	8.041960E+02	8.041960E+02	-7.437941E+02		



1	Element ID
2	Distance from end A as a fraction of the element's length
3	Normal stresses due to combined bending and axial loads at points C, D, E, and F
4	Maximum and minimum value over points C, D, E, and F
5	Minimum margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element

- Standard STRESS output, requested in Case Control Section.
- Positive stress is tension.
- Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output, $S-MAX = \max(SA-MAX, SB-MAX)$. M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 card and S-MIN is the minimum stress given in the output, $S-MIN = \min(SA-MIN, SB-MIN)$. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Forces in CBEND Elements

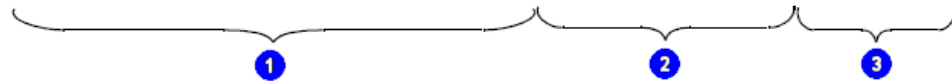
ELEMENT-ID	GRID	END	- BENDING MOMENTS -		- SHEARS -		AXIAL FORCE	TORQUE
			PLANE 1	PLANE 2	PLANE 1	PLANE 2		
991	1	A	-9.507964E+02	-1.118982E+02	-7.252229E+01	4.371233E+00	6.426835E+01	-3.338711E+02
	2	B	-6.589756E+02	-4.884176E+02	6.426834E+01	4.371233E+00	7.252229E+01	2.664446E+02

1	Element ID
2	Grid points at ends A and B

- See CBAR in the “NX Nastran Element Library Reference” for definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force, and torque.
- Standard FORCE output, requested in Case Control Section.

Stresses in CBEND Elements

ELEMENT-ID	GRID	END	CIRC. ANG.	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
991	1	A	0	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01	6.426835E+01		
	2	B	0	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	7.252229E+01	3.8E-01	



1	Normal stresses due to combined bending and axial loads at points C, D, E, and F
2	Maximum and minimum value over points C, D, E, and F
3	Minimum margin-of-safety in tension (T) and compression (C). Minimum M.S. found over entire length of element.

- Standard STRESS output, requested in Case Control Section.
- Positive stress is tension.
- Definition of Margin-of-Safety (M.S.):

The M.S. in tension is: $M.S.-T = (ST / S-MAX) - 1.0$, where ST is the tension allowable given on the MAT1 entry and S-MAX is the maximum stress given in the output, $S-MAX = \max(SA-MAX, SB-MAX)$. M.S. in tension is computed and printed if and only if ST is greater than zero and S-MAX is greater than zero.

The M.S. in compression is: $M.S.-C = (-SC / S-MIN) - 1.0$, where SC is compression allowable given on the MAT1 card and S-MIN is the minimum stress given in the output, $S-MIN = \min(SA-MIN, SB-MIN)$. M.S. in compression is computed and printed if and only if SC is greater than zero and S-MIN is less than zero.

Forces in CBUSH Elements

	ELEMENT-ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
0	10000	-6.489702E+00	-5.997736E+00	6.265953E-01	3.955022E+00	-9.723748E-02	1.147097E-02
0	10001	-3.201495E+00	-1.951144E+00	2.724363E-01	9.105669E+00	-1.866592E-01	2.313610E-02
0	10002	9.125268E-01	2.533982E+00	2.790101E-01	9.008885E+00	-1.804384E-01	2.278207E-02
0	10003	2.939918E+00	5.093895E+00	3.161362E-01	7.944401E+00	-1.380170E-01	2.264773E-02
0	10004	3.530923E+00	7.028401E+00	3.253268E-01	7.911948E+00	-8.659782E-02	2.265486E-02
0	10005	6.848238E+00	4.654799E+00	3.589581E-01	7.842767E+00	-2.292452E-02	2.255161E-02
0	10006	6.789301E+00	3.347241E+00	3.445033E-01	7.912354E+00	4.136241E-02	2.287587E-02
0	10007	4.012078E+00	2.324608E+00	3.328075E-01	7.965162E+00	1.027839E-01	2.302680E-02
0	10008	5.789543E-01	-4.556357E-01	3.059701E-01	9.035738E+00	1.431401E-01	2.305412E-02
0	10009	-4.496515E+00	-5.173875E+00	2.923176E-01	9.129048E+00	1.063870E-01	2.287990E-02
0	10010	-7.422128E+00	-7.604536E+00	5.459386E-01	3.958032E+00	-3.207400E-02	1.047710E-02

- The forces are in the CBUSH element coordinate system.

Forces Acting on Shear Panel Elements (CSHEAR)

ELEMENT ID	----- POINT 1 -----		----- POINT 2 -----		----- POINT 3 -----		----- POINT 4 -----	
	F-FROM-4	F-FROM-2	F-FROM-1	F-FROM-3	F-FROM-2	F-FROM-4	F-FROM-3	F-FROM-1
	KICK-1	SHEAR-12	KICK-2	SHEAR-23	KICK-3	SHEAR-34	KICK-4	SHEAR-41
11	-1.49620E+00	-1.49620E+00	1.49620E+00	1.49620E+00	-1.49620E+00	1.49620E+00	1.49620E+00	1.49620E+00
	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01	.0	-2.99239E-01
12	-1.28268E+00	-1.28268E+00	1.28268E+00	1.28268E+00	-1.28268E+00	1.28268E+00	1.28268E+00	1.28268E+00
	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01	.0	-2.56536E-01

- See 2D Elements in the “NX Nastran Element Library Reference” for definitions of element coordinate system and force components.
- Standard FORCE output, requested in the Case Control Section.

Stresses in Shear Panels (CSHEAR)

ELEMENT ID.	MAX SHEAR	AVG SHEAR	SAFETY MARGIN	ELEMENT ID.	MAX SHEAR	AVG SHEAR	SAFETY MARGIN
11	2.992391E+00	-2.992391E+00	1.5E+00	12	2.565355E+00	-2.565355E+00	

1	Maximum and average shear stresses and margin-of-safety due to shear only
---	---

- Standard STRESS output, requested in Case Control Section.
- Definition of Margin-of-Safety (M.S.): $M.S. = (SS/MAX\ SHEAR) - 1.0$, where SS is the shear allowable given on the MAT1 entry and MAX SHEAR is the maximum shear stress given in the output. M.S. is computed and printed if and only if SS is greater than zero and MAX SHEAR is not equal to zero.

Forces in CQUAD4 Elements without Corner Option

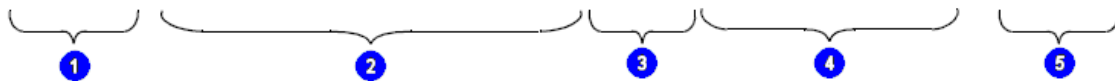
ELEMENT ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -	
	FX	FY	FXI	MX	MY	MXI	QX	QY
1	-2.349886E+00	-3.348329E-01	-2.992391E-01	-1.250000E+00	-2.983451E+00	-5.573193E-01	-5.000000E-01	2.352497E-02
2	-1.593343E+00	2.805547E-02	-2.563553E-01	-7.500000E+00	-5.912674E-01	-1.125781E-01	-5.000000E-01	3.125528E-02
3	-8.648885E-01	2.233199E-02	-4.740243E-01	-2.500000E+00	6.004672E-01	3.312782E-01	-5.000000E-01	1.284359E-02
4	1.349886E+00	1.656714E-01	-4.015217E-01	-1.250000E+00	-2.983451E+00	5.573193E-01	-5.000000E-01	-2.352497E-02
5	5.933433E-01	-8.414105E-02	-4.869289E-01	-7.500000E+00	-5.912674E-01	1.125781E-01	-5.000000E-01	-3.125528E-02
6	-1.351115E-01	-4.106952E-03	-5.259757E-01	-2.500000E+00	6.004672E-01	-3.312782E-01	-5.000000E-01	-1.284359E-02

1	In-plane element forces in element coordinate system (force/length).
2	Element internal moments in element coordinate system (moment/length).
3	Traverse shear forces (force/length).

- Positive force is tension.
- See 2D Elements in the “NX Nastran Element Library Reference” for definitions of element coordinate system and positive shears and moments.
- This output is typical for all shell elements
- Standard FORCE output, requested in Case Control Section.

Stresses in CQUAD4 Elements without Corner Option

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	-5.000000E-02	-7.523499E+03	-1.793419E+03	-3.373840E+02	-86.6419	-1.773622E+03	-7.543295E+03	6.831404E+03
	5.000000E-02	7.476501E+03	1.786722E+03	3.313992E+02	3.3222	7.495738E+03	1.767485E+03	6.786861E+03
2	-5.000000E-02	-4.515933E+03	-3.544799E+02	-7.011219E+01	-89.0350	-3.532990E+02	-4.517114E+03	4.351235E+03
	5.000000E-02	4.484066E+03	3.550410E+02	6.498148E+01	.9014	4.485089E+03	3.540186E+02	4.318975E+03
3	-5.000000E-02	-1.508649E+03	3.605036E+02	1.940267E+02	84.1357	3.804320E+02	-1.528577E+03	1.750085E+03
	5.000000E-02	1.491351E+03	-3.600570E+02	-2.035071E+02	-6.1993	1.513457E+03	-3.621626E+02	1.736371E+03

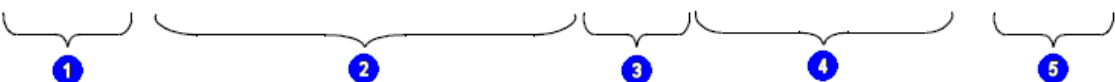


1	Fiber distance Z1 and Z2 specified on the PSHELL property entry
2	Stresses in element coordinate system
3	Angle of principal stress in element coordinate system
4	Major and minor principal stresses
5	Maximum shear stress or Hencky-von Mises stress

- Standard STRESS output, requested in Case Control Section.
- This output is typical for all plate elements
- See 2D Elements in the “NX Nastran Element Library Reference” for definitions of element coordinate system.

Strains in CQUAD4 Elements without Corner Option

ELEMENT ID.	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	.0	-2.249436E-06	3.701331E-07	-7.780218E-07	-81.7292	4.266812E-07	-2.305984E-06	1.697520E-06
	-1.000000E+00	-1.392596E-02	9.199587E-04	-1.738836E-03	-86.6598	9.706012E-04	-1.397670E-02	9.657605E-03
2	.0	-1.601760E-06	5.060585E-07	-6.669924E-07	-81.2203	5.575652E-07	-1.653267E-06	1.327650E-06
	-1.000000E+00	-8.787143E-03	1.990479E-03	-3.512435E-04	-89.0667	1.993340E-03	-8.790004E-03	6.625173E-03
3	.0	-8.715881E-07	2.817985E-07	-1.232463E-06	-66.5508	5.490937E-07	-1.138883E-06	9.941859E-07
	-1.000000E+00	-3.216168E-03	1.620561E-03	1.033588E-03	83.9688	1.675163E-03	-3.270770E-03	2.904646E-03



1	If fiber strains are requested (see Note 2) the fiber distances Z1 and Z2 will appear here. If strain-curvature data is requested, this entry will appear as shown
2	Strains and curvatures in element coordinate system
3	Angle of principal strain (degrees)
4	Major and minor principal stresses
5	Maximum shear stress or Hencky-von Mises stress

- Strain output is available for plate elements and is requested using the Case Control command STRAIN..
- Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.
- This output is typical for all plate elements.
- See 2D Elements in the “NX Nastran Element Library Reference” for definitions of element coordinate system.
- Material coordinate systems are specified on MATi entries.

Forces in CQUAD4 Elements with Corner Option

```

OPTION = BILIN
ELEMENT
ID      GRID-ID  - MEMBRANE FORCES -      - BENDING  MOMENTS -      - TRANSVERSE SHEAR FORCES -
      FX      FY      FXY      MX      MY      MXY      QX      QY
1  CEN/4 -2.349886E+00 -3.198563E-01 -5.233257E-01 -1.250000E+01 -2.983451E+00 -5.573193E-01 -5.000000E-01 2.352497E-02
      1 -4.448889E+00 -1.334667E+00 3.150085E-01 -1.434321E+01 -4.302963E+00 -6.366503E-01 -5.000000E-01 2.352497E-02
      2 -4.194971E+00 -4.882719E-01 -1.065422E+00 -9.086258E+00 -9.787024E-01 -1.686622E+00 -5.000000E-01 2.352497E-02
      6 -2.508836E-01 6.549542E-01 -1.361660E+00 -1.040833E+01 -1.803477E+00 -5.073156E-01 -5.000000E-01 2.352497E-02
      5 -5.048020E-01 -1.514406E-01 1.877032E-02 -1.616221E+01 -4.848663E+00 5.426558E-01 -5.000000E-01 2.352497E-02
2  CEN/4 -1.593343E+00 6.351992E-02 -4.918268E-01 -7.500000E+00 -5.912674E-01 -1.125781E-01 -5.000000E-01 3.125528E-02
      2 -2.714164E+00 -4.402977E-02 -1.611118E-01 -1.134483E+01 -1.656274E+00 -1.138151E+00 -5.000000E-01 3.125528E-02
      3 -2.864952E+00 -5.466592E-01 -9.984620E-01 -6.194139E+00 -3.528372E-03 7.047635E-01 -5.000000E-01 3.125528E-02
      7 -4.725232E-01 1.710696E-01 -8.225417E-01 -3.723025E+00 5.971142E-01 8.980479E-01 -5.000000E-01 3.125528E-02
      6 -3.217343E-01 6.736990E-01 1.480843E-02 -8.738006E+00 -1.302381E+00 -9.448670E-01 -5.000000E-01 3.125528E-02

```



1	Denotes CEN/4 for the center stress or the grid point ID for a corner stress
---	--

- Positive force is tension.
- See 2D Elements in the “NX Nastran Element Library Reference” for definitions of element coordinate system and definition of positive shears and moments.
- This output is typical for all shell elements.
- Standard FORCE output, requested in Case Control Section.

- Force output obtained when using the FORCE(CORNER) in Case Control Section.
- The transverse shear forces (QX and QY) are computed at the center only, and those same values are displayed at the corners.

Stresses in CQUAD4 Elements with Corner Option

OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	-7.523499E+03	-1.793269E+03	-3.396249E+02	-86.6199	-1.773210E+03	-7.543558E+03	6.831780E+03
		5.000000E-02	7.476501E+03	1.786872E+03	3.291584E+02	3.3000	7.495480E+03	1.767893E+03	6.786491E+03
	1	-5.000000E-02	-8.650413E+03	-2.595124E+03	-3.788401E+02	-86.4339	-2.571515E+03	-8.674022E+03	7.716604E+03
		5.000000E-02	8.561436E+03	2.568431E+03	3.851403E+02	3.6620	8.586085E+03	2.543781E+03	7.638754E+03
	2	-5.000000E-02	-5.493704E+03	-5.921041E+02	-1.022627E+03	-78.6755	-3.873087E+02	-5.698500E+03	5.515055E+03
		5.000000E-02	5.409805E+03	5.823387E+02	1.001319E+03	11.2654	5.609259E+03	3.828847E+02	5.427954E+03
	6	-5.000000E-02	-6.247504E+03	-1.075137E+03	-3.180060E+02	-86.4949	-1.055658E+03	-6.266982E+03	5.811513E+03
		5.000000E-02	6.242486E+03	1.089036E+03	2.907728E+02	3.2192	6.258840E+03	1.072681E+03	5.797412E+03
	5	-5.000000E-02	-9.702373E+03	-2.910712E+03	3.257812E+02	87.2600	-2.895121E+03	-9.717964E+03	8.642099E+03
		5.000000E-02	9.692277E+03	2.907683E+03	-3.254057E+02	-2.7397	9.707849E+03	2.892112E+03	8.633103E+03



1

Denotes CEN/4 for the center stress or the grid point ID for a corner stress

- This is standard STRESS output requested in Case Control Section.
- Stress output obtained using the STRESS(CORNER) command in the Case Control Section.
- See 2D Elements in the “NX Nastran Element Library Reference” for a definition of element coordinate system.

Strains in CQUAD4 Elements with Corner Option

OPTION = BILIN

ELEMENT ID	GRID-ID	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	.0	-2.253929E-06	3.851096E-07	-1.360647E-06	-76.3625	5.501681E-07	-2.418988E-06	1.823920E-06
		-1.000000E+00	-1.392596E-02	9.198587E-04	-1.738836E-03	-86.6598	9.706012E-04	-1.397670E-02	9.657605E-03
	1	.0	-4.048489E-06	.0	8.190221E-07	84.2816	4.100733E-08	-4.089496E-06	2.740102E-06
		-1.000000E+00	-1.566278E-02	.0	-1.986349E-03	-86.3862	6.272583E-05	-1.572551E-02	1.050464E-02
	2	.0	-4.048489E-06	7.702193E-07	-2.770097E-06	-75.0535	1.139956E-06	-4.418226E-06	3.389972E-06
		-1.000000E+00	-1.055118E-02	2.096610E-03	-5.262259E-03	-78.7048	2.622131E-03	-1.107670E-02	8.396119E-03
	6	.0	-4.593699E-07	7.702193E-07	-3.540316E-06	-54.5763	2.029306E-06	-1.718457E-06	2.166252E-06
		-1.000000E+00	-1.184074E-02	1.582825E-03	-1.582825E-03	-86.6375	1.629323E-03	-1.188724E-02	8.520021E-03
	5	.0	-4.593699E-07	.0	4.880284E-08	86.9679	1.292550E-09	-4.606624E-07	3.075400E-07
		-1.000000E+00	-1.764913E-02	.0	1.693086E-03	87.2602	4.051156E-05	-1.768964E-02	1.180662E-02



1

Denotes CEN/4 for the center strain and curvatures and a grid point ID for a corner strain or curvature

- Corner strain output is available for the plate elements and is requested using the Case Control command STRAIN(CORNER).
- Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.
- This output is typical for all plate elements.
- See 2D Elements in the “NX Nastran Element Library Reference” for a definition of element coordinate system.
- Material coordinate systems are specified on MATi entries.

Forces in CQUAD8 Elements

ELEMENT ID	GRID-ID	- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -		
		FX	FY	FXI	MY	MX	MY	MXI	QX	QY
1	CEN/4	-2.197889E+00	-3.254840E-01	-5.232677E-01	-1.250000E+01	-3.691373E+00	-3.563170E-02	-5.205615E+00	8.254652E-02	
	1	-4.108907E+00	-1.232672E+00	-1.282016E+00	-1.483570E+01	-4.450711E+00	-5.782321E-02	-5.205615E+00	8.254652E-02	
	2	-3.888765E+00	-4.988650E-01	-2.135196E-02	-1.014609E+01	-2.877548E+00	-5.853834E-02	-5.205615E+00	8.254652E-02	
	6	-2.868679E-01	5.817043E-01	2.354806E-01	-9.853907E+00	-2.887942E+00	-1.344019E-02	-5.205615E+00	8.254652E-02	
	5	-5.070101E-01	-1.521030E-01	-1.025193E+00	-1.516430E+01	-4.549290E+00	-1.272506E-02	-5.205615E+00	8.254652E-02	
2	CEN/4	-1.499658E+00	4.452617E-02	-4.931346E-01	-7.500000E+00	-2.112700E+00	-1.628812E-02	-1.779801E+00	2.273268E-02	
	2	-2.536411E+00	-9.315875E-02	-8.093295E-01	-9.912405E+00	-2.807442E+00	-2.389918E-02	-1.779801E+00	2.273268E-02	
	3	-2.650702E+00	-4.741292E-01	-4.360040E-02	-5.061935E+00	-1.334132E+00	-2.409612E-02	-1.779801E+00	2.273268E-02	
	7	-4.629044E-01	1.822111E-01	-1.769397E-01	-4.938065E+00	-1.351179E+00	-8.677053E-03	-1.779801E+00	2.273268E-02	
	6	-3.486136E-01	5.631805E-01	-9.426688E-01	-1.008759E+01	-2.958048E+00	-8.480112E-03	-1.779801E+00	2.273268E-02	



1

Denotes CEN/4 for the center forces or the grid point ID for a corner force

- Positive force is tension.
- See 2D Elements in the “NX Nastran Element Library Reference” for a definition of element coordinate system and a definition of positive shears and moments.
- This output is typical for all shell elements.
- Standard FORCE output, requested in Case Control Section.

Stresses in CQUAD8 Elements

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON MISES
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	
1	CEN/4	-5.000000E-02	-7.523499E+03	-1.793269E+03	-3.396249E+02	-86.6199	-1.773210E+03	-7.843558E+03	6.831780E+03
		5.000000E-02	7.476501E+03	1.786872E+03	3.291584E+02	3.3000	7.495480E+03	1.767893E+03	6.786491E+03
	1	-5.000000E-02	-8.650413E+03	-2.595124E+03	-3.788401E+02	-86.4339	-2.571515E+03	-8.674022E+03	7.736604E+03
		5.000000E-02	8.561436E+03	2.568431E+03	3.851403E+02	3.6620	8.586085E+03	2.543781E+03	7.638754E+03
	2	-5.000000E-02	-5.493704E+03	-5.921041E+02	-1.022627E+03	-78.6755	-3.873087E+02	-5.698500E+03	5.515055E+03
		5.000000E-02	5.409805E+03	5.823387E+02	1.001319E+03	11.2654	5.609259E+02	3.828847E+02	5.427954E+03
	6	-5.000000E-02	-6.247504E+03	-1.075137E+03	-3.180060E+02	-86.4949	-1.055658E+03	-6.266982E+03	5.811513E+03
		5.000000E-02	6.242486E+03	1.089036E+03	2.907728E+02	3.2192	6.258840E+03	1.072681E+03	5.797412E+03
	5	-5.000000E-02	-9.703373E+03	-2.910712E+03	-3.257812E+02	87.2600	-2.895121E+03	-9.717964E+03	8.642099E+03
		5.000000E-02	9.692277E+03	2.907683E+03	-3.254057E+02	-2.7397	9.707849E+03	2.892112E+03	8.633103E+03



1

Denotes CEN/4 for the center stress or the grid point ID for a corner stress

- This is standard STRESS output requested in Case Control Section.
- This output is typical for all plate elements.
- See 2D Elements in the “NX Nastran Element Library Reference” for a definition of element coordinate system.

Strains in CQUAD8 Elements

ELEMENT ID	GRID-ID	STRAIN CURVATURE	STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)	VON MISES		
			NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE MAJOR MINOR			
1	CEN/4	.0	-2.100242E-06	3.339823E-07	-1.360496E-06	-75.3990	5.110961E-07	-2.277446E-06	1.714246E-06
		-1.000000E+00	-1.367111E-02	7.035311E-05	-1.111709E-04	-89.7682	7.057795E-05	-1.367133E-02	9.137837E-03
	1	.0	-3.739106E-06	5.151080E-22	-3.333241E-06	-69.1422	6.350135E-07	-4.374119E-06	3.149165E-06
		-1.000000E+00	-1.620059E-02	2.948739E-20	-1.804084E-04	-89.6810	5.022378E-07	-1.620109E-02	1.080089E-02
	2	.0	-3.739106E-06	6.677646E-07	-5.551511E-08	-89.6391	6.679394E-07	-3.739280E-06	2.742746E-06
	-1.000000E+00	-1.113939E-02	1.995355E-04	-1.826396E-04	-89.5386	2.002709E-04	-1.114013E-02	7.494402E-03	
6	.0	-4.613792E-07	6.677646E-07	6.122495E-07	75.7662	7.454184E-07	-5.390330E-07	7.447625E-07	
		-1.000000E+00	-1.078503E-02	8.187693E-05	-4.193340E-05	-89.8895	8.191738E-05	-1.078507E-02	7.217507E-03
	5	.0	-4.613792E-07	-1.419557E-22	-2.665477E-06	-49.9101	1.121967E-06	-1.593246E-06	1.569352E-06
	-1.000000E+00	-1.655941E-02	2.215808E-22	-3.970219E-05	-89.9313	2.379707E-08	-1.655944E-02	1.103963E-02	



1

Denotes CEN/4 for the center strain and curvatures and a grid point ID for a corner strain and curvature

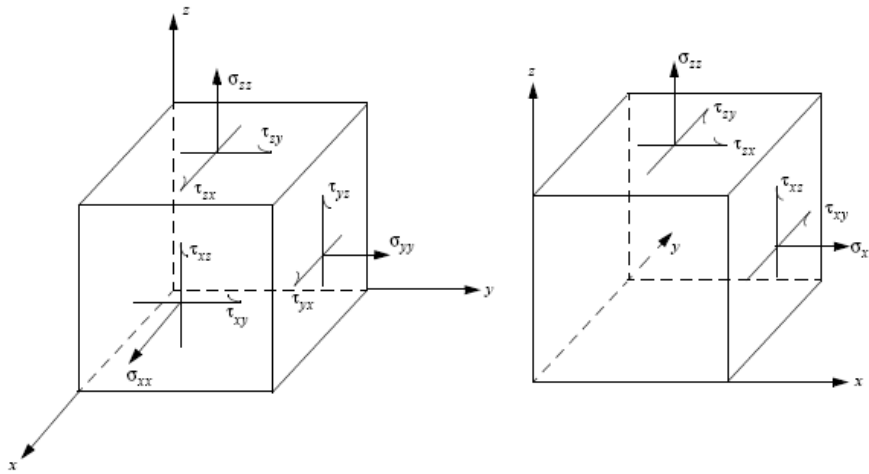
- Strain output is available for plate elements and is requested using the Case Control command STRAIN.
- Fiber strains at fiber distances Z1 and Z2 may be obtained by requesting for STRAIN (FIBER) in the Case Control Section. Z1 and Z2 are specified on the property entry.
- This output is typical for all plate elements.
- See 2D Elements in the “NX Nastran Element Library Reference” for a definition of element coordinate system.
- Material coordinate systems are specified on MATi entries.

Stresses in CHEXA Elements

ELEMENT-ID	CORNER GRID-ID	CS	-----CENTER AND CORNER POINT STRESSES-----						DIR. COSINES			MEAN PRESSURE	VON MISES
			NORMAL	SHEAR	PRINCIPAL	-A-	-B-	-C-					
1	1 GRID	CS	8 GP										
CENTER													
	X		-1.559055E+01	XY	-4.613456E-01	A	1.642750E+01	LX	.01	1.00	.03	-6.818652E-02	2.776379E+01
	Y		-6.143418E-01	YZ	-4.613456E-01	B	-1.560962E+01	LY	-.03	.03	-1.00		
	Z		1.640945E+01	ZX	4.094507E-01	C	-6.133184E-01	LZ	1.00	-.01	-.03		
51													
	X		7.173539E+01	XY	5.611941E+00	A	2.073488E+02	LX	-.59	.81	-.03	-9.944785E+01	1.848900E+02
	Y		6.159913E+01	YZ	-1.955275E+00	B	-6.360914E-01	LY	-.03	-.06	-1.00		
	Z		1.350090E+02	ZX	-9.999012E+01	C	6.163087E+01	LZ	.81	.59	-.06		
55													
	X		6.940665E+01	XY	4.212266E+00	A	1.836086E+02	LX	-.61	.79	-.02	-9.151105E+01	1.614635E+02
	Y		5.992439E+01	YZ	-6.249875E-01	B	9.289488E-01	LY	-.02	-.05	-1.00		
	Z		1.152021E+02	ZX	-8.930753E+01	C	5.999560E+01	LZ	.79	.61	-.05		

1	Stress coordinate system definition 0 = Basic Coordinate System -1 = Element Coordinate System +X = Material Coordinate System Defined on PSOLID entry.
2	Location of Stresses GRID = Stresses at center and vertex points GAUSS = Stresses at center and Gauss points
3	Number of active grid points for the element
4	Center and corner point stresses in the stress coordinate system
5	Directional cosines of the principal stress with respect to the stress coordinate system
6	Hencky-von Mises stress or octahedral shear stress

- This output is typical for the CHEXA, CPENTA, CPYRAM, and CTETRA elements.
- Standard STRESS output, requested in Case Control Section.
- The convention for the principal stresses are such that ""(A ≥ C ≥ B). In the case where the principal stresses are equal, the directional cosines are not unique, and the values of zero are output for the directional cosines.



Strains in CHEXA Elements

ELEMENT-ID	CORNER	-----CENTER AND CORNER POINT				STRAINS-----			DIR. COSINES			MEAN	VON MISES
	GRID-ID	NORMAL		SHEAR		PRINCIPAL			-A-	-B-	-C-	PRESSURE	
1	1GRID CS	8 GP											
	CENTER	X	-3.315102E-06	XY	4.167708E-08	A	2.637197E-06	LX	.01	1.00	-.01	1.582972E-07	3.456435E-06
		Y	2.051620E-07	YZ	-9.459202E-08	B	-3.316474E-06	LY	-.02	-.01	-1.00		
		Z	2.635048E-06	ZX	1.719858E-07	C	2.043855E-07	LZ	1.00	-.01	-.02		
51	X	2.388123E-06	XY	-2.219673E-06	A	1.648163E-05	LX	-.57	.82	.01		-3.979110E-06	1.271542E-05
		Y	-1.419658E-07	YZ	2.219673E-06	B	-4.278821E-06	LY	.09	.07	-.99		
		Z	9.691174E-06	ZX	-1.925896E-05	C	-2.654761E-07	LZ	.82	.56	.12		
55	X	3.108307E-06	XY	-2.219673E-06	A	1.451382E-05	LX	-.61	.79	-.02		-3.651943E-06	1.105123E-05
		Y	1.419658E-07	YZ	2.219673E-06	B	-3.543073E-06	LY	.11	.06	-.99		
		Z	7.705556E-06	ZX	-1.727543E-05	C	-1.491768E-08	LZ	.79	.60	.12		

- This output is typical for the CHEXA, CPENTA, CPYRAM and CTETRA elements.
- Standard STRAIN output, requested in Case Control Section.

Forces in CWELD Elements

0

SUBCASE 1

ELEMENT ID	BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL	
	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX
109	-2.135744E+02	1.586358E+02	7.728133E+01	1.390482E+01	-2.908656E+02	-1.447359E+02	-8.345229E+01	0.0
110	2.135769E+02	1.586297E+02	-7.728750E+01	1.390158E+01	2.908643E+02	-1.447281E+02	-8.346558E+01	0.0
111	-2.363556E+02	-3.536731E+02	-2.396099E+02	-2.361487E+02	3.254667E+00	1.175355E+02	-1.198793E+01	0.0
112	2.363570E+02	-3.536719E+02	2.396101E+02	-2.361489E+02	-3.253387E+00	1.175340E+02	-1.198794E+01	0.0

- Welded shell model, point-to-point option

Grid Point Force Balance

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	5.000000E+03	4.990313E+02	.0	.0	.0	.0
1	3	ROD	-2.495156E+03	-4.990313E+02	.0	.0	.0	.0
1	4	ROD	-2.504844E+03	.0	.0	.0	.0	.0
1		*TOTALS*	.0	.0	.0	.0	.0	.0
2		F-OF-SPC	-5.000000E+03	5.009687E+02	.0	.0	.0	.0
2	1	ROD	2.495156E+03	.0	.0	.0	.0	.0
2	2	ROD	2.504844E+03	-5.009687E+02	.0	.0	.0	.0
2		*TOTALS*	.0	.0	.0	.0	.0	.0
3	1	ROD	-2.495156E+03	.0	.0	.0	.0	.0
3	3	ROD	2.495156E+03	4.990313E+02	.0	.0	.0	.0
3	5	ROD	.0	-4.990313E+02	.0	.0	.0	.0
3		*TOTALS*	-4.547474E-13	-1.347189E-11	.0	.0	.0	.0

1

1

Forces and moments acting on the grid point from each source in the global coordinate system

- Only the forces due to elements, SPCs and applied loads are considered. The totals will be nonzero due to round-off error, or to the fact that forces due to MPCs, rigid elements, GENELs, DMIG, or inertia loads (in dynamic analysis) are not included.
- Standard GPFORCE output, requested in Case Control Section.

Grid Point Stress Output

GRID ID	ELEMENT ID	SURFACE X-AXIS X NORMAL(Z-AXIS) Z			REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID					
		STRESSES IN ELEMENT SYSTEM			PRINCIPAL STRESSES					
		FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	SHEAR	VON MISES
20	0	Z1	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		Z2	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		MID	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
21	0	Z1	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		Z2	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		MID	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
60	0	Z1	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		Z2	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		MID	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
61	0	Z1	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		Z2	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		MID	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03

4

5

6

7

8

1

Grid point ID where stress averaging is performed

2

Surface or volume ID referenced on the Case Control SURFACE or VOLUME command

3

Referenced coordinate system ID

4

Fiber location as specified on the PSHELL entry

5	Average component stresses at grid points in reference coordinate system
6	Angle of principal stress in referenced coordinate system
7	Major and minor principal stresses
8	Maximum shear and von Mises stresses

- Stress surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).
- Stress volume output is available for all solid elements (CHEXA, CPENTA, CPYRAM and CTETRA).

Grid Point Stress Discontinuities Output

1	SURFACE X-AXIS X NORMAL(Z-AXIS) Z					REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID					91	3
GRID	NORMAL STRESS DISCONTINUITY IN SURFACE SYSTEM				PRINCIPAL STRESS DISCONTINUITY				ERROR		CID	
ID	FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	MAJOR	MINOR	MAX SHEAR	VON MISES	EST.			
20	Z1	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02	5.345E+02			
	Z2	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02	5.345E+02			
	MID	9.247E+02	4.628E-09	4.392E+01	1.732E+00	9.230E+02	4.606E+02	9.221E+02	5.345E+02			
21	Z1	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02	5.092E+02			
	Z2	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02	5.092E+02			
	MID	8.809E+02	1.101E-09	4.392E+01	1.818E+00	8.791E+02	4.386E+02	8.782E+02	5.092E+02			
61	Z1	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02	3.775E+02			
	Z2	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02	3.775E+02			
	MID	6.531E+02	3.196E-09	3.106E+01	1.041E+01	6.441E+02	3.176E+02	6.395E+02	3.775E+02			
62	Z1	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02	3.596E+02			
	Z2	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02	3.596E+02			
	MID	6.221E+02	7.408E-10	3.106E+01	1.089E+01	6.127E+02	3.017E+02	6.079E+02	3.596E+02			

1	Grid point ID where stress discontinuities are calculated
2	Surface or volume ID referenced on the Case Control SURFACE or VOLUME command
3	Referenced coordinate system ID
4	Fiber location as specified on the PSHELL entry
5	Probable error of component stresses δ_g
6	Probable error of major principal, minor principal, maximum shear and von Mises stresses

7

Stress error measured at each grid point

- This output is obtained by using the STRFIELD, STRESS, GPSDCON, and SURFACE (or VOLUME) Case Control command.
- Grid point stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).
- Grid point stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, CPYRAM and CTETRA).

Element Stress Discontinuities Output

1		2										3	
ELEMENT		REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID										91	
ID	ELEMENT TYPE	SURFACE X-AXIS X NORMAL(Z-AXIS) Z				SYSTEM PRINCIPAL STRESS DISCONTINUITY				ERROR			
		FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	MAJOR	MINOR	MAX SHEAR	VON MISES	EST.			
19	QUAD4	Z1	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02			
		Z2	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02			
		MID	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02			
20	QUAD4	Z1	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02			
		Z2	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02			
		MID	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02			
59	QUAD4	Z1	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02			
		Z2	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02			
		MID	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02			
60	QUAD4	Z1	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02			
		Z2	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02			
		MID	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02			
4		5				6				7			

1	Element ID where element stress discontinuities are calculated
2	Surface or volume ID referenced on the Case Control SURFACE or VOLUME command
3	Referenced coordinate system ID
4	Fiber location as specified on the PSHELL entry
5	Probable error of component stresses δ_e
6	Probable error of major principal, minor principal, maximum shear and von Mises stresses
7	Stress error measured at each element

- This output is obtained by using the STRFIELD, STRESS, ELSDCON, and SURFACE (or VOLUME) Case Control commands.

- Element stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).
- Element stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, CPYRAM and CTETRA).

Element Strain Energy

```

ELEMENT-TYPE = ROD          * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM = 7.578219E+00
SUBCASE          2          TOTAL ENERGY OF ALL ELEMENTS IN SET   -1 = 7.578219E+00

```

ELEMENT-ID	STRAIN-ENERGY	PERCENT OF TOTAL	STRAIN-ENERGY-DENSITY
1	5.742940E+00	75.7822	2.392892E-03
2	6.220422E-01	8.2083	2.541511E-04
3	6.220422E-01	8.2083	2.541511E-04
4	5.865024E-01	7.7393	2.443760E-04
5	4.692019E-03	.0619	9.775041E-06

```

TYPE = ROD      SUBTOTAL      7.578219E+00      100.0000

```

Annotations: 1 points to the SUBTOTAL line; 2 points to the STRAIN-ENERGY column; 3 points to the total energy values at the top.

1	Total strain energy for all elements of this type in the requested set
2	Strain energy in each element
3	Total strain energy in requested SET (SET –1 denotes ALL)

- This output is incorrect in the presence of thermal loads or element deformations.
- Standard output, requested with Case Control command ESE.

Sequence Processor Output

```

1
-----
ELEMENT TYPE      NUMBER      ASSEMBLY TIME(SEC):
HEXA              4              .07
-----
TOTAL MATRIX ASSEMBLY TIME FOR      4 ELEMENTS IS      .07 SECONDS.
ORIGINAL PERFORMANCE DATA
SUPER(GROUP) ID   NO. GRIDS   AV. CONNECTIVITY   C-AVERAGE   C-RMS   C-MAXIMUM   P-GROUPS   P-AVERAGE   DECOMP TIME(SEC)
(6.0 DOF/GRID )
0                 20         10.40             6.70        7.20        11          0           .00          .017
RESEQUENCED PERFORMANCE DATA
SUPER(GROUP) ID   NO. GRIDS   AV. CONNECTIVITY   C-AVERAGE   C-RMS   C-MAXIMUM   P-GROUPS   P-AVERAGE   DECOMP TIME(SEC)   METHOD
(6.0 DOF/GRID )
0                 20         10.40             5.70        6.02         8          0           .00          .012          ACTIVE

```

Annotations: 1 points to the header line; 2 points to the C-AVERAGE, C-RMS, and C-MAXIMUM columns; 3 points to the METHOD column.

1	Connection data
---	-----------------

2	Average, RMS, and maximum number of active columns
3	Resequencer method selected

- There are 20 points divided into one group.
- The average and RMS values depend on the manner in which the grid points are sequenced.
- This output is available only if the sequencer is turned on (param,oldseq,x).
- Further information can be found in the NX Nastran Numerical Methods User's Guide.

Grid Point Singularity Table


GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET		NEW USET			
EXCLUSIVE	UNION	EXCLUSIVE	UNION						
53	G	4	0.00E+00	B	F	SB	S	*	
53	G	5	0.00E+00	B	F	SB	S	*	
53	G	6	0.00E+00	B	F	SB	S	*	
54	G	4	0.00E+00	B	F	SB	S	*	
54	G	5	0.00E+00	B	F	SB	S	*	
54	G	6	0.00E+00	B	F	SB	S	*	
57	G	4	0.00E+00	B	F	SB	S	*	
57	G	5	0.00E+00	B	F	SB	S	*	
57	G	6	0.00E+00	B	F	SB	S	*	
58	G	4	0.00E+00	B	F	SB	S	*	
58	G	5	0.00E+00	B	F	SB	S	*	
58	G	6	0.00E+00	B	F	SB	S	*	

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET		NEW USET			
EXCLUSIVE	UNION	EXCLUSIVE	UNION						
53	G	4	0.00E+00	B	F	SB	S	*	
53	G	5	0.00E+00	B	F	SB	S	*	
53	G	6	0.00E+00	B	F	SB	S	*	
54	G	4	0.00E+00	B	F	SB	S	*	
54	G	5	0.00E+00	B	F	SB	S	*	
54	G	6	0.00E+00	B	F	SB	S	*	
57	G	4	0.00E+00	B	F	SB	S	*	
57	G	5	0.00E+00	B	F	SB	S	*	
57	G	6	0.00E+00	B	F	SB	S	*	
58	G	4	0.00E+00	B	F	SB	S	*	
58	G	5	0.00E+00	B	F	SB	S	*	
58	G	6	0.00E+00	B	F	SB	S	*	

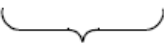
- This is automatically output from the Grid Point Singularity Processor (GPSP1) and is obtained anytime a stiffness ratio is less than the parameter EPZERO (default = 10^{-8}).

User Information Messages 5293 for Data Block KLL

LOAD SEQ. NO.	EPSILON	EXTERNAL WORK	EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
1	-4.5888836E-15	6.4518387E+01	
2	-2.8991481E-15	7.5782189E+00	



1



2

<p>1</p>	<p>Residual vector error</p> $\varepsilon = \frac{\{u_I\}^T \{\delta P_I\}}{\{u_I\}^T \{P_I\}}$ <p>Where:</p> <p>$\{P_I\}$ = Load on the I-set degrees of freedom</p> <p>$\{u_I\}$ = Displacement of the I-set degrees of freedom</p> <p>$[K_{II}]$ = Stiffness matrix of the I-set</p> <p>$\{\delta P_I\} = [K_{II}]\{u_I\} - \{P_I\}$</p>
<p>2</p>	<p>Work performed by the applied loads = $\frac{1}{2} P_I u_I$</p>

- Small epsilons are due to machine roundoff are acceptable. Large epsilons indicate a potential modeling error.
- One line of output is printed for each static loading condition.

OLOAD Resultant

SUBCASE/ DAREA ID		LOAD TYPE	OLOAD			RESULTANT		
			T1	T2	T3	R1	R2	R3
0	1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	3.307500E+02	0.000000E+00	-9.268750E+02	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-9.268750E+02	0.000000E+00

1

2

①	Subcase number
②	Resultant of the applied loads about the point specified on the "PARAM,GRDPNT,x". The default is at the origin of the basic coordinate system.

- One line is printed for each subcase.

- This output is printed by default. It can be deactivated by adding PARAM,PRTRESLT,NO in the bulk data.
- The OLOAD output table sums all the applied loads to a single location. Fx, Fy, Fz, Mx, My, Mz are relative to the applied loads in the basic coordinate system directions. T1, T2, T3, R1, R2, R3 are the basic coordinate directions of the summed loads in the table. The origin of the summed loads is specified with the PARAM,GRDPNT,x entry which is the origin of the basic coordinate system by default.

SPCFORCE Resultant

```

0
SUBCASE/      LOAD
DAREA ID  TYPE      T1      T2      T3      R1      R2      R3
0      1  FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY  ----  0.000000E+00  ----  ----  0.000000E+00  ----  0.000000E+00
      FZ  ----  ----  -1.000000E+02  0.000000E+00  0.000000E+00  ----
      MX  ----  ----  ----  0.000000E+00  ----  ----
      MY  ----  ----  ----  ----  5.000000E+02  ----
      MZ  ----  ----  ----  ----  ----  0.000000E+00
      TOTALS 0.000000E+00  0.000000E+00 -1.000000E+02  0.000000E+00  5.000000E+02  0.000000E+00
0      2  FX  0.000000E+00  ----  ----  ----  0.000000E+00  0.000000E+00
      FY  ----  0.000000E+00  ----  ----  0.000000E+00  ----  0.000000E+00
      FZ  ----  ----  -3.307500E+02  0.000000E+00  0.000000E+00  ----
      MX  ----  ----  ----  0.000000E+00  ----  ----
      MY  ----  ----  ----  ----  8.268750E+02  ----
      MZ  ----  ----  ----  ----  ----  0.000000E+00
      TOTALS 0.000000E+00  0.000000E+00 -3.307500E+02  0.000000E+00  8.268750E+02  0.000000E+00

```



1	Subcase number
2	Resultant of the applied loads about the point specified on the "PARAM,GRDPNT,x". The default is at the origin of the basic coordinate system

- One line is printed for each subcase.
- This output can be obtained by adding PARAM,PRTRESLT,YES in the bulk data file.

Maximum SPCFORCES, DISPLACEMENT, and Applied Loads

```

MAXIMUM SPCFORCES
      T1      T2      T3      R1      R2      R3
1  5.000000E+03  5.0096872E+02  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
2  2.0000000E+03  9.6871246E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
MAXIMUM DISPLACEMENTS
      T1      T2      T3      R1      R2      R3
1  1.2524217E-02  1.2903677E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
2  7.5782188E-03  2.5048435E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
MAXIMUM APPLIED LOADS
      T1      T2      T3      R1      R2      R3
1  0.0000000E+00  1.0000000E+03  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
2  2.0000000E+03  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00

```



1	Subcase number
2	<ol style="list-style-type: none"> 1. Quantities are transformed to the basic coordinate system. 2. Absolute value of transformed quantities are taken 3. Largest value for each component is found and printed

- The largest magnitude of quantities transformed into the basic coordinate system is printed. The maximums shown for T1 may not be at the same grid point as T2, etc.
- One line is printed for each subcase.
- This output can be obtained by adding PARAM,PRTMAXIM,YES in the bulk data file.

Displacement Vectors

SUBCASE 2

D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	.0	.0	.0	.0	
3	G	7.578219E-03	-2.504843E-02	.0	.0	.0	.0	
4	G	-2.421781E-03	-2.495156E-02	.0	.0	.0	.0	

1	G for grid point, S for scalar point
2	Displacements expressed in the global coordinate system

- Standard DISPLACEMENT output for a static analysis, requested in Case Control Section.

Load Vectors

SUBCASE 1

L O A D V E C T O R							
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
4	G	.0	-1.000000E+03	.0	.0	.0	.0

1	Applied loads and moments expressed in the global coordinate system
---	---

- Standard OLOAD output, requested in Case Control Section.

Database Summary Table

*** TOTAL MEMORY AND DISK USAGE STATISTICS ***

HIWATER (WORDS)	DAY_TIME	SUB_DMAP NAME	DMAP MODULE
408922	09:56:52	SEKRRS	18 DCMF

*** DATABASE USAGE STATISTICS ***

DBSET	ALLOCATED (BLOCKS)	BLOCKSIZE (WORDS)	USED (BLOCKS)	USED %
MASTER	5000	2048	149	2.98
DBALL	250000	2048	9	.00
OBJSCR	5000	2048	130	2.60
SCRATCH	500100	2048	20	.00

*** BUFFER POOL AND SCRATCH 300 USAGE STATISTICS ***

OPTION SELECTED	BLOCKS ALLOCATED	BLOCKS REUSED	BLOCKS RELEASED
GINO,EXEC	37	2729	768

*** MAXIMUM DISK USAGE ***

HIWATER (MB)	DAY_TIME	SUB_DMAP NAME	DMAP MODULE
2.266	09:56:54	SESTATIC	186 EXIT

*** DBSET FILES ***

FILE	ALLOCATED (BLOCKS)	HIWATER (BLOCKS)	HIWATER (MB)	I/O TRANSFERRED (GB)
MASTER	5000	149	1.164	.010
DBALL	250000	9	.070	.000
OBJSCR	5000	130	1.016	.003
(MEMFILE	100	62	.484	.000)
SCRATCH	250000	1	.008	.000
SCR300	250000	1	.008	.000
TOTAL:				.013

*** SCRATCH 300 ***

OPTION SELECTED	HIWATER (BLOCKS)	DAY_TIME	SUB_DMAP NAME	DMAP MODULE	OPN/CLS COUNTER
2	1	9:56:40	PREFACE	0 PREFACE	0

1	Maximum disk space allocated in GINO blocks
2	Maximum disk space used in GINO blocks
3	Disk spaced used in megabytes
4	Amount of I/O transferred between each DBset and memory
5	Maximum disk space used by SCR300 in GINO blocks

- Standard OLOAD output, requested in Case Control Section.

Restartable Versions

PROJECT_ID	ASSIGNED INT. VALUE	VERSION_ID	CREATION TIME
0 " B L A N K "	1	** 1	3/ 1/93 10:44.42
		** 2	3/ 1/93 10:45.37
		3	3/ 1/93 10:46. 5
		** 4	3/ 1/93 10:46.38
		** 5	3/ 1/93 10:47. 2
		** 6	3/ 1/93 10:47.25
		7	3/ 1/93 10:47.49

1	There were seven versions created in the database. Only Versions 3 and 7 are, however, restartable. The other versions denoted by a "***" are not restartable.
---	--

4.7 Understanding the .DBALL and .MASTER Files

The .DBALL and .MASTER files are database files (DBsets) that NX Nastran generates automatically during a solve. The files are generated in the scratch directory. These files are necessary if you want to later perform a restart analysis.

- The .MASTER file is a directory of the database. It contains the names of all DBsets, DBset members and their physical file names, and a directory of projects, versions, data blocks, parameters, DMAP source and object files. It also contains the Nastran Data Definition Language (NDDL) scheme used to describe the database. The default maximum size for .MASTER is 5,000 blocks.
- The .DBALL file contains all the DMAP data blocks you can save for reuse in a subsequent run. In the solution sequences that don't allow restarts, such as SOL 101, this file is empty. The default size for .DBALL is 250,000 blocks.

Specifying Whether to Save .DBALL and .MASTER Files

The “scratch” option on the nastran command line allows you to specify whether you want the software to save the files .DBALL and .MASTER files. By default, NX Nastran deletes the files at the end of the run.

- If you specify “scratch=yes,” the software deletes the files.
- If you specify “scratch=no,” the software keeps the files.
- If you specify “scratch=mini,” the software retains a reduced size database that you can use to perform data recovery restarts.

See also

- “scratch” in the *NX Nastran Quick Reference Guide*
- “Mini-Database Restarts” in the *NX Nastran User's Guide*
- “Understanding DBsets” in the *NX Nastran User's Guide*

Manually Deleting .DBALL and .MASTER Files

In some cases, you may need to manually delete the .DBALL and .MASTER files. For example:

- If your solve doesn't complete successfully, NX Nastran may not automatically delete the .DBALL and .MASTER files, even though you specified “scratch=yes.” In those cases, you must delete the files manually before resubmitting your job.
- If you specified “scratch=no,” but you decide that you want to submit a .dat file a second time as a cold start (you don't use any restart commands), you must first delete the corresponding .DBALL and .MASTER files before you resubmit the job.

4.8 Understanding the .xdb and .op2 Files

The .xdb and .op2 files are binary files that contain NX Nastran results for use with certain types of postprocessors.

- The .xdb file is known as the “results database” file. It contains results that can be postprocessed by certain postprocessors.
- The .op2 file is known as an “OUTPUT2” file. It is written to FORTRAN unit 12. Also, if an .op2 file already exists for the run, it too should be deleted prior to resubmitting the run. It contains results that can be postprocessed by certain postprocessors, such as NX MasterFEM (formerly known as I-DEAS MasterFEM), NX Advanced Simulation, and MSC.Patran.

The setting for the PARAM,POST option in the Bulk Data section of the input file controls whether NX Nastran creates an .xdb or op2 file. For example, to have the software create an .xdb file, you would specify PARAM,POST,0. Additionally, the options you specify for PARAM,POST control whether the file contains geometry.

Note: Before you perform a restart analysis, you should delete the .xdb and .op2 files (if any) from your original run.

See also

- “POST” in the *NX Nastran Quick Reference Guide*

Chapter 5: Defining Grid Points and Scalar Points

- *Overview of Grid Points and Scalar Points*
- *Defining Grid Points*
- *Specifying Grid Point Properties*
- *Understanding Scalar Points*

5.1 Overview of Grid Points and Scalar Points

Whenever you analyze a finite element model using NX Nastran, you must generate a set of grid points (nodes). Those grid points, together with the elements, define the size and shape of your model. The physical behavior of your finite element model is represented by the elements connecting the grid points as well as the associated constraints and their coordinate system.

When you create a finite element model of a structure, you're actually constructing a mathematical model that represents your structure in matrix form. The unknowns in the matrix equation are the displacements in the model. These displacements consist of:

- The six components for each of the grid points
- One component for each of the scalar points

In general, the displacements at the grid points and the scalar points are referred to as the model's degrees of freedom.

This chapter describes grid points, grid point properties, and scalar points. Detailed information about coordinate systems is provided in the next chapter.

For many practical modeling situations, you may have several different output coordinate systems in a model.

See also

- “[Understanding Coordinate Systems](#)” in the *NX Nastran User's Guide*

5.2 Defining Grid Points

Each grid point has six degrees of freedom (DOF): three translations and three rotations. The six degrees of freedom are identified as 1, 2, 3, 4, 5, and 6, as shown below.

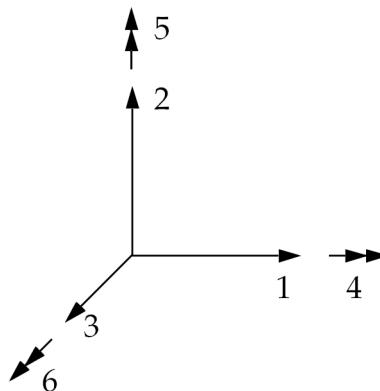


Figure 5-1. DOF Nomenclature

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

DOF 1 = T1 = U_1 = translation in direction 1

DOF 2 = T2 = U_2 = translation in direction 2

DOF 3 = $T_3 = U_3$ = translation in direction 3

DOF 4 = $T_4 = \theta_4$ = rotation in direction 4

DOF 5 = $T_5 = \theta_5$ = rotation in direction 5

DOF 6 = $T_6 = \theta_6$ = rotation in direction 6

When you define a location of the grid point in space using these components, you are inherently defining the point relative to a coordinate system.

In NX Nastran, you use the GRID bulk data entry to define a grid point. GRID allows you to:

- Assign a unique ID to the grid point
- Specify the location of the grid point in space with respect to a reference coordinate system
- Assign permanent constraints to the grid point
- Define the directions of motions at the grid point

Format of the GRID Entry

The format of the grid point entry is as follows:

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

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

See also

- “GRID” in the *NX Nastran Quick Reference Guide*

Specifying the Grid Point ID (Field 1)

Field 1 lets you assign an ID to the grid point. That ID must be unique with respect to all other grid points and scalar points in your model. In NX Nastran, grid point IDs must begin at 1, however, they don't have to be contiguous.

You will often find it helpful to assign groups of IDs to the different sections of your model. For example, if you're working on an automotive FE model, you may assign IDs in the range from 1 to 1000 for the grid points for the driver's door and assign IDs in the range from 5001 to 6000 for the grid points for the hood. A clear and logical scheme for numbering grid points can be very helpful on complex models.

Defining Grid Point Coordinate Systems (Fields 3 and 5)

In NX Nastran, each grid point refers to two coordinate systems, CP and CD:

- The CP system that you specify in field 3 is used to locate the grid point in space. This is known as the “position” coordinate system. It defines the grid point's location in space.
- The CD system that you specify in field 7 is used to establish the grid point's displacement (output) coordinate system. The CD system defines the direction of displacements, constraints, and other grid point related quantities such as reaction forces. It is the system in which the software outputs the results.

NX Nastran has two main categories of coordinate system:

- The basic coordinate system (the default for the CP and CD fields)
- Local coordinate systems, which you define yourself. These can be rectangular, cylindrical, or spherical

With the CP and CD fields, you can use either the basic coordinate system or a local coordinate system.

Specifying the Location Coordinate System in the CP Field

To specify the coordinate system for the location of the grid point, enter the ID of that coordinate system in the CP field (Field 3). If you leave the CP field blank, or enter a 0 in the CP field, the software defines the grid point in the basic coordinate system.

X1, X2, and X3 in fields 4-6 let you define the location of the grid point in the CP coordinate system you specified in field 3. They have the following meanings for different types of coordinate systems:

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ (degrees)	Z
Spherical	R	θ (degrees)	ϕ (degrees)

Specifying the Output Coordinate System in the CD Field

To specify the coordinate system in which you want NX Nastran to generate and output the results at grid points (displacements, grid point forces, etc.), enter the ID of the coordinate system in the CD field (Field 7). For many practical modeling situations, you may have several different output coordinate systems in a single model.

Although the CD field is referred to as the displacement coordinate system, it is more than the coordinate system used to output the results. It has a fundamental role in NX Nastran. The collection of all the coordinate systems defined on the CD field of all the grid points constitutes the global coordinate system. It is important to note that the global coordinate system is not necessarily a

single coordinate system but rather the collection of all of your CD coordinate systems. NX Nastran assembles the global stiffness matrix in the global coordinate system.

See also

- “Comparing the Basic and Global Coordinate Systems” in the *NX Nastran User's Guide*

Understanding How the CP and CD Fields Work Together

Although you can have the CP and CD fields reference the same coordinate system, you can also have them reference different coordinate systems. For example, you may want to specify a local coordinate system to define the location of the grid point in the CP field, and then leave the CD field blank or enter a 0 to output your results in the basic coordinate system. On the other hand, if you need to output the displacement information in particular coordinate system, you enter the ID of that local coordinate system in the CD field.

Defining the Directions of Displacement (Fields 4-6)

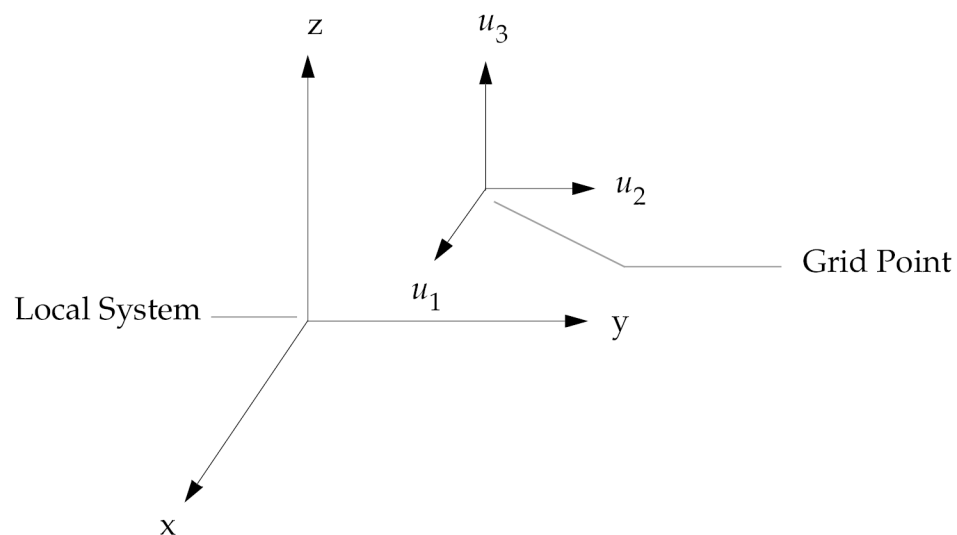
In NX Nastran, each grid point has six degrees of freedom or “displacement components”; three translational and three rotational. The degrees of freedom are denoted as u_1 , u_2 , u_3 , θ_1 , θ_2 , and θ_3 , or as T1, T2, T3, R1, R2, and R3.

- T1, T2, and T3 are the three orthogonal components of translation parallel to the 1, 2, and 3 directions of the grid point's displacement coordinate system, respectively.
- R1, R2, and R3 are the components of rotation in the same directions. The center of rotation is at the grid point location.

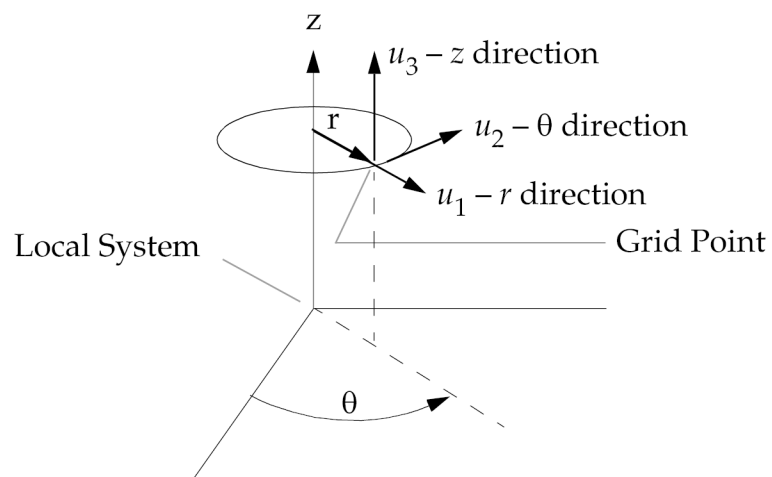
The coordinate system you specify in the CD field determines the orientation of these displacement components.

- If the system you specify in the CD field is rectangular, the displacement components are parallel to the local system as indicated in (a) below.
- If the system you specify in the CD field is cylindrical, the displacement components are in the radial, tangential, and axial directions as indicated in (b) below.
- If the system you specify in the CD field is spherical, the displacement components are in the radial, meridional, and azimuthal directions as indicated in (c) below.

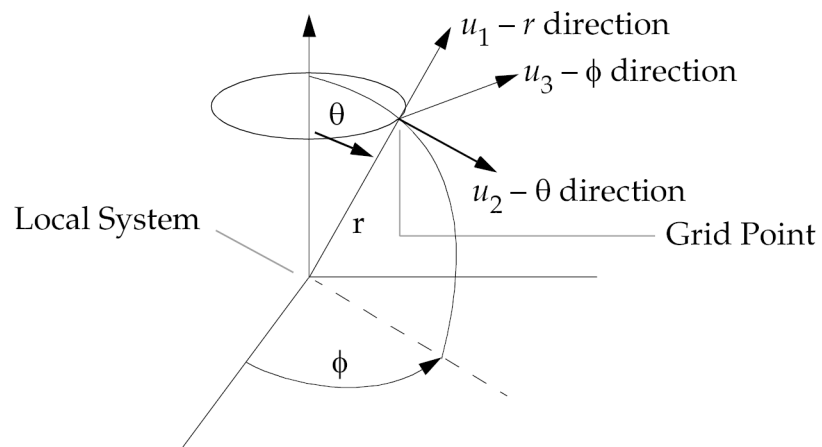
(a) Rectangular



(b) Cylindrical



(c) Spherical

**Figure 5-2. Displacement Components**

Specifying Permanent Single Point Constraints (Field 8)

You use field 8 (PS) of the GRID entry to specify permanent single point constraints to constrain any or all the degrees of freedom associated with the grid point. In this field, you enter the integers (1-6) that correspond to the degrees of freedom you want to constrain.

The constraints that you define in field 8 are considered permanent because you can't change them during the run. Any degrees of freedom you specify in the PS field are constrained for all subcases. Constraints specified on the GRID entry are usually restricted to those degrees-of-freedom that will not be elastically constrained and hence must be removed from the model in order to avoid singularities in the stiffness matrix.

In many situations, you want to analyze your structure using more than one set of constraints. For these situations, don't specify the constraints on the GRID entry. Instead, specify the constraints using an SPC bulk data entry (Single-Point Constraint). The only difference from using the GRID entry is that constraints applied using an SPC entry must be selected through the Case Control section. Therefore, you can apply different SPC entries for different subcases.

NX Nastran applies the constraints you define in field 8 in the output coordinate system (CD), which you defined in field 7. This gives you greater flexibility for modeling a variety of joints and boundary conditions.

Note: NX Nastran applies all constraints, whether you define them with an SPC bulk data entry or using the PS field of the GRID entry, in the output coordinate system.

See also

- “**SPC**” in the *NX Nastran Quick Reference Guide*
- “Single-Point Constraints” in the *NX Nastran User's Guide*

Defining the Superelement ID (Field 9)

The SEID option in field 9 lets you define superelements within the bulk data section of your input file.

See also

- “**Superelement Definition**” in the *NX Nastran Superelement Users's Guide*

Understanding the GRDSET Entry

If many or all of the GRID entries in your model have the same values in the location coordinate system (CP), displacement coordinate system (CD), single point constraint (PS) fields, or superelement ID (SEID), you can use the GRDSET bulk data entry to avoid having to repeatedly specify the same information. If you specify a value for any of these fields on the GRDSET entry, the software then uses those values to replace corresponding blank fields on any GRID entry in your model.

GRDSET is useful in the case of such problems as space trusses where you want to remove all the rotational degrees-of-freedom. It is also helpful with plane structures where you want to remove all out-of-plane or all in-plane motions.

Note: You can only include a single GRDSET entry in the bulk data section of your input file.

See also

- “**GRDSET**” in the *NX Nastran Quick Reference Guide*

5.3 Specifying Grid Point Properties

In NX Nastran, you specify some of the characteristics of the model as properties of grid points, rather than as properties of elements.

Defining Mass, Damping, and Stiffness through Direct Input Matrices

You use direct input matrices to define mass, damping, and stiffness properties either in part or entirely, as properties of grid points.

- You can use the DMIG bulk data entry define direct input matrices. You can associate these matrices with components of geometric grid points, scalar points or extra points introduced for dynamic analysis.
- You can use the TF bulk data entry define transfer functions that NX Nastran automatically converts to direct matrix input.
- The DMIAX entry is an alternate form of direct matrix input that is used for hydroelastic problems (see).

See also

- “**DMIG**” in the *NX Nastran Quick Reference Guide*
- “**TF**” in the *NX Nastran Quick Reference Guide*
- “**DMIAX**” in the *NX Nastran Quick Reference Guide*
- “**Overview of Coupled Fluid-Structure Interaction**” in the *NX Nastran User's Guide*

Defining Mass through the Concentrated Mass Element

You can also use the concentrated mass element to input mass properties as properties of grid points.

- You can use the CONM1 entry to define a 6×6 matrix of mass coefficients at a geometric grid point in any selected coordinate system.
- You can use the CONM2 entry to define a concentrated mass at a geometric grid point in terms of its mass, the three coordinates of its center of gravity, the three moments of inertia about its center of gravity, and its three products of inertia, referred to any selected coordinate system.

See also

- “**CONM1**” in the *NX Nastran Quick Reference Guide*
- “**CONM2**” in the *NX Nastran Quick Reference Guide*

Defining Temperatures at Grid Points

You can define thermal fields by specifying the temperatures at grid points.

- You use TEMP bulk data entry to specify the temperature at grid points for use in connection with thermal loading and temperature-dependent material properties.
- You use the TEMPD bulk data entry to specify a default temperature to avoid a large number of duplicate entries on a TEMP entry when the temperature is uniform over a large portion of the structure.
- You use the TEMPAX bulk data entry for conical shell problems.

See also

- “TEMP” in the *NX Nastran Quick Reference Guide*
- “TEMPD” in the *NX Nastran Quick Reference Guide*
- “TEMPAX” in the *NX Nastran Quick Reference Guide*

Assigning a Singularity Normal to a Degree-of-Freedom

The SNORM Bulk Data entry and/or the PARAM,SNORM assigns the singularity normal of a shell element to a degree-of-freedom. NX Nastran then uses the AUTOSPC capability to remove this singularity.

See also

- “SNORM” in the *NX Nastran Quick Reference Guide*
- “Automatically Applying Single-Point Constraints” in the *NX Nastran User's Guide*

5.4 Understanding Scalar Points

In NX Nastran, a scalar point has only one degree of freedom. A scalar point is similar to a grid point in the way that NX Nastran assembles it into the matrix equation. However, a scalar point has no spatial orientation, so it does not require a coordinate system. It is simply an additional degree of freedom that you can define in your model. Scalar points are useful for modeling with scalar elements, multipoint constraints (MPCs), computing relative and average motions, and defining the warping coefficients of a thin-walled beam.

Comparing Grid Points to Scalar Points

While you can use a grid point as a scalar point, a grid point requires more input and requires you to constrain five of the six degrees of freedom. Consequently, using scalar points instead of a grid point is more efficient, both computationally and in terms of the actual input required.

Defining Scalar Points

In NX Nastran, you define use the SPOINT bulk data entry to define a scalar point. You can specify up to eight scalar points on a single SPOINT entry. You can also implicitly define scalar points on a connection entry for a scalar element SPOINT entries are used primarily to define scalar points appearing in constraint equations but to which no structural elements are connected.

See also

- “**SPOINT**” in the *NX Nastran Quick Reference Guide*

Using Scalar Points as Extra Points for Dynamics

You can also introduce special scalar points, called “extra points,” for dynamic analyses. Extra points are used in connection with transfer functions and other forms of direct matrix input used in dynamic analyses and are defined on EPOINT entries.

See also

- “**DMIGs, Extra Points, and Transfer Functions**” in the *NX Nastran Advanced Dynamic Analysis User's Guide*

Scalar Point Output Format

Scalar point output is formatted differently in f06 files as compared to punch files. There are four scalar points in the example below labeled 101, 102, 103 and 104 to demonstrate these formats.

Scalar point results from an *.f06 file:

POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R				
			T2	T3		R1	R2		R3	
1	G	0.0	0.0	0.0	0.0	0.0	0.0	0.0		
2	G	0.0	0.0	1.228000E-04	0.0	-2.333333E-05		0.0		
3	G	0.0	0.0	4.456000E-04	0.0	-4.000000E-05		0.0		
40	G	0.0	0.0	9.017334E-04	0.0	-5.000000E-05		0.0		
50	G	0.0	0.0	1.424533E-03	0.0	-5.333333E-05		0.0		
100	G	1.024533E-03	1.870133E-03	0.0	0.0	0.0	0.0	0.0		
101	S	1.024533E-03	1.870133E-03	1.024533E-03	1.870133E-03					

In the f06 displacement results shown above, the “TYPE” column determines if the data is for a grid point “G” or a scalar point “S”. Grid points have six degrees of freedom, whereas the scalar points have only one. To reduce the f06 file, NX Nastran prints up to six scalar point outputs per line. The ID at the beginning of a row of scalar point values corresponds to the first scalar point output listed at the start of that line. In the above example, scalar point “101” corresponds to the value listed at the start of the line, then the subsequent values correspond to the single output for scalar points 102, 103 and 104.

Scalar point results from a punch file:

\$DISPLACEMENTS					4
\$REAL OUTPUT					5
\$\$SUBCASE ID =					6
1	G	0.000000E+00	0.000000E+00	0.000000E+00	7
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00	8
2	G	0.000000E+00	0.000000E+00	1.228000E-04	9
-CONT-		0.000000E+00	-2.333333E-05	0.000000E+00	10
3	G	0.000000E+00	0.000000E+00	4.456000E-04	11
-CONT-		0.000000E+00	-4.000000E-05	0.000000E+00	12
40	G	0.000000E+00	0.000000E+00	9.017334E-04	13
-CONT-		0.000000E+00	-5.000000E-05	0.000000E+00	14
50	G	0.000000E+00	0.000000E+00	1.424533E-03	15
-CONT-		0.000000E+00	-5.333333E-05	0.000000E+00	16
100	G	1.024533E-03	1.870133E-03	0.000000E+00	17
-CONT-		0.000000E+00	0.000000E+00	0.000000E+00	18

	101	S	1.024533E-03	0.000000E+00	0.000000E+00	19
-CONT-			0.000000E+00	0.000000E+00	0.000000E+00	20
	102	S	1.870133E-03	0.000000E+00	0.000000E+00	21
-CONT-			0.000000E+00	0.000000E+00	0.000000E+00	22
	103	S	1.024533E-03	0.000000E+00	0.000000E+00	23
-CONT-			0.000000E+00	0.000000E+00	0.000000E+00	24
	104	S	1.870133E-03	0.000000E+00	0.000000E+00	25
-CONT-			0.000000E+00	0.000000E+00	0.000000E+00	26

As shown in the punch file example above, a new row begins for each scalar point output beginning with the ID. Since a scalar point has one degree of freedom, only the first value is significant and the remaining 5 zero values can be ignored.

The modal degree of freedom output from a dynamic response solution is also a scalar quantity and has the same format as the scalar point output described above, except it will have "M" under the "TYPE" column.

Chapter 6: Understanding Coordinate Systems

- *Overview of Coordinate Systems in NX Nastran*
- *Understanding the Basic Coordinate System*
- *Defining a Local Coordinate System*
- *Understanding the Element and Material Coordinate System*
- *Other Special Coordinate System Entries*

6.1 Overview of Coordinate Systems in NX Nastran

In NX Nastran, there are two general types of coordinate systems:

- The “basic” coordinate system, which is an implicitly defined reference coordinate system
- “Local” coordinate systems, which you define explicitly

Both of these different types of coordinate system is described in more detail below.

6.2 Understanding the Basic Coordinate System

NX Nastran has a built-in Cartesian system called the basic coordinate system. It is also called the default coordinate system. The basic system is a rectangular system, except when you're using axisymmetric elements.

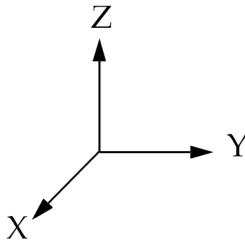


Figure 6-1. The Basic Coordinate System

ID of the Basic Coordinate System

In NX Nastran, all coordinate systems have a coordinate system identification number (CID). The basic coordinate system's identification number is zero (0) or blank.

Comparing the Basic and Global Coordinate Systems

Many finite element textbooks refer to a “global coordinate system.” However, this term means something different in NX Nastran. It is important to understand the NX Nastran definition of the global coordinate system as it differs from the classical textbook approach. These differences in terminology can often be confusing to new NX Nastran users.

In the classical approach, individual element stiffness matrices are generated for all the elements. Those individual elemental matrices are then transformed into a single stiffness matrix commonly referred to as the global stiffness matrix. Each row and column of this global stiffness matrix represents a degree of freedom of the structural model. The coordinate system associated with each one of these degrees of freedom is the same and is known as the global coordinate system.

In NX Nastran, the software generates the element stiffness matrix similarly to the classical approach. NX Nastran assembles all the individual matrices into the global stiffness matrix. However, instead of transforming the element stiffness matrices into a unique coordinate system, NX Nastran transforms these matrices into the coordinate system that you specified in the CD field of the GRID entry for the grid points to which each element is attached. NX Nastran then assembles the individual elemental stiffness matrices to form the global stiffness matrix.

Therefore, in NX Nastran, the “global” coordinate system isn't a unique coordinate system. It is simply the collection of all the CD coordinate systems specified on all of the GRID entries. This approach facilitates a variety of elegant modeling techniques. If you prefer to have the global coordinate system be the same as the basic coordinate system, which is a unique system, you can leave the CD field blank for all the grid points in your model.

Figure 6-2 shows the coordinate system definition where:

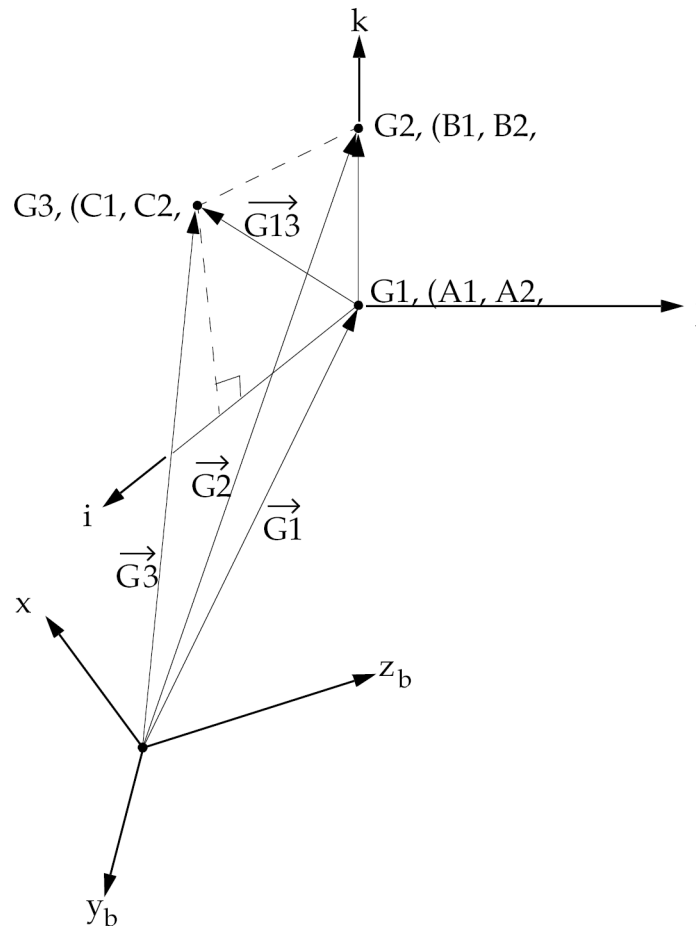


Figure 6-2. Coordinate System Definition

$$\begin{aligned}
 (i, j, k) &= \text{local (user defined) coordinate system } (x, y, z) \text{ or } (r, \theta, z) \text{ or } (r, \theta, \phi) \\
 (x_b, y_b, z_b) &= \text{basic coordinate system} \\
 \vec{k} &= (\vec{G2} - \vec{G1}) \text{ normalized} \\
 \vec{G13} &= (\vec{G3} - \vec{G1}) \\
 \vec{j} &= (\vec{k} \times \vec{G13}) \text{ normalized} \\
 \vec{i} &= (\vec{j} \times \vec{k})
 \end{aligned}$$

The GRID bulk data entry lets you define six rectangular displacement components (three translations and three rotations) are defined at each grid point. You also specify both the location coordinate system and the displacement coordinate system are specified on the GRID entry for each geometric grid point.

See also

- “[Defining Grid Points](#)” in the *NX Nastran User's Guide*

6.3 Defining a Local Coordinate System

Because you may need to create your model, or portions of your model, in a coordinate system other than the basic coordinate system, NX Nastran also lets you construct alternate coordinate systems. In NX Nastran, such coordinate systems are known as “local” coordinate systems. Local coordinate systems may be:

- Rectangular
- Cylindrical
- Spherical

You can relate local coordinate systems either directly or indirectly to the basic coordinate system. Local coordinate systems play an important role in generating your models, solving for the solution, and displaying the results.

For example, consider the farm silo in [Figure 6-3](#). It is cylindrical with a hemispherical dome. It is offset from the origin of the basic coordinate system. Creating local cylindrical (r, θ, z) and spherical (r, θ, ϕ) systems makes it easier to define model geometry or review displacement results.

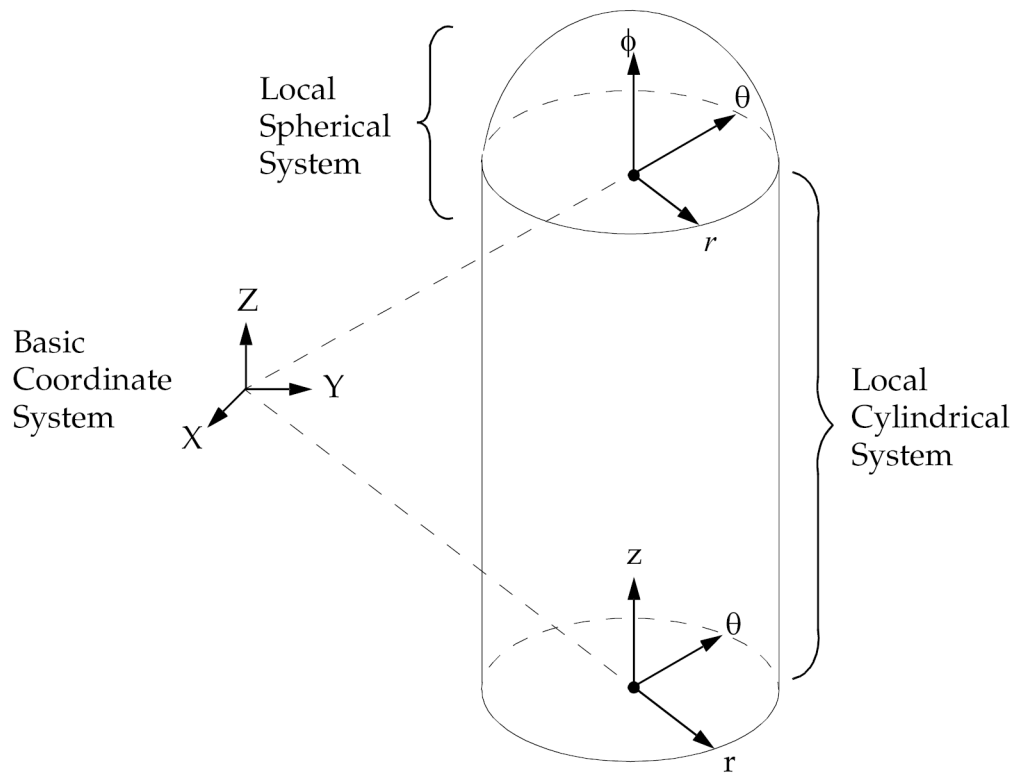


Figure 6-3. Example of Local Coordinate Systems to Aid in Establishing Model Geometry

Note: In cylindrical and spherical coordinate systems, angular input is in degrees. For example, when defining the position of a grid point with a cylindrical coordinate system, the X2 field on the GRID entry is entered in degrees. Although, rotational displacements defined with the SPCD entry and rotational displacement output relative to any coordinate system are in radians.

Bulk Data Entries for Defining Coordinate Systems

In NX Nastran, there are six different bulk data entries that you can use to define local coordinate systems:

CORD1R	}	Rectangular
CORD2R		
CORD1C	}	Cylindrical
CORD2C		
CORD1S	}	Spherical
CORD2S		

With these different bulk data entries, you can define coordinate systems by referencing:

- Three existing grid points
- Three existing grid points in an existing reference coordinate system

Defining a Coordinate System by Referencing Existing Grid Points

The CORD1R, CORD1C, and CORD1S entries let you define a local coordinate system by referencing three defined grid points. This is also known as the “Method 1” approach of coordinate system definition in NX Nastran.

The method for defining a coordinate system with CORD1R, CORD1C, or CORD1S is very similar. With each of these bulk data entries, you must:

- Specify a unique coordinate system ID (CID) in field 2 of the entry.
- Specify the IDs for each of the grid points that you want to reference. The three points must be noncolinear and can't be coincident. NX Nastran's geometry processor checks the points to ensure that they're not colinear.

Additionally, with the CORD1R and CORD1C entries, you can define up to two coordinate systems on a single entry.

Understanding the Importance of Using Fully Constrained, Separate Grid Points

Importantly, with Method 1s, if your model is later modified and any of the reference grid point locations change, your local coordinate system orientation will also change. To avoid this type of problem when using Method 1, you should always define three fully constrained grid points that are not part of the structure. Then, if your model is modified, your local coordinate system remains unchanged.

This example illustrates the problems you can encounter if you use Method 1 to define a coordinate system with the CORD1R entry using three existing grid points instead of defining three new ones.

Consider the truss structure shown in [Figure 6-4](#). Suppose you want the positive T1 component of the displacements for all the grid points to be in the negative X_b -direction as shown in (a).

In this case, we want to define CORD1R using existing grid points 2, 4, and 1 and assigning it an ID of 11. You enter coordinate system 11 in the CD field of the GRID entry for all the grid points. The model solves without problems and all results are in the correct direction.

Problems will occur if you need to subsequently modify the model. For example, suppose you need to move the location of grid point 4 as shown in (b). However, in the process of moving grid point 4, you also inadvertently rotate the local coordinate system 11.

While the deformed shape may look accurate if you post process the model, if you examine the results in the output file, you will notice that they are not in the correct direction.

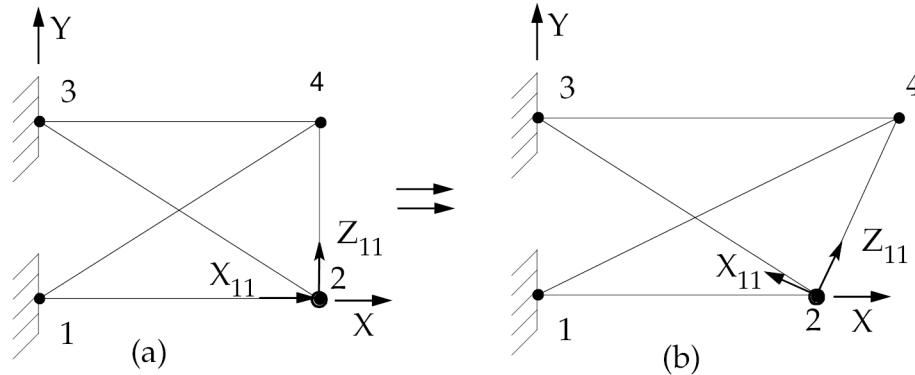


Figure 6-4. Model Change

Defining a Coordinate System By Referencing an Existing Coordinate System

The CORD2R, CORD2C, and CORD2S entries let you define a local coordinate system by specifying the location of three points in a previously defined reference coordinate system. This is also known as the “Method 2” approach of coordinate system definition in NX Nastran. This reference coordinate system can either be the basic coordinate system or another local coordinate system. Importantly, if this reference coordinate system is modified, the associated local coordinate system will also change.

The method for defining a coordinate system with CORD2R, CORD2C, or CORD2S is very similar. With each of these bulk data entries, you must:

- Specify a unique coordinate system ID (CID) in field 2 of the entry.
- Specify the coordinate system ID (RID) in field 3 of the reference coordinate system you're using. If you enter a 0 or a blank in this field, NX Nastran uses the basic coordinate system for the reference coordinate system.
- Enter the coordinates of three grid points in the reference coordinate system you defined in field 3.

Defining a Rectangular Coordinate System

You use the CORD1R and CORD2R Bulk Data entries to define a rectangular coordinate system.

Understanding CORD1R

When you define a rectangular coordinate with CORD1R, you must specify three reference grid points: G1, G2, and G3.

- G1 defines the origin of the coordinate system.
- A vector taken from G1 to G2 defines the Z-axis
- G3, together with the Z-axis, define the XZ plane.
- The X-axis is defined to be in this XZ plane and is perpendicular to the Z-axis.
- The Y-axis is generated from the X- and Z-axes using the right-hand rule.

CORD1R Format

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Field**Contents**

CIDA, CIDB

Coordinate system identification number.

GiA, GiB

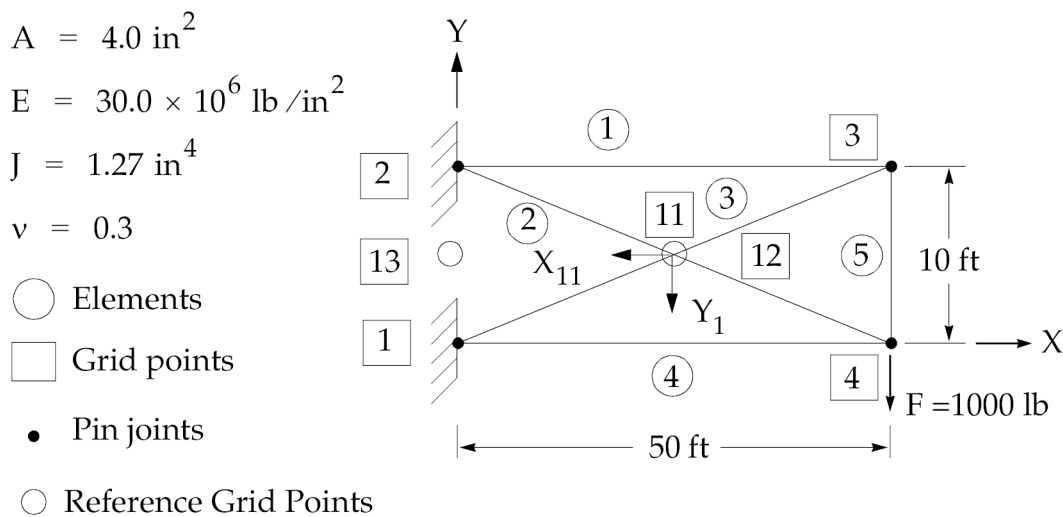
Grid point identification numbers.

See also

- “CORD1R” in the *NX Nastran Quick Reference Guide*

Example: Adding Three Additional Grid Points to Define CORD1R

To better understand the use of a CORD1R entry, consider the truss structure introduced earlier in this book. The input file has been modified so that the location of grid points 1 through 4 are defined in a rectangular coordinate system with an ID of 11 as shown in **Figure 6-5**.

**Figure 6-5. Grid Point Locations Using a Local Coordinate System**

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

The corresponding input file for the structure is given in **Listing 6-1**. Note that it is necessary to add three additional grid points to the input file. Importantly, these additional grid points don't affect the overall stiffness of the structure.

- Grid point 11 is located at the center of the truss structure, but it isn't attached to the structure. It is fully constrained.
- Grid point 12 is located 10 inches above grid point 11 in the basic Z-direction. The Z-axis for the new coordinate system is therefore parallel to the basic Z-axis. The choice of 10 inches is arbitrary since any positive value produces the same results.

- Grid point 13 defines the local XZ plane from which the X- and Y-axes of coordinate system 11 are determined. The choice of 0 inches for the X location of grid point 13 is also somewhat arbitrary since it is used only to define the XZ plane. Any real number less than 300 inches produces the same results. If the X-component of grid point 13 is greater than 300 inches, then the X-axis will be in the basic X-direction instead of the direction shown.

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

Listing 6-1. Truss Using CORD1R

Understanding CORD2R

With CORD2R, you define the locations of points A, B, and C in the same manner as with the CORD1R entry. You define these points in space using coordinates relative to a reference coordinate system that you specify in Field 3. You must define the reference coordinate system independently.

CORD2R Format

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

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

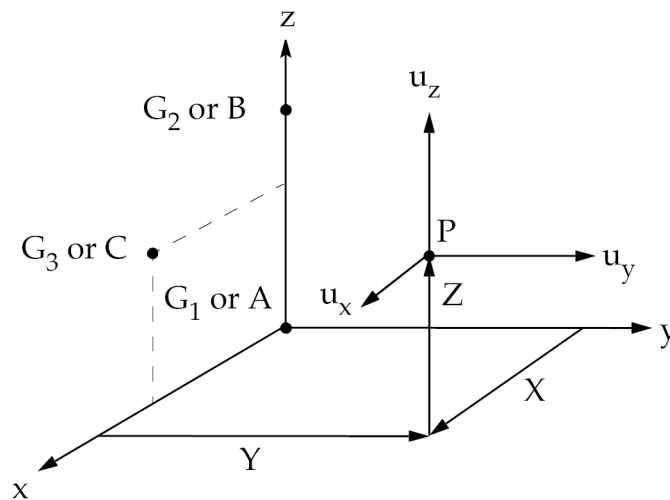


Figure 6-6. CORDIR and CORD2R Definitions

See also

- “**CORD2R**” in the *NX Nastran Quick Reference Guide*

Defining Cylindrical Coordinate Systems

The CORD1C and CORD2C bulk data entries allow you to define cylindrical coordinate systems.

Understanding CORD1C

The CORD1C entry is used to define a cylindrical coordinate system using three grid points to define its location and orientation in space as shown in [Figure 6-7](#).

- Grid point G1 defines the origin of the cylindrical system.
- A vector from grid point G1 to grid point G2 defines the Z-axis.
- Grid point G3 together with the Z-axis defines the $\theta = 0$ plane.

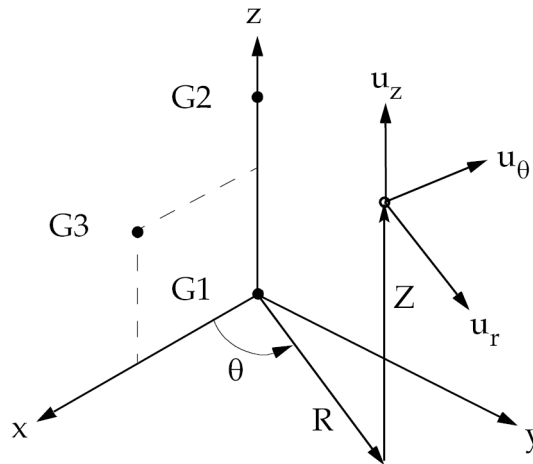


Figure 6-7. CORD1C Definition

CORD1C Format

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

Field

Contents

CIDA, CIDB

Coordinate system identification number.

GiA, GiB

Grid point identification numbers.

See also

- “CORD1C” in the *NX Nastran Quick Reference Guide*

Understanding CORD2C

The CORD2C entry uses the location of three points A, B, and C in the same manner as the three grid points are used for the CORD1C entry. These points are defined in space using coordinates relative to the reference coordinate system that you specify in Field 3.

- Point A defines the origin.
- Point B defines the direction of the z-axis.
- Point C lies in the plane of the azimuthal origin.

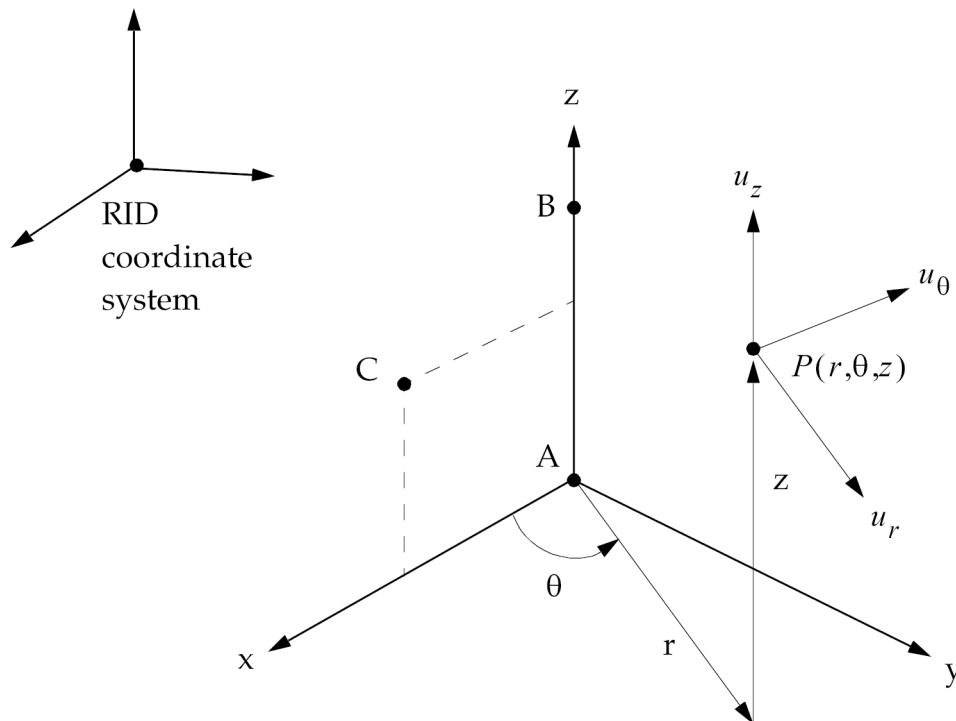
You must independently define the reference coordinate system (RID).

The location of a grid point (P in Figure 6-8) in this coordinate system is given by (r, θ, z) where θ is measured in degrees. If this coordinate system is used to specify the displacement directions they are given by u_r , u_θ and u_z (the displacements are u_r , u_θ and u_z).

CORD2C Format

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

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0 ; Default (zero) is the basic coordinate system)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

**Figure 6-8. CORD2C Definition****See also**

- “CORD2C” in the *NX Nastran Quick Reference Guide*

Example: Using the CORD2C Entry

The following example illustrates the use of the CORD2C entry. An arch with a semicircular top is shown in [Figure 6-9](#). To facilitate grid point input, we want to establish a local cylindrical coordinate system for grid points 3 through 7.

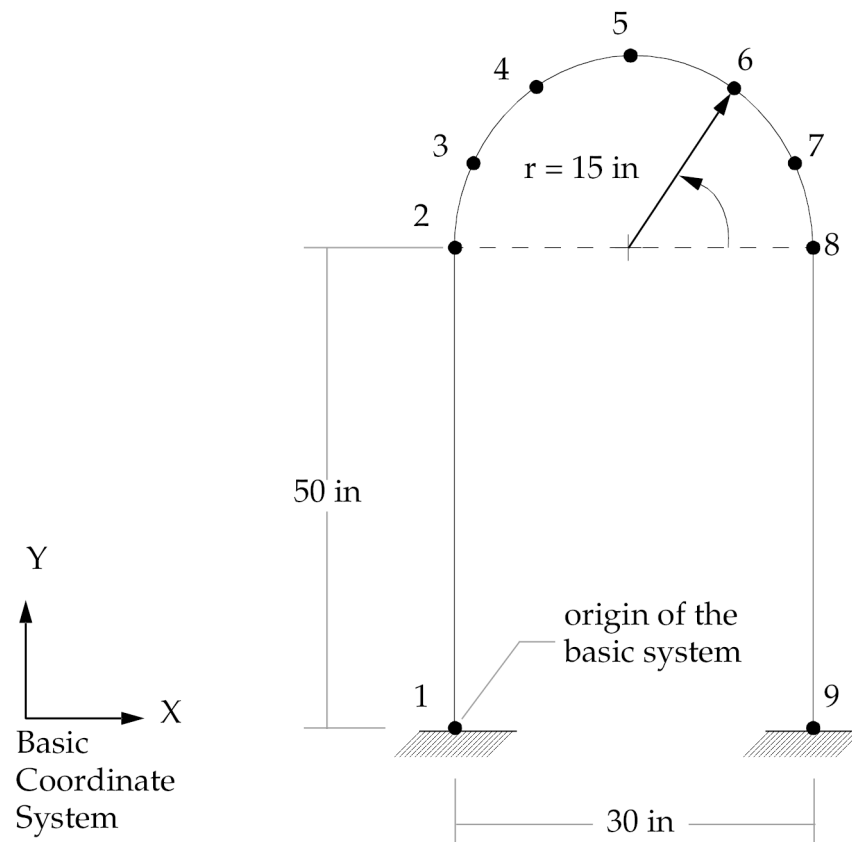


Figure 6-9. Arch Structure

A cylindrical coordinate system with an ID of 100 is defined as shown in [Figure 6-10](#).

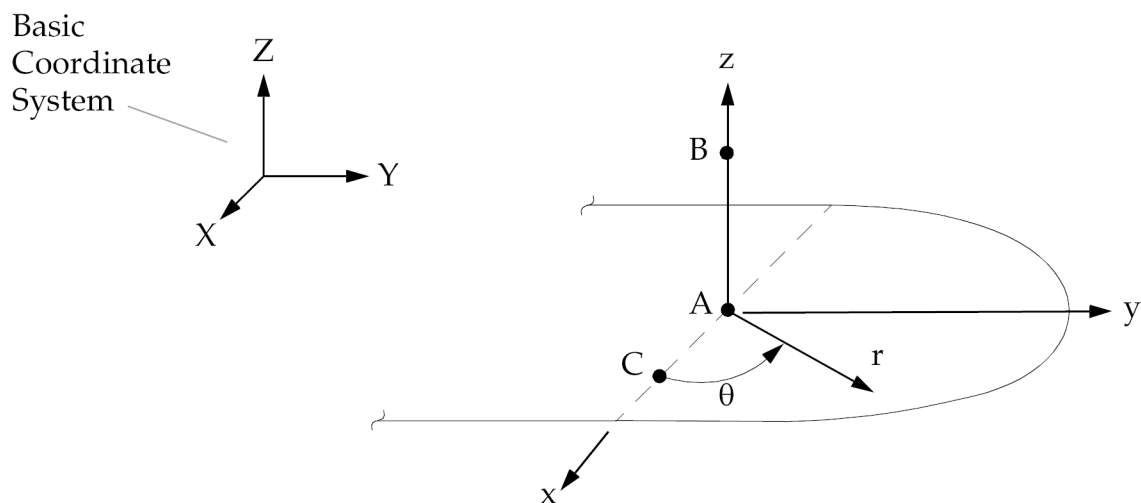


Figure 6-10. Definition of Local Cylindrical Coordinate System

The local cylindrical coordinate system is defined by referencing the basic coordinate system. Three points are required to define the local system:

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

This coordinate system has an identification number in field CID which will be referenced by other entities, such as the CP and CD fields of a GRID entry. In this example, the coordinates of points A, B, and C are in the basic coordinate system.

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						
CORD2C	100		15.	50.	0.	15.	50.	1.	
	16.	50.	0.						

The associated GRID entries are defined as follows:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.				
GRID	2		0.	50.	0.				
GRID	3	100	15.	150.	0.				
GRID	4	100	15.	120.	0.				
GRID	5	100	15.	90.	0.				
GRID	6	100	15.	60.	0.				
GRID	7	100	15.	30.	0.				
GRID	8		30.	50.	0.				
GRID	9		30.	0.	0.				

Grid points 3 through 7 use (r, θ , z) coordinates with r = 15.0 inches and θ varying from 30 degrees (GRID 7) to 150 degrees (GRID 3). The output for all grid points is in the basic rectangular coordinate system since field 7 is left blank.

Understanding CORD1S

The CORD1S entry is similar to the CORD1C entry since three grid points (G1, G2, and G3) are used to define the new coordinate system. As shown in [Figure 6-11](#):

- Grid point G1 defines the origin of the spherical system.
- The vector starting at grid point G1 and passing through grid point G2 is the $\theta = 0$ axis.
- This axis, together with grid point G3, determines the $\phi = 0$ plane.
- The $\phi = 0$ axis is in this plane and is oriented using the right-hand rule.

CORD1S Format

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

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

See also

- “CORD1S” in the *NX Nastran Quick Reference Guide*

Understanding CORD2S

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

CORD2S Format

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

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

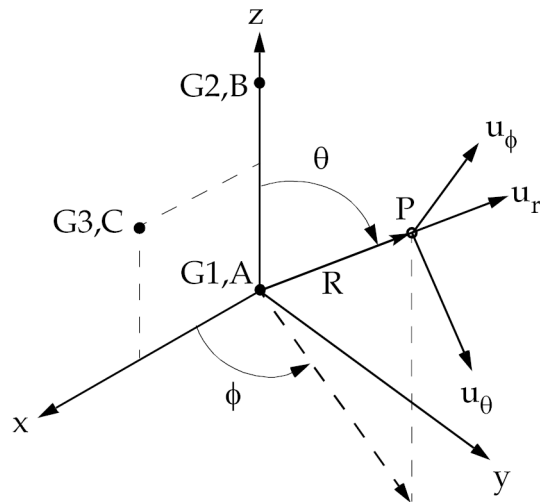


Figure 6-11. CORD1S and CORD2S Definitions

Note

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

Example: Using Cylindrical and Spherical Coordinate Systems

To see the use of cylindrical and spherical coordinate systems, consider the grain silo shown below.

- The cylindrical portion of the silo is modeled using a cylindrical coordinate system.
- The roof of the silo is modeled using a spherical coordinate system.

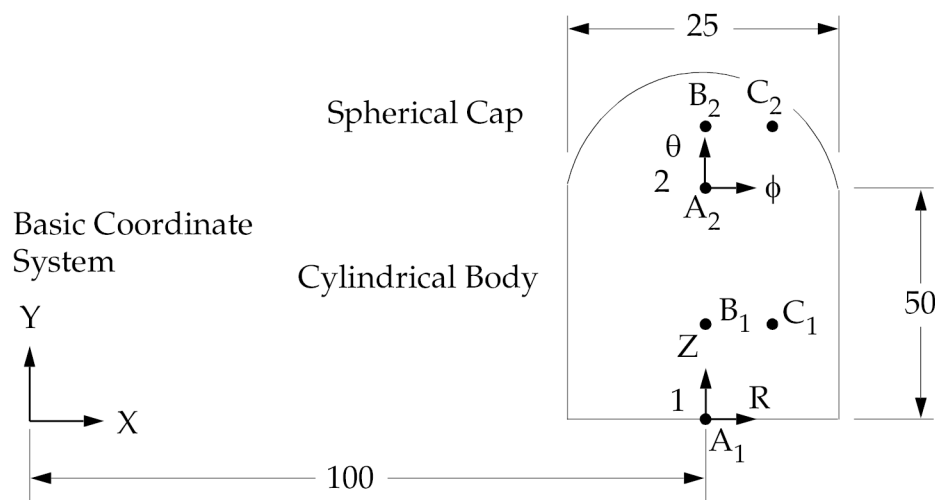


Figure 6-12. Silo Model

The cylindrical coordinate system has an ID of 1 and is defined with the CORD2C entry. It is defined with respect to the basic coordinate system.

- The origin is given by point A_1 , and located by the vector (100,0,0) as shown in [Figure 6-12](#).
- The Z-axis is defined by the line extending from point A_1 to point B_1 , which is located at (100,1,0).
- The Z-axis along with point C_1 , which is located at (101,1,0), defines the $\theta = 0$ plane.
- The R-axis is defined in this plane and is perpendicular to the Z-axis.
- The direction of the θ -axis is determined using the right-hand rule.

The spherical coordinate system has an ID of 2 is defined with the CORD2S entry. It is defined with respect to the reference coordinate system 1.

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

Since coordinate system 1 is defined in terms of the basic coordinate system, coordinate system 2 is also indirectly defined in terms of the basic coordinate system. You could also directly define both coordinate systems in terms of the basic coordinate system. However, since coordinate system 2 is linked to coordinate system 1, adjusting the position of the silo (for example, due to a change in the module's geometry) is only a matter of changing the locations of the grid points on the CORD2C entry.

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

Listing 6-2. Grain Silo Coordinate Systems

6.4 Understanding the Element and Material Coordinate System

Every element in the NX Nastran library has its own unique element coordinate system that is used to output the element forces, moments, and stresses as well as to orient section properties. For some elements, such as the CROD element, the element coordinate systems are implicitly defined based on the element connectivity. For other elements, you must explicitly define the orientation of the element coordinate system. For further discussion of element coordinate systems, refer to the *NX Nastran Element Library*.

In addition to the element coordinate system, an element may also have an optional material coordinate system that can be used to define the orientation of orthotropic or anisotropic material.

Special Considerations for Grid Points on the Polar Axis

If the displacements of a grid point are defined to be in a spherical or cylindrical coordinate system and the grid point is located on, or relatively close to, the polar axis, then it will be replaced by the “defining” rectangular coordinate system. The z-axis of the “defining” rectangular coordinate system is coincident with the polar axis and the x-axis is coincident with $\theta = 0$ in cylindrical system, or $\theta = 0$ and $\phi = 0$ in the spherical system. In this case, you should explicitly specify a rectangular coordinate system.

Defining Element Offsets

Element offsets are defined by a Cartesian coordinate system, which is also parallel to the displacement coordinate system at the connecting grid point. However, the components of the offsets are always defined in units of translation, even if the displacement coordinate system is cylindrical or spherical.

6.5 Other Special Coordinate System Entries

In addition to the main coordinate system bulk data entries discussed above, there are some additional entries that you can use when defining coordinate systems in NX Nastran.

CORD3G

CORD3G lets you define a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. You use the CORD3G entry with the MAT9 entry to orient the material principal axes for 3D composite analyses.

See also

- “CORD3G” in the *NX Nastran Quick Reference Guide*.

GMCORD

GMCORD lets you define a convective/follower coordinate system on a FEEDGE, GMCURV, FEFACE, or GMSURF bulk data entry for use in adaptive (p-element) analysis.

See also

- “GMCORD” in the *NX Nastran Quick Reference Guide*

Chapter 7: Material Properties

- *Introduction*
- *Material Properties for Linear Structural Analysis*
- *Material Properties for Nonlinear Structural Analysis*
- *Material Properties for Thermal Analysis*
- *Material Properties for Fluids*

7.1 Introduction

NX Nastran allows you to define material properties for solids and fluids. You use material bulk entries to specify properties like Young's modulus, Poisson's ratio, density, thermal conductivity, and so on. In some cases, material bulk entries are also used to specify parameters that are not normally considered to be material properties. For example, in heat transfer analysis, the heat transfer coefficient can be specified using a MAT4 bulk entry.

Generally, material properties are assigned to elements indirectly with each element bulk entry referencing a property bulk entry and each property bulk entry referencing a material bulk entry. Although infrequent, the material properties for some elements are also defined:

- Directly on the element bulk entry. CELAS2 is an example of this.
- Indirectly on a property bulk entry. CELAS1 is an example of this.
- Indirectly on a material bulk entry referenced by the element bulk entry. CONROD is an example of this.

Table 7-1 summarizes the material bulk entries that are available to you. In Table 7-1, the material bulk entries are categorized as primary or supplemental. The material bulk entries categorized as primary are stand-alone bulk entries, whereas those categorized as supplemental are only used in combination with a primary material bulk entry. For example, the strain hardening characteristic of a material can be defined using a MATS1 bulk entry. However, the MATS1 bulk entry must be referenced by either a MAT1, MAT2, or MAT9 bulk entry.

Table 7-1. Summary of Material Bulk Entries			
Material Bulk Entry	Primary or Supplemental (1)	Property Type	Description
MAT1	Primary	Mechanical (2)	Defines properties for isotropic, linearly elastic materials.
MAT2	Primary	Mechanical (2)	Defines properties for anisotropic, linearly elastic materials. Valid for shell elements only.
MAT3	Primary	Mechanical (2)	Defines properties for orthotropic, linearly elastic materials. Valid for axisymmetric elements. Also valid for plane stress and plane strain elements when using an alternate form.
MAT4	Primary	Thermal	Defines properties for isotropic materials.
MAT5	Primary	Thermal	Defines properties for anisotropic materials.
MAT8	Primary	Mechanical (2)	Defines properties for orthotropic, linearly elastic materials. Valid for shell elements only.
MAT9	Primary	Mechanical (2)	Defines properties for anisotropic, linearly elastic materials. Valid for solid elements only.
MAT10	Primary	Fluid	Defines properties for compressible fluids.

Table 7-1. Summary of Material Bulk Entries

Material Bulk Entry	Primary or Supplemental (1)	Property Type	Description
MAT11	Primary	Mechanical (2)	Defines properties for orthotropic, linearly elastic materials. Valid for solid elements only.
MATFT	Supplemental	Mechanical	Defines failure criterion when used in combination with MAT1, MAT9, or MAT11 bulk entries.
MATG	Supplemental	Mechanical	Defines properties for gasket materials when used in combination with a MAT1 bulk entry. Valid for SOL 601 only.
MATHE	Primary	Mechanical (2)	Defines properties for hyperelastic materials. Valid for SOLs 601 and 701 only.
MATHP	Primary	Mechanical (2)	Defines properties for hyperelastic materials.
MATS1	Supplemental	Mechanical	Defines stress dependency for properties on MAT1, MAT2, or MAT9 bulk entries.
MATSMA	Primary	Mechanical (2)	Defines properties for shape-memory alloys.
MATT1	Supplemental	Mechanical	Defines temperature dependency for properties on MAT1 bulk entries.
MATT2	Supplemental	Mechanical	Defines temperature dependency for properties on MAT2 bulk entries.
MATT3	Supplemental	Mechanical	Defines temperature dependency for properties on MAT3 bulk entries.
MATT4	Supplemental	Thermal	Defines temperature dependency for properties on MAT4 bulk entries.
MATT5	Supplemental	Thermal	Defines temperature dependency for properties on MAT5 bulk entries.
MATT8	Supplemental	Mechanical	Defines temperature dependency for properties on MAT8 bulk entries.
MATT9	Supplemental	Mechanical	Defines temperature dependency for properties on MAT9 bulk entries.
MATT11	Supplemental	Mechanical	Defines temperature dependency for properties on MAT11 bulk entries.
MATTC	Supplemental	Mechanical	Defines temperature dependency for properties on CREEP bulk entries.
MFLUID	Primary	Fluid	Defines properties of an incompressible fluid for the purpose of representing a fluid mass.
CREEP	Supplemental	Mechanical	Defines creep properties when used in combination with MAT1, MAT2, or MAT9 bulk entries.

(1) Primary material bulk entries are stand-alone, whereas supplemental material bulk entries are only used in combination with a primary material bulk entry.

(2) Includes properties for thermal expansion.

Table 7-1 is useful for identifying the material bulk entries that you can use to model a particular material behavior. However, not every material bulk entry can be used with every element for a given solution sequence. Table 7-2 is provided to assist you in identifying which elements can be used when performing a particular type of analysis. In order to determine applicability:

1. Use Table 7-2 to determine which elements are available for a particular type of analysis.
2. Use Tables 7-4 through 7-8 to identify what combinations of material bulk entries and property bulk entries can be used with the elements found in the previous step to obtain the desired material behavior.

For example, Table 7-2 indicates that the CQUAD4 element can be used to model plasticity using SOL 106. Table 7-5 indicates that in order for the CQUAD4 element to model plasticity, the CQUAD4 element must reference a PSHELL entry that in turn references either a MAT1, MAT2, or MAT8 entry that is used in combination with a MATS1 entry. If the MATS1 entry is not used, CQUAD4 elements can still be used in a SOL 106 plasticity analysis, but they will behave linear elastically.

Table 7-2. Element Applicability for Solutions 101–200

Type	Element	Linear Struc- tural SOLs (1)	Nonlinear Structural, SOLs 106 and 129					Heat Transfer SOLs 153 and 159
			Material Nonlinear (2)				Geometric Nonlinear (3)	
			Plasticity	Nonlinear Elastic	Hyper- elasticity	Creep		
1D	CBAR(a)	X						X
	CBEAM(b)	X	X(4)				X	X
	CBEND(a)	X						X
	CONROD(b)	X	X(5)	X(5)			X	X
	CROD(b)	X	X(5)	X(5)		X(5)	X	X
	CTUBE(b)	X	X(5)	X(5)			X	X
2D	CPLSTN3(a)	X						X
	CPLSTN4(a)	X						X
	CPLSTN6(a)	X						X
	CPLSTN8(a)	X						X
	CPLSTS3(a)	X						X
	CPLSTS4(a)	X						X
	CPLSTS6(a)	X						X
	CPLSTS8(a)	X						X
	CQUAD(c)				X			
	CQUAD4(d)	X	X	X	X	X	X	X
	CQUAD8(e)	X			X			X
	CQUADR(a)	X						X
	CRAC2D(a)	X						
	CSHEAR(a)	X						
	CTRIA3(d)	X	X	X	X	X	X	X
	CTRIA6(e)	X			X			X
	CTRIAR(a)	X						X

Table 7-2. Element Applicability for Solutions 101–200

Type	Element	Linear Struc- tural SOLs (1)	Nonlinear Structural, SOLs 106 and 129					Heat Transfer SOLs 153 and 159
			Material Nonlinear (2)				Geometric Nonlinear (3)	
			Plasticity	Nonlinear Elastic	Hyper- elasticity	Creep		
Axi- sym	CCONEAX ^(a)	X	(2)	(2)		(2)	(2)	
	CQUADX ^(c)		(2)	(2)	X	(2)	(2)	
	CQUADX4 ^(e)	X	(2)	(2)	X	(2)	(2)	X
	CQUADX8 ^(e)	X	(2)	(2)	X	(2)	(2)	X
	CTRAX3 ^(e)	X	(2)	(2)	X	(2)	(2)	X
	CTRAX6 ^(e)	X	(2)	(2)	X	(2)	(2)	X
	CTRIAX ^(c)		(2)	(2)	X	(2)	(2)	
	CTRIAX6 ^(a)	X	(2)	(2)		(2)	(2)	X
3D	CHEXA ^(d)	X	X	X	X	X	X	X
	CPENTA ^(d)	X	X	X	X	X	X	X
	CPYRAM ^(b)	X	X	X		X	X	X
	CRAC3D ^(a)	X						
	CTETRA ^(d)	X	X	X	X	X	X	X

Element Footnotes:

(a) Behaves linear elastically if used in a material nonlinear analysis. Stiffness does not reformulate if used in a geometric nonlinear analysis.

(b) Limited material nonlinear capability that is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness reformulates if used in a geometric nonlinear analysis.

(c) Behavior is plane strain and nearly incompressible, and stress and strain results are not output, if used for any analysis other than hyperelastic analysis.

(d) Comprehensive material nonlinear capability that is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness reformulates if used in a geometric nonlinear analysis.

(e) Hyperelastic capability is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness does not reformulate if used in a geometric nonlinear analysis.

Other footnotes:

(1) Includes SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 187, and 200.

(2) Hyperelasticity is the only nonlinear capability that allows axisymmetric analysis.

(3) Small strain only. For large strain, refer to the hyperelasticity column.

(4) Elastic-perfectly plastic material behavior used to model the ends of beams as plastic hinges.

(5) Nonlinear capability for axial deflections only.

Element, Property, and Material Bulk Entries

Tables 7-4 through 7-8 list the properties and materials you can use with a given element for solution sequences 101 through 200. To assist you in navigating to the correct table, Table 7-3 is provided.

Table 7-3. Summary of Material, Property, and Element Cross-Reference Tables

Element Type	Elements	Table
1D	CBAR, CBEAM, CBEND, CONROD, CROD, CTUBE	7-4
2D	CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, CQUAD, CQUAD4, CQUAD8, CQUADR, CRAC2D, CSHEAR, CTRIA3, CTRIA6, CTRIAR	7-5
Axisymmetric	CCONEAX, CQUADX, CQUADX4, CQUADX8, CTRAX3, CTRAX6, CTRIAX, CTRIAX6	7-6
3D	CHEXA, CPENTA, CPYRAM, CRAC3D, CTETRA	7-7
Special Purpose	CAABSF, CAERO1, CAERO2, CAERO3, CAERO4, CAERO5, CAXIFI, CBUSH, CBUSH1D, CDAMP1, CDAMP2, CDAMP3, CDAMP4, CDAMP5, CDUMi, CELAS1, CELAS2, CELAS3, CELAS4, CFAST, CFLUIDi, CGAP, CHACAB, CHACBR, CHBDYE, CHBDYG, CHBDYP, CMASS1, CMASS2, CMASS3, CMASS4, CONM1, CONM2, CONV, CONVM, CSLOT3, CSLOT4, CVISC, CWELD	7-8

Table 7-4. 1D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CBAR	PBAR	MAT1			X									
		MAT4						X						
		MAT5							X					
	PBARL	MAT1			X									
		MAT4						X						
		MAT5							X					
CBEAM	PBEAM	MAT1		X ⁽¹⁾	X									
		MAT4						X						
		MAT5							X					
	PBEAML	MAT1			X									
		MAT4						X						
		MAT5							X					
	PBCOMP	MAT1		X ⁽¹⁾	X									
		MAT4						X						
		MAT5							X					
CBEND	PBEND	MAT1			X									
		MAT4						X						
		MAT5							X					
CONROD	N/A	MAT1		X ⁽²⁾	X									
		MAT4						X						
		MAT5							X					

Table 7-4. 1D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CROD	PROD	MAT1		X ⁽²⁾	X									X ⁽²⁾
		MAT4						X						
		MAT5							X					
CTUBE	PTUBE	MAT1		X ⁽²⁾	X									
		MAT4						X						
		MAT5							X					

(1) For material nonlinear analysis, elastic-perfectly plastic material behavior only. Strain hardening is not supported. The ends of the element are modeled as plastic hinges.

(2) Nonlinear capability for axial deflections only.

Table 7-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CPLSTN3	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTN4	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTN6	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTN8	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTS3	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					

Table 7-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CPLSTS4	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTS6	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CPLSTS8	PPLANE	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
CQUAD	PLPLANE	MATHP												
CQUAD4	PSHELL	MAT1		X	X								X	
		MAT2		X		X							X	
		MAT4						X						
		MAT5							X					
		MAT8		X						X			X	
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
		MAT8								X				
	PLPLANE	MATHP												
	N/A	MFLUID												
CQUAD8	PSHELL	MAT1			X									
		MAT2				X								
		MAT4						X						
		MAT5							X					
		MAT8								X				
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
MAT8									X					

Table 7-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
	PLPLANE	MATHP												
CQUADR	PSHELL	MAT1			X									
		MAT2				X								
		MAT4						X						
		MAT5							X					
		MAT8								X				
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
		MAT8								X				
CRAC2D	PRAC2D	MAT1			X									
		MAT2				X								
		MAT8							X					
CSHEAR	PSHEAR	MAT1												
CTRIA3	PSHELL	MAT1		X	X								X	
		MAT2		X		X							X	
		MAT4						X						
		MAT5							X					
		MAT8		X						X			X	
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
		MAT8								X				
	PLPLANE	MATHP												
N/A	MFLUID													

Table 7-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CTRIA6	PSHELL	MAT1			X									
		MAT2				X								
		MAT4						X						
		MAT5							X					
		MAT8								X				
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
		MAT8								X				
	PLPLANE	MATHP												
CTRIAR	PSHELL	MAT1			X									
		MAT2				X								
		MAT4						X						
		MAT5							X					
		MAT8								X				
	PCOMP	MAT1			X									
		MAT2				X								
		MAT8								X				
	PCOMPG	MAT1			X									
		MAT2				X								
		MAT8								X				

Table 7-6. Axisymmetric Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CCONEAX	PCONEAX	MAT1			X									
		MAT2				X								
CQUADX	PLPLANE	MATHP												

Table 7-6. Axisymmetric Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CQUADX4	PSOLID	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
		MAT9									X			
	PLSOLID	MATHP												
CQUADX8	PSOLID	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
		MAT9									X			
	PLSOLID	MATHP												
CTRAX3	PSOLID	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
		MAT9									X			
	PLSOLID	MATHP												
CTRAX6	PSOLID	MAT1			X									
		MAT3					X							
		MAT4						X						
		MAT5							X					
		MAT9									X			
	PLSOLID	MATHP												
CTRIAX	PLPLANE	MATHP												
CTRIAX6	N/A	MAT1			X									
		MAT3					X							
		MAT4						X						

Table 7-7. 3D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CHEXA	PSOLID	MAT1		X	X									X
		MAT4						X						
		MAT5							X					

Table 7-7. 3D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
		MAT9		X							X		X	
		MAT10												
		MAT11		X								X	X	
	PLSOLID	MATHP												
	PCOMPS	MAT1	X		X									
		MAT9	X								X			
		MAT11	X									X		
CPENTA	PSOLID	MAT1		X	X								X	
		MAT4						X						
		MAT5							X					
		MAT9		X							X		X	
		MAT10												
		MAT11		X								X	X	
	PLSOLID	MATHP												
	PCOMPS	MAT1	X		X									
		MAT9	X								X			
MAT11		X									X			
CPYRAM	PSOLID	MAT1		X	X								X	
		MAT4						X						
		MAT5							X					
		MAT9		X							X		X	
		MAT10												
		MAT11		X								X	X	
CRAC3D	PRAC3D	MAT1			X									
		MAT9									X			
CTETRA	PSOLID	MAT1		X	X								X	
		MAT4						X						
		MAT5							X					
		MAT9		X							X		X	
		MAT10												
		MAT11		X								X	X	
	PLSOLID	MATHP												

Table 7-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CAABSF	PAABSF	N/A												
CAERO1	PAERO1	N/A												
CAERO2	PAERO2	N/A												
CAERO3	PAERO3	N/A												
CAERO4	PAERO4	N/A												
CAERO5	PAERO5	N/A												
CAXIFi (1)	N/A	N/A												
CBUSH	PBUSH (2)	N/A												
CBUSH1D	PBUSH1D	N/A												
CDAMP1	PDAMP	N/A												
CDAMP2	N/A	N/A												
CDAMP3	PDAMP	N/A												
CDAMP4	N/A	N/A												
CDAMP5	PDAMP5	MAT4						X						
		MAT5						X						
CDUMi (3)	PDUMi (3)	(4)												
CELAS1	PELAS	N/A												
CELAS2	N/A	N/A												
CELAS3	PELAS	N/A												
CELAS4	N/A	N/A												
CFAST	PFAST	N/A												
CFLUIDi (1)	N/A	N/A												
CGAP	PGAP	N/A												
CHACAB	PACABS	N/A												
CHACBR	PACBAR	N/A												
CHBDYE	N/A	N/A												
CHBDYG	N/A	N/A												
CHBDYP	PHBDY	N/A												
CMASS1	PMASS	N/A												
CMASS2	N/A	N/A												
CMASS3	PMASS	N/A												
CMASS4	N/A	N/A												
CONM1	N/A	N/A												
CONM2	N/A	N/A												
CONV	PCONV	MAT4						X						
CONVM	PCONVM	MAT4						X						
CSLOT3	N/A	N/A												

Table 7-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material											CREEP/ MATTC
			MATi											
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11		
CSLOT4	N/A	N/A												
CVISC	PVISC	N/A												
CWELD	PWELD	MAT1			X									
RBAR	N/A	N/A												
RBEi (5)	N/A	N/A												
(1) i = 2, 3, or 4														
(2) Use a PBUSHT entry with the same PID as a PBUSH entry to define frequency dependent or stress dependent properties.														
(3) i = 1 through 9														
(4) Per user definition.														
(5) i = 1, 2, or 3														

7.2 Material Properties for Linear Structural Analysis

For linear structural analysis, you can use the MAT1, MAT2, MAT3, MAT8, MAT9, or MAT11 bulk entries to define the mechanical properties, coefficient of thermal expansion, and reference temperature for a linear elastic material. The choice of which material bulk entry to use depends on the type of element used and the degree of anisotropy required. Table 7-9 is provided to simplify the selection process.

Table 7-9. Summary of Material Bulk Entries for Linear Structural Analysis

Material Bulk Entry	Element Type(s)	Constitutive Model	Number of Independent Elastic Constants
MAT1	1D, 2D, 3D, Axisymmetric	Isotropic	2
MAT2	2D	Anisotropic (a)	6 (b)
MAT3	Axisymmetric	Orthotropic	7 (c)
MAT8	2D	Orthotropic	4 (d)
MAT9	3D	Anisotropic (a)	21
MAT11	3D	Orthotropic	9

(a) Also referred to as triclinic or aelotropic.

(b) The number of independent elastic constants required to define an anisotropic material is reduced from 21 for a 3D stress state to 6 for a 2D stress state.

(c) The number of independent elastic constants required to define an orthotropic material is reduced from 9 for a 3D stress state to 7 for an axisymmetric representation of a 3D stress state.

(d) The number of independent elastic constants required to define an orthotropic material is reduced from 9 for a 3D stress state to 4 for a 2D stress state.

The material properties defined on any of the material bulk entries listed in Table 7-9 can be made temperature dependent by defining a corresponding MATTi bulk entry. For example, to define an

isotropic material having a temperature dependent Young's modulus, you can use a MATT1 bulk entry in combination with a MAT1 bulk entry to define the temperature dependency.

Isotropic Materials

For an isotropic material, the properties are the same in all directions. Two independent elastic constants are required to characterize the mechanical behavior of an isotropic material. The constitutive relation for a linear elastic isotropic material subjected to a three-dimensional stress state is:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} & \frac{\nu E}{(1+\nu)(1-2\nu)} & \frac{\nu E}{(1+\nu)(1-2\nu)} & 0 & 0 & 0 \\ & \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} & \frac{\nu E}{(1+\nu)(1-2\nu)} & 0 & 0 & 0 \\ & & \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} & 0 & 0 & 0 \\ & & & \frac{E}{2(1+\nu)} & 0 & 0 \\ & & & & \frac{E}{2(1+\nu)} & 0 \\ & & & & & \frac{E}{2(1+\nu)} \end{bmatrix}$$

symmetric

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = (T - T_{ref}) \begin{Bmatrix} A \\ A \\ A \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

where:

$\sigma_x \sigma_y \sigma_z$	Normal stresses
$\tau_{xy} \tau_{yz} \tau_{zx}$	Shear stresses
$\varepsilon_x \varepsilon_y \varepsilon_z$	Normal strains
$\gamma_{xy} \gamma_{yz} \gamma_{zx}$	Engineering shear strains
E	Young's modulus
ν	Poisson's ratio
A	Coefficient of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

Linear elastic isotropic materials can be defined directly using MAT1 bulk entries. On the MAT1 bulk entry, there are three fields for entering elastic constants including fields for Young's modulus, Poisson's ratio, and shear modulus even though only two independent elastic constants are required

to define a linear elastic isotropic material. When you specify only two of these elastic constants, NX Nastran calculates the third from:

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

If you enter all three elastic constants and they do not satisfy this relationship, NX Nastran prints a warning message indicating that the isotropic relationship has been violated. However, depending on the element type, NX Nastran may use all three values to formulate the element stiffness matrix even though the values of the elastic constants are incompatible. Because of this, we strongly recommend that you input only two of the three elastic constants on a MAT1 bulk entry.

Which elastic constants are used in the element stiffness matrix formulation depends on the element type. Table 7-10 shows which elastic constants are used to formulate the stiffness matrices for various elements.

Table 7-10. Elastic Constant Usage in Stiffness Formulation for Various Elements				
Type	Element	E	ν	G
1D	CROD	Axial and Bending	Not Used	Torsion and Transverse Shear
	CBEAM			
	CBAR			
2D	CCONEAX	Membrane and Bending		Transverse Shear
	CQUADi			
	CTRIAi			
	CRAC2D	All Terms		Not Used
	CSHEAR	Not Used		Shear
Axisymmetric	CQUADXi	Radial, Axial, Circumferential	All Coupled Ratios	Shear
	CTRAXi			
	CTRIAX6			
3D	CHEXA	All Terms		Not Used
	CPENTA			
	CPYRAM			
	CRAC3D			
	CTETRA			

Constitutive Relation for Plane Stress

The stress state for 2D elements is modeled as either plane stress (the default) or plane strain. For a plane stress state, $\sigma_z = \tau_{yz} = \tau_{zx} = 0$ and the constitutive relation for an isotropic material reduces to:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{\nu E}{1-\nu^2} & 0 \\ \frac{\nu E}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{bmatrix} \left\{ \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A \\ A \\ 0 \end{Bmatrix} \right\}$$

Equation 7-1.

Constitutive Relation for Plane Strain

You can obtain plane strain behavior for a 2D element by specifying certain settings on the property bulk entry. For a state of plane strain, $\varepsilon_z = \gamma_{yz} = \gamma_{zx} = 0$ and the constitutive relation for an isotropic material reduces to:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} & \frac{\nu E}{(1+\nu)(1-2\nu)} & 0 \\ \frac{\nu E}{(1+\nu)(1-2\nu)} & \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{bmatrix} \left\{ \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T - T_{ref})(1+\nu) \begin{Bmatrix} A \\ A \\ 0 \end{Bmatrix} \right\}$$

Equation 7-2.

Constitutive Relation for Transverse Shear

For calculating transverse shear stresses in 2D elements, the transverse shear strains are related to the transverse shear stresses through the shear modulus, G , as follows:

$$\begin{pmatrix} \sigma_{zx} \\ \sigma_{zy} \end{pmatrix} = \begin{bmatrix} G & 0 \\ 0 & G \end{bmatrix} \begin{pmatrix} \gamma_{zx} \\ \gamma_{zy} \end{pmatrix}$$

Equation 7-3.

MAT 1 Format

The MAT1 entry has the following format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Field

Contents

MID

Material identification number

Field	Contents
E	Young's modulus
G	Shear modulus
NU	Poisson's ratio
RHO	Mass density
A	Coefficient of thermal expansion
TREF	Reference temperature for the calculation of thermal loads
GE	Structural element damping coefficient
ST, SC, SS	Stress limits for tension, compression, and shear used only to compute margins of safety in certain elements; they have no effect on the computational procedures
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See the “Parameters” in the <i>NX Nastran Quick Reference Guide</i>

See also

- “MAT1” in the *NX Nastran Quick Reference Guide*

Example: Using MAT1 to Define a Structural Steel

As an example, create a MAT1 bulk entry for a structural steel that is used in a SOL 101 analysis that accounts for thermal stresses. Assume the reference temperature is 25°C and the material properties are as follows:

- Young's modulus of 204 GPa.
- Poisson's ratio of 0.29.
- Mass density of 7860 kg/m³.
- Coefficient of thermal expansion of $12 \times 10^{-6} / ^\circ\text{C}$.

Because NX Nastran is unitless, the values you use for the MAT1 inputs must have consistent units. If Newton, millimeter, and second are the base units, you should use 204×10^3 MPa for Young's modulus and 7860×10^{-9} kg/mm³ for mass density.

Given these material constants, the MAT1 entry in small field format is:

1	2	3	4	5	6	7	8	9	10
MAT1	5	204.E3		0.29	7860.E-9	12.E-6	25.		

The same MAT1 entry in free field format is:

```
MAT1,5,204.E3,,0.29,7860.E-9,12.E-6,25.
```


Orthotropic Materials

Orthotropic materials have three planes of material property symmetry. Nine independent elastic constants are required to fully describe three-dimensional orthotropic material behavior. For two-dimensional stress states and axisymmetric representations of three-dimensional stress states, the number of independent elastic constants required to define orthotropic material behavior is reduced. Four independent elastic constants are required for a two-dimensional stress state and seven independent elastic constants are required for an axisymmetric stress state.

You can use MAT3, MAT8 and MAT11 bulk entries to define orthotropic material properties.

- MAT3 allows you to define orthotropic material properties for axisymmetric elements.
- MAT8 allows you to define orthotropic material properties for 2D elements.
- MAT11 allows you to define orthotropic material properties for 3D elements.

Understanding the MAT11 Entry

The MAT11 bulk entry is used to define orthotropic material behavior for 3D (solid) elements. The constitutive relation expressed in terms of the principal material directions is given by:

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & -\frac{\nu_{21}}{E_2} & -\frac{\nu_{31}}{E_3} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & -\frac{\nu_{32}}{E_3} & 0 & 0 & 0 \\ -\frac{\nu_{13}}{E_1} & -\frac{\nu_{23}}{E_2} & \frac{1}{E_3} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{Bmatrix} + (T - T_{ref}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

Equation 7-4.

subject to:

$$\frac{\nu_{12}}{E_1} = \frac{\nu_{21}}{E_2} \quad \frac{\nu_{13}}{E_1} = \frac{\nu_{31}}{E_3} \quad \frac{\nu_{23}}{E_2} = \frac{\nu_{32}}{E_3}$$

Equation 7-5.

where:

$\sigma_1 \ \sigma_2 \ \sigma_3$	Normal stresses
$\tau_{12} \ \tau_{23} \ \tau_{31}$	Shear stresses
$\varepsilon_1 \ \varepsilon_2 \ \varepsilon_3$	Normal strains

$\gamma_{12}, \gamma_{23}, \gamma_{31}$	Engineering shear strains
E_1, E_2, E_3	Young's moduli
$\nu_{12}, \nu_{21}, \nu_{13}, \nu_{31}, \nu_{23}, \nu_{32}$	Poisson's ratios
G_{12}, G_{23}, G_{31}	Shear modulus
A_1, A_2, A_3	Coefficients of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

Twelve elastic constants are included in Equation 7-4. However, the three equalities in Equation 7-5 reduce the number of independent elastic constants to nine. Accordingly, the MAT11 bulk entry requires you to enter nine elastic constants: E_1 , E_2 , E_3 , ν_{12} , ν_{13} , ν_{23} , G_{12} , G_{13} , and G_{23} . Using the elastic constants you define and Equation 7-5, NX Nastran calculates the remaining terms in Equation 7-4.

MAT11 Format

The format of the MAT11 bulk entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT11	MID	E1	E2	E3	NU12	NU13	NU23	G12	
	G13	G23	RHO	A1	A2	A3	TREF	GE	

Field	Contents
MID	Material identification number
E1	Modulus of elasticity in the longitudinal direction, also defined as the fiber direction or 1-direction
E2	Modulus of elasticity in the lateral direction, also defined as the matrix direction or 2-direction
E3	Modulus of elasticity in the ply layup direction, also defined as the thickness direction or 3-direction
NU12	Poisson's ratio (ϵ_2/ϵ_1 for uniaxial loading in 1-direction)
NU13	Poisson's ratio (ϵ_3/ϵ_1 for uniaxial loading in 1-direction)
NU23	Poisson's ratio (ϵ_3/ϵ_2 for uniaxial loading in 2-direction)
G12	Shear modulus in plane 1-2
G13	Transverse shear modulus in plane 1-3
G23	Transverse shear modulus in plane 2-3
RHO	Mass density
A1	Thermal expansion coefficient in the longitudinal direction
A2	Thermal expansion coefficient in the lateral direction
A3	Thermal expansion coefficient in the thickness direction
TREF	Reference temperature for the calculation of thermal loads

Field	Contents
GE	Structural damping coefficient

See also

- “**MAT11**” in the *NX Nastran Quick Reference Guide*

Understanding the MAT3 Bulk Entry

The MAT3 entry allows you to define orthotropic material behavior for the axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6, CTRIAX6, the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, and the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8. These elements, except for the CTRIAX6, can be defined on either the XZ plane or the XY plane of the basic coordinate system. The CTRIAX6 only supports the XZ plane orientation.

X, Y, and Z on the MAT3 fields designate the material coordinates. The material coordinate system is defined by THETA on the element entries, for example CQUADX4. See [Figure 1](#) and [Figure 2](#)

Elements defined on the XZ plane of the basic coordinate system have different material coordinates than elements defined on the XY plane of the basic coordinate system. As a result, MAT3/MATT3 entries defined for elements on one plane can not be used for elements on the other plane.

Note

When axisymmetric elements are defined on the XZ plane, X is the radial direction, and Z is the axial direction. When axisymmetric elements are defined on the XY plane, Y is the radial direction, and X is the axial direction. See CQUADX4, CQUADX8, CTRAX3, and CTRAX6 in the Quick Reference Guide.

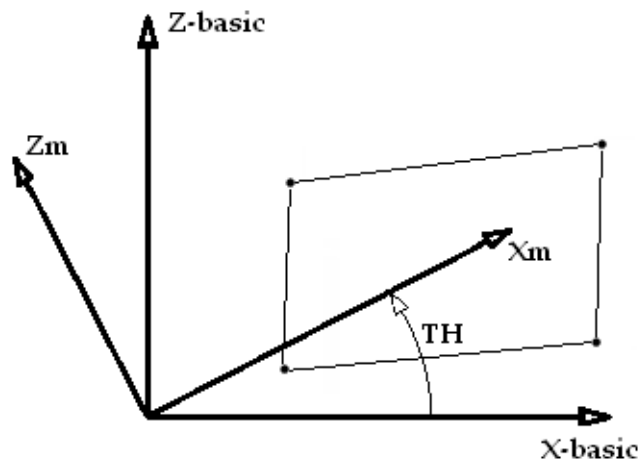


Figure 7-1. Axisymmetric, Plane Stress, and Plane Strain Material Coordinate System, XZ-Plane

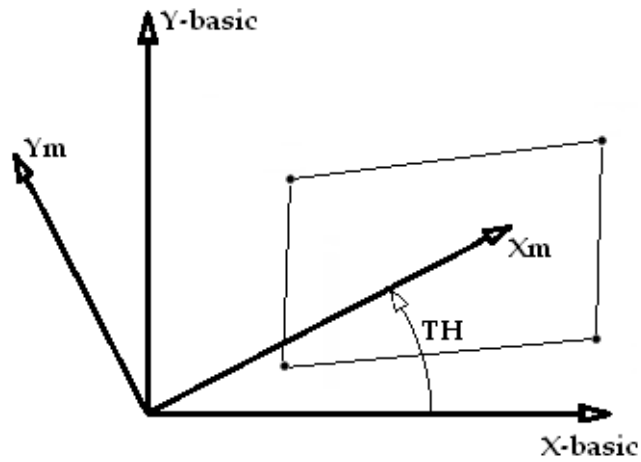


Figure 7-2. Axisymmetric, Plane Stress, and Plane Strain Element Geometry and Coordinate Systems, XY-Plane

Axisymmetry Elements with Orthotropic Materials

The strain-stress relation in material coordinates for axisymmetric elements oriented on the XZ plane is given by:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & -\frac{\nu_{zx}}{E_z} & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & -\frac{\nu_{zy}}{E_z} & 0 \\ -\frac{\nu_{xz}}{E_x} & -\frac{\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - T_{ref}) \begin{Bmatrix} A_x \\ A_y \\ A_z \\ 0 \end{Bmatrix}$$

Equation 7-6.

where

$$\frac{\nu_{xy}}{E_x} = \frac{\nu_{yx}}{E_y} \quad \frac{\nu_{xz}}{E_x} = \frac{\nu_{zx}}{E_z} \quad \frac{\nu_{yz}}{E_y} = \frac{\nu_{zy}}{E_z}$$

Equation 7-7.

and

$\sigma_x \sigma_y \sigma_z$

Normal stresses

τ_{zx}	Shear stress
$\varepsilon_x, \varepsilon_y, \varepsilon_z$	Normal strains
γ_{zx}	Engineering shear strain
E_x, E_y, E_z	Young's moduli
$\nu_{yx}, \nu_{zx}, \nu_{xy}, \nu_{zy}, \nu_{xz}, \nu_{yz}$	Poisson's ratios
G_{zx}	Shear modulus when elements are modeled on the XZ plane of the basic coordinate system
A_x, A_y, A_z	Coefficients of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

For axisymmetric elements oriented on the XZ plane, you must define the elastic constants: E_x , E_y , E_z , ν_{xy} , ν_{yz} , ν_{zx} , and G_{zx} .

For axisymmetric elements oriented on the XY plane, in the equations above, Y replaces X, X replaces Z, Z replaces Y. The shear modulus for the XY orientation is G_{xy} .

Plane Stress Elements with Orthotropic Materials

The strain-stress relation in material coordinates for plane stress elements oriented on the XZ plane is given by:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{zx}}{E_z} & 0 \\ -\frac{\nu_{xz}}{E_x} & \frac{1}{E_z} & 0 \\ 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - T_{ref}) \begin{Bmatrix} A_x \\ A_z \\ 0 \end{Bmatrix}$$

where

$$\frac{\nu_{xz}}{E_x} = \frac{\nu_{zx}}{E_z}$$

and

$$\varepsilon_y = \frac{\nu_{xy}}{E_x} \sigma_x - \frac{\nu_{zy}}{E_z} \sigma_z + (T - T_{ref}) A_y$$

For plane stress elements oriented on the XZ plane, you must define the elastic constants: E_x , E_z , ν_{xy} , ν_{yz} , ν_{zx} , and G_{zx} .

For plane stress elements oriented on the XY plane, in the equations above, Y replaces X, X replaces Z, Z replaces Y. The shear modulus for the XY orientation is G_{xy} .

Plane Strain Elements with Orthotropic Materials

The strain-stress relation in material coordinates for plane strain elements oriented on the XZ plane is given by:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_z \\ \gamma_{xz} \end{Bmatrix} = \begin{bmatrix} \frac{1 - \nu_{yx} \nu_{xy}}{E_x} & \frac{-\nu_{zx} - \nu_{yx} \nu_{zy}}{E_z} & 0 \\ -\frac{\nu_{xz} - \nu_{yx} \nu_{xy}}{E_x} & \frac{1 - \nu_{yz} \nu_{zy}}{E_z} & 0 \\ 0 & 0 & \frac{1}{G_{xz}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_z \\ \tau_{xz} \end{Bmatrix} + (T - T_{ref}) \begin{Bmatrix} A_x + \nu_{yx} A_y \\ A_z + \nu_{yz} A_y \\ 0 \end{Bmatrix}$$

where

$$\frac{\nu_{xz}}{E_x} = \frac{\nu_{zx}}{E_z}$$

and

$$\sigma_y = E_y \left(\frac{\nu_{xy}}{E_x} \sigma_x + \frac{\nu_{zy}}{E_z} \sigma_z - (T - T_{ref}) A_y \right)$$

For plane strain elements oriented on the XZ plane, you must define the elastic constants: E_x , E_z , ν_{xy} , ν_{yz} , ν_{zx} , and G_{zx} .

For plane strain elements oriented on the XY plane, in the equations above, Y replaces X, X replaces Z, Z replaces Y. The shear modulus for the XY orientation is G_{xy} .

MAT3 Format

The format of the MAT3 bulk entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	EY	EZ	NUXY	NUYZ	NUZX	RHO	
	GXY		GZX	AX	AY	AZ	TREF	GE	

Field

Contents

MID Material identification number

EX, EY, EZ Young's moduli

NUXY, NUYZ, NUZX Poisson's ratios

RHO Mass density

GZX Shear modulus when elements are defined on the XZ plane of the basic coordinate system

Field	Contents
GXY	Shear modulus when elements are defined on the XY plane of the basic coordinate system
AX, AY, AZ	Thermal expansion coefficients
TREF	Reference temperature

See also

- “**MAT3**” in the *NX Nastran Quick Reference Guide*

Understanding the MAT8 Entry

The MAT8 bulk entry is used to define orthotropic materials for 2D (plate and shell) elements. For a 2D stress state, $\sigma_3 = \tau_{23} = \tau_{31} = 0$ and the constitutive relation expressed in terms of the principal material directions is given by:

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & -\frac{\nu_{21}}{E_2} & 0 \\ -\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} + (T - T_{ref}) \begin{Bmatrix} A_1 \\ A_2 \\ 0 \end{Bmatrix}$$

Equation 7-8.

subject to:

$$\frac{\nu_{12}}{E_1} = \frac{\nu_{21}}{E_2}$$

Equation 7-9.

where:

$\sigma_1 \sigma_2$	Normal stresses
τ_{12}	Shear stress
$\varepsilon_1 \varepsilon_2$	Normal strains
γ_{12}	Engineering shear strain
$E_1 E_2$	Young's moduli
$\nu_{12} \nu_{21}$	Poisson's ratios
G_{12}	Shear modulus
$A_1 A_2$	Coefficients of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

Five elastic constants are included in Equation 7-8. However, Equation 7-9 reduces the number of independent elastic constants to four. Accordingly, the MAT8 bulk entry requires you to enter four

elastic constants: E_1 , E_2 , ν_{12} , and G_{12} . Using the elastic constants you define and Equation 7-9, NX Nastran calculates the remaining terms in Equation 7-8.

To account for transverse shear stresses, you can enter values for G_{1z} and G_{2z} on the MAT8 entry where the z-direction is through the thickness of the 2D element. The transverse shear stresses are related to the transverse shear strains by:

$$\begin{Bmatrix} \tau_{1z} \\ \tau_{2z} \end{Bmatrix} = \begin{bmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{bmatrix} \begin{Bmatrix} \gamma_{1z} \\ \gamma_{2z} \end{Bmatrix}$$

Equation 7-10.

where:

τ_{1z} τ_{2z}	Transverse shear stresses
γ_{1z} γ_{2z}	Engineering shear strains
G_{1z} G_{2z}	Shear moduli

When G_{1z} or G_{2z} are input as zero, NX Nastran neglects the corresponding effects of transverse shear flexibility in the shell formulation.

You can use the MAT8 bulk entry in conjunction with either a PCOMP or PCOMPG property bulk entry to define composite lamina properties for thin shell elements.

See also

- “PCOMP and PCOMPG” in the *NX Nastran Quick Reference Guide*

MAT8 Format

The format of the MAT8 bulk entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						

Field

Contents

MID	Material identification number
E1	Modulus of elasticity in the longitudinal direction
E2	Modulus of elasticity in the lateral direction
NU12	Poisson's ratio (ϵ_2/ϵ_1 for uniaxial loading in 1-direction)
G12	In-plane shear modulus
G1Z	Transverse shear modulus for shear in the 1-Z plane

Field	Contents
G2Z	Transverse shear modulus for shear in the 2-Z plane
RHO	Mass density
Ai	Thermal expansion coefficient in the i-direction
TREF	Reference temperature
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction
S	Allowable stress or strain for in-plane shear
GE	Structural damping coefficient
F12	Interaction term used in the Tsai-Wu theory
STRN	Request for maximum strain theory

See also

- “**MAT8**” in the *NX Nastran Quick Reference Guide*

MAT8 Example

The composite tube structure shown in Figure 7-3 illustrates the modeling of an orthotropic material. The longitudinal direction is the fiber direction and is oriented at an angle of 45 degrees relative to the tube axes as shown. The tube is held fixed at one end, and a uniform axial load is applied at the other end.

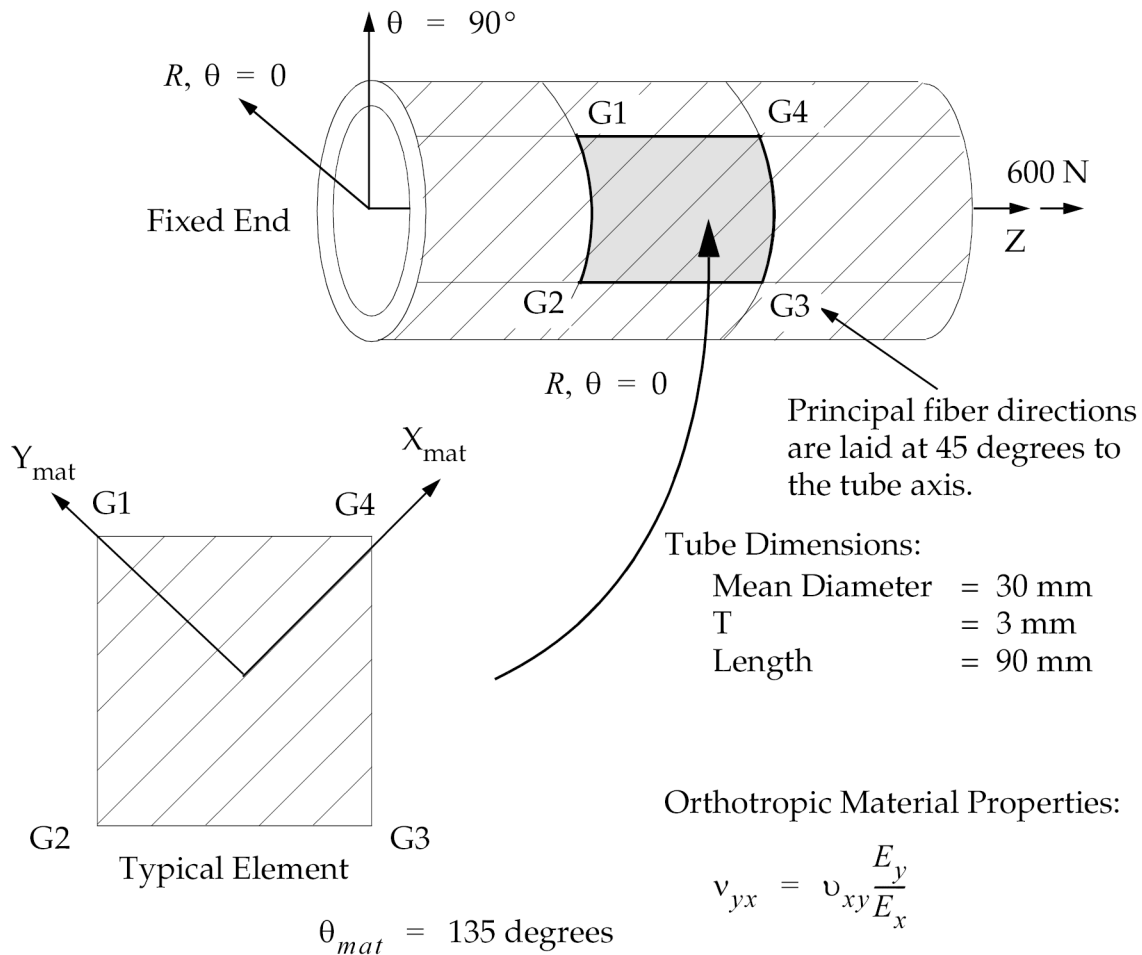


Figure 7-3. Composite Tube with Orthotropic Properties

A sample of the tube material has been tested, and the material properties are determined in the longitudinal (X_{mat}) and transverse (Y_{mat}) directions. The material properties are shown in Table 7-11.

Table 7-11. Composite Tube Material Properties			
	Longitudinal Direction	Transverse Direction	Shear
E	$25 \cdot 10^4 \frac{N}{mm^2}$	$10 \cdot 10^4 \frac{N}{mm^2}$	—
v	0.4	0.16	—
G_{12}	—	—	$80,000 \frac{N}{mm^2}$

Listing 7-1 shows the input file. The geometry has been defined with a local cylindrical coordinate system for convenience. This does not affect the orientation of the material coordinate system.

```
$
$ GEOMETRY DEFINED AND OUTPUT IN THE CYL COORD SYS 1
```

```

$
CORD2C  1      0      0.0      0.0      0.0      0.0      0.0      1
        1.      0.0      0.0
$
GRID    110     1      15.      150.     30.      1
GRID    111     1      15.      150.     60.      1
GRID    114     1      15.     -150.     30.      1
GRID    115     1      15.     -150.     60.      1
$
CQUAD4  8      1      110      114      115      111      135.
$
PSHELL  1      1      3.      1
$
MAT8    1      250000. 100000. .4      80000.  80000.  80000.
$

```

Listing 7-1. Orthotropic Tube Input File

Note

Only element 8 and its connected grid points are shown.

The finite element model for the tube is shown in Figure 7-4. Element 8 is highlighted to illustrate how the material coordinate system is used. The material coordinate system is oriented at an angle of 135 degrees (field 8 of the CQUAD4 entry). The 135 degrees is with respect to a line drawn from G1 (which has a grid ID of 110) to G2 (which has a grid ID of 114). The resulting material coordinate system is therefore the same as that shown in Figure 7-3. If you look at the deformed model, it is interesting to note that the tube rotates about the z-axis even though a uniform axial load is applied. This rotation is due to the orthotropic material behavior and the fact that the fiber direction is oriented in a different direction than the applied load.

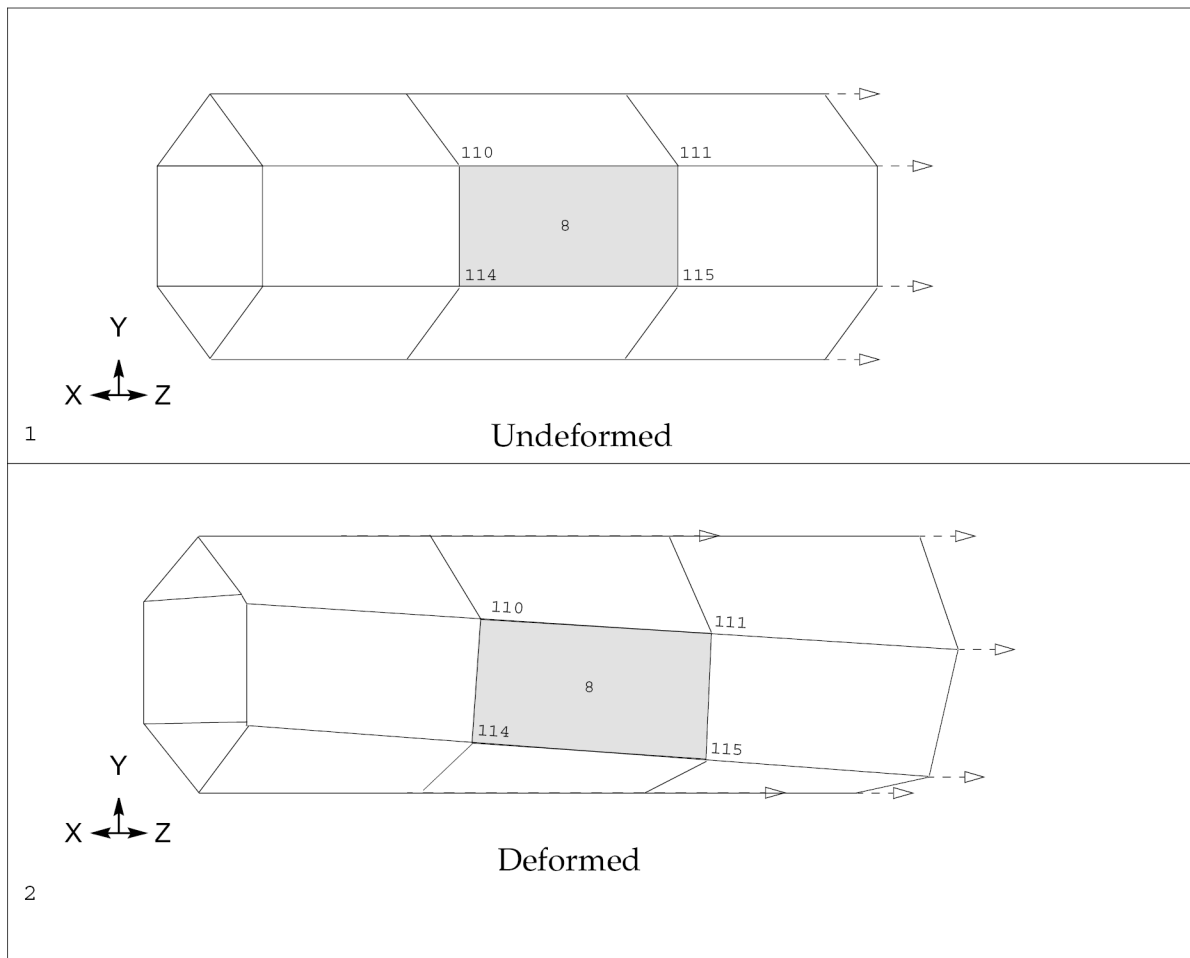


Figure 7-4. Finite Element Model of the Orthotropic Tube

Anisotropic Materials

Materials that have no planes of symmetry for the material properties are termed anisotropic. Frequently, they are also referred to as triclinic or aelotropic. You can define anisotropic materials using the MAT2 and MAT9 bulk entries. You can also define other classes of material behavior like monoclinic, orthotropic, transverse isotropic, and isotropic through proper specification of the elastic constants.

- MAT2 allows you to define anisotropic material properties for 2D elements.
- MAT9 allows you to define anisotropic material properties for 3D elements.

To characterize an anisotropic material for a three-dimensional stress state, 21 independent elastic constants are required. For a two-dimensional stress state, the number of independent elastic constants is reduced to six.

Understanding MAT9

The MAT9 bulk entry is used to define anisotropic material behavior for 3D (solid) elements. The constitutive relation is:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ & & C_{33} & C_{34} & C_{35} & C_{36} \\ & & & C_{44} & C_{45} & C_{46} \\ & \text{symmetric} & & & C_{55} & C_{56} \\ & & & & & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix}$$

Equation 7-11.

where:

$\sigma_x \sigma_y \sigma_z$	Normal stresses
$\tau_{xy} \tau_{yz} \tau_{zx}$	Shear stresses
$\varepsilon_x \varepsilon_y \varepsilon_z$	Normal strains
$\gamma_{xy} \gamma_{yz} \gamma_{zx}$	Engineering shear strains
C_{ij}	Elements of the 6 × 6 symmetric material property matrix in the material coordinate system
A_i	Coefficients of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

MAT9 Format

The format of the MAT9 bulk entry is as follows:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	C11	C12	C13	C14	C15	C16	C22	
	C23	C24	C25	C26	C33	C34	C35	C36	
	C44	C45	C46	C55	C56	C66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

Field	Contents
MID	Material identification number
Cij	Elements of the 6 × 6 symmetric material property matrix in the material coordinate system
RHO	Mass density
Ai	Thermal expansion coefficients

Field	Contents
TREF	Reference temperature
GE	Structural damping coefficient

When you use the MAT9 entry, you should define a material coordinate system on the PSOLID entry (using field 4). For solid elements, NX Nastran outputs stresses in the material coordinate system, which by default, is the basic coordinate system. In general, for solid elements, it is difficult to determine the orientation of the element coordinate system.

See also

- “MAT9” in the *NX Nastran Quick Reference Guide*
- “PSOLID” in the *NX Nastran Quick Reference Guide*

MAT9 Example

The following example illustrates the use of a MAT9 bulk entry to define an anisotropic material. This example consists of three steps:

1. Obtain the elastic constants using a pseudo-anisotropic model consisting of a single CHEXA element and CBAR elements applied along the edges and across the diagonals of the CHEXA element.
2. Verify that the constitutive relation developed in the previous step is correct.
3. Use the constitutive relation in a structural analysis.

This three-step approach is analogous to constructing an anisotropic material test coupon and then testing that coupon to obtain a strain matrix. From the strain matrix, you can develop the constitutive relation. Then, you can verify the constitutive relation by modeling the test coupon. Finally, you use the newly developed constitutive relation in a structural analysis.

Step 1 – Develop the Material Matrix

To simulate a three-dimensional anisotropic material, this example uses a single CHEXA element with CBAR elements of various cross-sectional areas and bending stiffness defined along the edges and across the diagonals of the CHEXA element as shown in Figure 7-5(a). The cross sections of the bars are not important, but if you are interested in the actual dimensions, refer to the input file “anis1.dat” in the Test Problem Library located on the delivery media. The dimensions of the cube are 1 x 1 x 1 inches. To constrain the cube, one of the corner grid points is fixed in all six component directions.

Six self-equilibrating load cases are used to represent each of the six stress components. Each direction was applied as a separate subcase. A self-equilibrating normal load consists of applying equal and opposite forces to opposite faces of the cube (i.e., forces F_1 , F_2 , and F_3).

For the shear load, a self-equilibrating load consists of applying four forces around the cube in order to place the cube in a state of pure shear. Note that for all self-equilibrating load cases, the net resultant should be zero; hence, the SPC force should also be zero. Because of the dimensions chosen for the CHEXA element, each load case represents a unit stress, and the resulting strains are the strains due to unit stress.

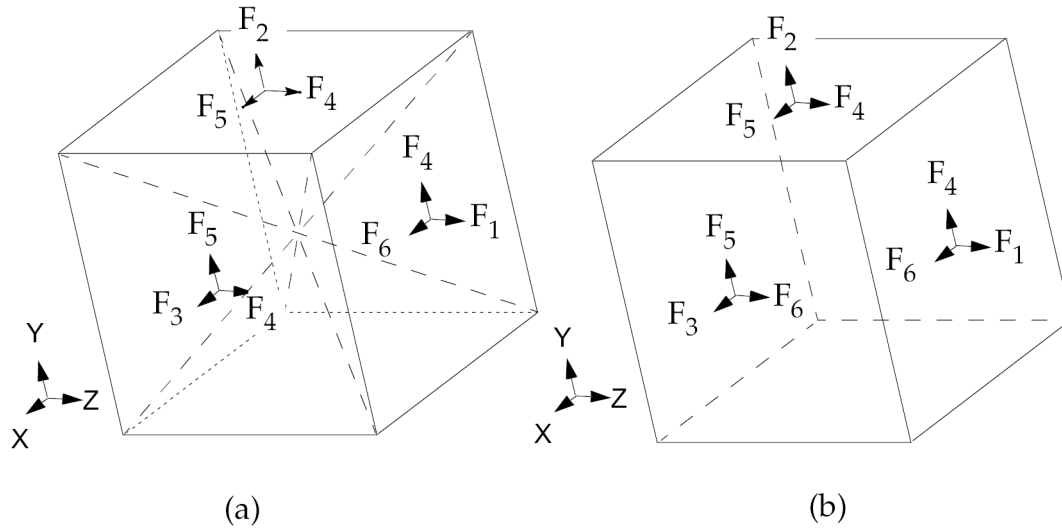


Figure 7-5. Pseudo-Anisotropic Model

The strains at the center of the CHEXA were extracted from the output file and are summarized in Table 7-12.

Table 7-12. Strains Due to Unit Loads on Pseudo Anisotropic Model							
		Applied Unit Load					
		FX	FY	FZ	FX Y	FY Z	FZ X
Resulting Strains (10^{-6})	ϵ_x	0.155	-0.034	-0.056	-0.016	0.002	-0.011
	ϵ_y	-0.034	0.072	-0.018	-0.005	0.001	-0.003
	ϵ_z	-0.056	-0.018	0.107	-0.008	0.003	-0.005
	γ_{xy}	-0.016	-0.005	-0.008	0.197	-0.011	0.009
	γ_{yz}	0.002	0.001	0.003	-0.011	0.219	-0.021
	γ_{zx}	-0.011	-0.003	-0.005	0.009	-0.021	0.180

Table 7-12 represents a strain matrix that, when inverted, is the material constitutive matrix $[C]$. Note that since the strain matrix is symmetric, the material constitutive matrix $[C]$ is also symmetric.

The result is the material constitutive matrix $[C]$ for the anisotropic material is given by Equation 7-12.

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

Equation 7-12.

Step 2 – Verify the Material Matrix

Now that the material constitutive matrix has been generated, the next step is to verify that the matrix is entered on the MAT9 entry correctly. To do so, return to the 1 x 1 x 1 inch cube. However, this time the analysis will not include the CBAR elements. The same six load cases are applied as in Step 1.

If the material constitutive matrix is correct, the strains produced should be numerically the same as those shown in Table 7-12.

The input file for the verification model is shown in Listing 7-2.

```
$
$ FILENAME - ANIS2.DAT
$
ID      LINEAR,ANIS2
SOL     101
TIME    5
CEND
TITLE = ANISOTROPIC TEST ELEMENT
SPC = 1
SET 1 =13
STRAIN = 1
SPCFORCE=ALL
SUBCASE 1
  LOAD = 1
SUBCASE 2
  LOAD = 2
SUBCASE 3
  LOAD = 3
SUBCASE 4
  LOAD = 4
SUBCASE 5
  LOAD = 5
SUBCASE 6
  LOAD = 6
BEGIN BULK$
GRID    1          0.0      0.0      0.0
GRID    2          1.       0.0      0.0
GRID    3          0.0      1.       0.0
GRID    4          1.       1.       0.0
GRID    6          0.0      0.0      1.
GRID    8          1.       0.0      1.
GRID   10          0.0      1.       1.
GRID   12          1.       1.       1.
CHEXA   13         1        1        6        8        2        3        10      +
+       12         4
$
PLOAD4  1         13        1.          2        12      +
+         1.
PLOAD4  1         13        1.          10         1      +
+        -1.
PLOAD4  2         13        1.          10         4      +
+         0.0      1.
PLOAD4  2         13        1.          6          2      +
+         0.0      -1.
PLOAD4  3         13        1.          6          12     +
+         0.0      0.0      1.
PLOAD4  3         13        1.          1          4      +
+         0.0      0.0     -1.
PLOAD4  4         13        1.          10         4      +
+         1.       0.0
PLOAD4  4         13        1.          6          2      +
+        -1.       0.0
PLOAD4  4         13        1.          12         2      +
+         1.
```



```

PLOAD4 4      13      1.          1      10      +
+
PLOAD4 5      13      1.          10      4      +
+
PLOAD4 5      13      1.          6       2      +
+
PLOAD4 5      13      1.          6      12      +
+
PLOAD4 5      13      1.          1       4      +
+
PLOAD4 6      13      1.          6      12      +
+
PLOAD4 6      13      1.          1       4      +
+
PLOAD4 6      13      1.          2      12      +
+
PLOAD4 6      13      1.          1      10      +
+
$
PSOLID 1      2      0
$
MAT9 2      10.67+6  6.91+6  6.89+6  1.25+6  -0.09+6  0.86+6  19.03+6
      6.90+6  1.27+6  -0.12+6  0.83+6  14.28+6  1.23+6  -0.15+6  0.82+6
      5.27+6  0.21+6  -0.11+6  4.63+6  0.52+6  5.72+6
SPC1 1      123      6
SPC1 1      12      1
SPC1 1      2       2
ENDDATA

```

Listing 7-2. Anisotropic Verification Input File

In the verification model input file, the six loads are applied separately, each with their own subcase. The constraints are applied to three separate grid points and are non-redundant. The non-redundant constraint set allows the element to expand in all directions without imposing any constraint forces while preventing rigid body motion. Also note that the basic coordinate system is used for the material definition.

The resulting center strains for the verification model are shown in Figure 7-6. As can be seen, there is good agreement between these values and those in Table 7-12.

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

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

Figure 7-6. Verification Model Strain Output

Step 3 - Use the Anisotropic Material in a Structural Analysis

The last step in the example is to use the material properties you found to model the structure depicted in Figure 7-7. The model consists of four elements with their material coordinate system oriented 45 degrees with respect to the basic coordinate system.

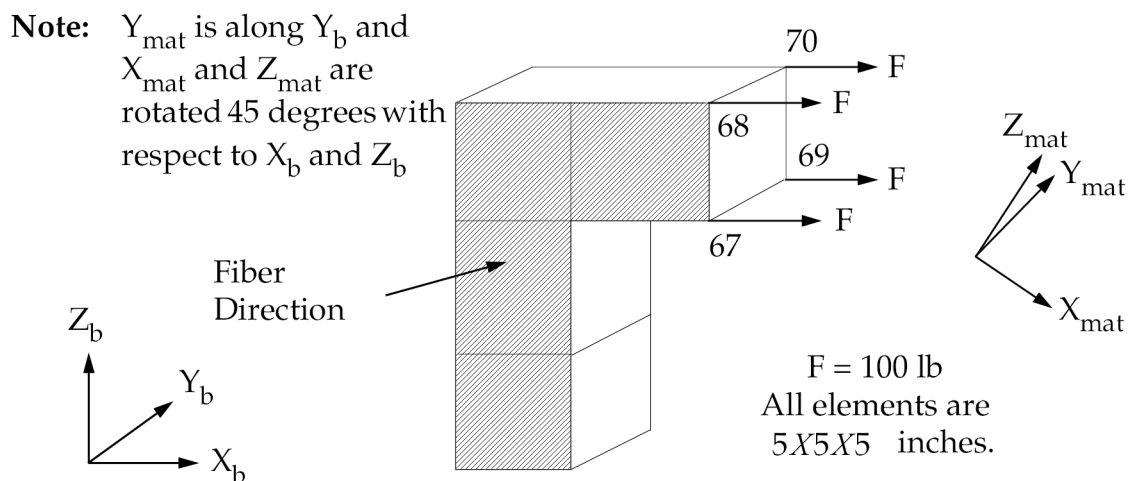


Figure 7-7. Anisotropic Solids Model

The input file is shown in Listing 7-3. Note the local coordinate system used to define the material orientation. The displacement results are shown in Figure 7-8.

```

$ FILENAME - ANIS3.DAT
$
ID      LINEAR, ANIS3
SOL     101
TIME    5
CEND
TITLE = USING THE MATERIAL MATRIX
LOAD = 1
SPC = 1
DISP = ALL
BEGIN BULK
PARAM   POST      0
PARAM   AUTOSPC YES
$ MATERIAL COORDINATE SYSTEM
$
CORD2R  1      0      0.0    0.0    0.0    .70710680.0    .7071068+
        .70710680.0    -.707107
GRID    51      0.0    5.    0.0
GRID    52      5.    5.    0.0
GRID    53      0.0    5.    5.
GRID    54      5.    5.    5.
GRID    55      0.0    0.0    0.0
GRID    56      5.    0.0    0.0
GRID    57      0.0    0.0    5.
GRID    58      5.    0.0    5.
GRID    59      0.0    5.    10.
GRID    60      5.    5.    10.
GRID    61      0.0    0.0    10.
GRID    62      5.    0.0    10.
GRID    63      0.0    5.    15.
GRID    64      5.    5.    15.
GRID    65      0.0    0.0    15.
GRID    66      5.    0.0    15.
GRID    67      10.    5.    10.
GRID    68      10.    5.    15.
GRID    69      10.    0.0    10.
GRID    70      10.    0.0    15.
$
CHEXA   1      1      51      55      56      52      53      57      +
+       58      54
CHEXA   2      1      53      57      58      54      59      61      +
+       62      60
CHEXA   3      1      59      61      62      60      63      65      +
+       66      64
CHEXA   4      1      60      62      69      67      64      66      +
+       70      68
$
$
FORCE   1      69      0      100.    1.    0.0    0.0
FORCE   1      67      0      100.    1.    0.0    0.0
FORCE   1      68      0      100.    1.    0.0    0.0
FORCE   1      70      0      100.    1.    0.0    0.0
$
PSOLID  1      2      1      GRID      SMECH
$
MAT9    2      10.67+6  6.91+6  6.89+6  1.25+6  -0.09+6  0.86+6  19.03+6
        6.90+6  1.27+6  -0.12+6  0.83+6  14.28+6  1.23+6  -0.15+6  0.82+6
        5.27+6  0.21+6  -0.11+6  4.63+6  0.52+6  5.72+6
$
SPC     1      51      123456  0.0
SPC     1      52      123456  0.0
SPC     1      56      123456  0.0
SPC     1      55      123456  0.0
$

```

ENDDATA

Listing 7-3. Structure with Anisotropic Material Input File

D I S P L A C E M E N T V E C T O R							
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
51	G	0.0	0.0	0.0	0.0	0.0	0.0
52	G	0.0	0.0	0.0	0.0	0.0	0.0
53	G	1.232738E-04	3.251065E-07	8.268995E-05	0.0	0.0	0.0
54	G	1.204173E-04	1.783952E-06	-8.797066E-05	0.0	0.0	0.0
55	G	0.0	0.0	0.0	0.0	0.0	0.0
56	G	0.0	0.0	0.0	0.0	0.0	0.0
57	G	1.002035E-04	2.286116E-05	7.448530E-05	0.0	0.0	0.0
58	G	9.797833E-05	-2.485534E-05	-7.772400E-05	0.0	0.0	0.0
59	G	3.605538E-04	1.530884E-05	1.199663E-04	0.0	0.0	0.0
60	G	3.610890E-04	-1.436989E-05	-1.294236E-04	0.0	0.0	0.0
61	G	3.284150E-04	1.772471E-05	1.189002E-04	0.0	0.0	0.0
62	G	3.284143E-04	-1.659684E-05	-1.265683E-04	0.0	0.0	0.0
63	G	6.271876E-04	1.717129E-05	1.256454E-04	0.0	0.0	0.0
64	G	6.300415E-04	-1.526032E-05	-1.351714E-04	0.0	0.0	0.0
65	G	5.944516E-04	1.910164E-05	1.262980E-04	0.0	0.0	0.0
66	G	5.975869E-04	-1.427919E-05	-1.336644E-04	0.0	0.0	0.0
67	G	3.671078E-04	-5.154453E-05	-4.031157E-04	0.0	0.0	0.0
68	G	6.358424E-04	-4.903843E-05	-4.012304E-04	0.0	0.0	0.0
69	G	3.339844E-04	-4.717636E-05	-4.023441E-04	0.0	0.0	0.0
70	G	6.039479E-04	-4.626662E-05	-3.997554E-04	0.0	0.0	0.0

Figure 7-8. Output for Structure with Anisotropic Material

Understanding MAT2

The MAT2 bulk entry is used to define anisotropic material behavior for 2D (plate and shell) elements. When using a MAT2 bulk entry, you define the in-plane material properties with respect to an element material coordinate system. The constitutive relation is given by:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{12} & C_{22} & C_{23} \\ C_{13} & C_{23} & C_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A_1 \\ A_2 \\ A_{12} \end{Bmatrix}$$

Equation 7-13.

where:

$\sigma_1 \sigma_2$	Normal stresses
τ_{12}	Shear stress
$\varepsilon_1 \varepsilon_2$	Normal strains
γ_{12}	Engineering shear strain
C_{ij}	Elements of the 3×3 symmetric material property matrix in the material coordinate system
$A_1 A_2 A_{12}$	Coefficients of thermal expansion
$(T - T_{ref})$	Temperature difference used to calculate thermal strain

You can also include material properties for transverse shear. When a MAT2 bulk entry is used for transverse shear, the transverse shear constitutive relation is:

$$\begin{pmatrix} \sigma_{zx} \\ \sigma_{zy} \end{pmatrix} = \begin{bmatrix} C_{11} & C_{12} \\ C_{12} & C_{22} \end{bmatrix} \begin{pmatrix} \gamma_{zx} \\ \gamma_{zy} \end{pmatrix}$$

Equation 7-14.

C_{33} must be set to 0.

MAT2 Format

1	2	3	4	5	6	7	8	9	10
MAT2	MID	C11	C12	C13	C22	C23	C33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID								

Field

Contents

MID	Material identification number
C_{ij}	Elements of the 3×3 symmetric material property matrix in the material coordinate system
RHO	Mass density
A_i	Thermal expansion coefficient vector
TREF	Reference temperature
GE	Structural damping coefficient
ST, SC, SS	Stress limits for tension, compression, and shear
MCSID	Material coordinate system identification number

See also

- “**MAT2**” in the *NX Nastran Quick Reference Guide*

7.3 Material Properties for Nonlinear Analysis

In NX Nastran, there are several different types of nonlinear analysis:

- With a nonlinear elastic analysis, you can use the MAT1 bulk entry in conjunction with the MATS1 and TABLES1 entries to define the stress-dependent material properties.
- With an elastic-plastic analysis, you can use the MAT1 bulk entry in conjunction with the MATS1 and TABLES1 entries to define the stress-dependent material properties.
- With a viscoelastic analysis, you can define elastoplastic properties to consider plastic deformation using the MATS1 bulk entry in conjunction with MAT1.

- With a temperature dependent analysis, you can specify temperature-dependent properties by using the MATTi, TABLEST, and TABLES1 bulk entries.
- With a SOL 601 analysis, you can use the MATG entry (described below) to define gasket materials.
- With a hyperelastic material analysis, you can use the MATHP entry (described below) to define the material properties.

With a geometric nonlinear analysis, you can use either MAT2 or MAT9 to define material properties.

See also

- “Material and Geometric Nonlinearity” in the *NX Nastran Basic Nonlinear Analysis User's Guide*

Understanding MATG

The MATG entry lets you define material properties for gasket-type materials for use in a SOL 601 analysis.

See also

- “**MATG**” in the *NX Nastran Quick Reference Guide*

Understanding MATHP

In NX Nastran, you can use the MATHP bulk entry to specify material properties for use in a fully nonlinear hyperelastic analysis.

The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{NA} A_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \sum_{i=1}^{ND} D_i (J - 1 - AV(T - T_0))^{2i}$$

$$A_{00} = 0$$

Equation 7-15.

and $2D_1 = K$ and $2(A_{10} + A_{01}) = G$ at small strains where:

\bar{I}_1	First distortional strain invariant
\bar{I}_2	Second distortional strain invariant
A_{ij}	Material constants related to shear deformation (distortion)
D_i	Material constants related to volumetric deformation
T	Current temperature
T_0	Initial temperature
K	Bulk modulus

G Shear modulus

The model reduces to a Mooney-Rivlin material if $N_a = 1$ and to a Neo-Hookean material if $N_a = 1$ and $A_{01} = 0$.

Using MATHP to Define Material Constants

The MATHP entry consists of five continuation entries that let you define the A_{ij} and D_i material constants of the strain energy function. You obtain the values to enter in the A_{ij} and D_i fields from least squares fitting of experimental data.

If you do not know the material constants, enter one of the following strings in the sixth continuation entry of MATHP to invoke NX Nastran's curve-fitting algorithm:

TAB1, TAB2, TAB3, TAB4, and TABD

These TABi fields reference experimental data that is contained in TABLES1 bulk entries.

- You can use one or more of four experiments (TAB1 to TAB4 options) to obtain A_{ij} .
- You can use pure volumetric compression data (TABD option) to obtain D_i .

If $ND = 10$ (order of the volumetric strain energy polynomial function) and you either enter a nonzero value for $D1$ or a nonzero value for $D1$ is calculated from experimental data in TABD, then the parameter estimation of the material constants A_{ij} takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, the software assumes full incompressibility in estimating the material constants.

See also

- [“Defining Hyperelastic Materials in NX Nastran”](#) in the *NX Nastran Basic Nonlinear Analysis User's Guide*

Using MATHP to Model Mooney-Rivlin or Neo-Hookean Materials

If you are modeling a Mooney-Rivlin or Neo-Hookean material, you do not need to specify a continuation entry for MATHP. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. With the MATHP entry, you cannot define full incompressibility. However, you can simulate full incompressibility by specifying a large enough value for the $D1$ entry (field 5). However, entering a value of $D1$ greater than $10^3 \cdot (A_{10} + A_{01})$ is not recommended.

See also

- [“MATHP”](#) in the *NX Nastran Quick Reference Guide*

7.4 Material Properties for Heat Transfer

Material properties for heat transfer analysis are defined using MAT4, MAT5, MATT4, MATT5, RADM and RADMT entries.

- The MAT4, MAT5, and RADM entries provide temperature independent (constant valued) material properties.

- The MATT4, MATT5, and RADMT entries supply information about temperature-dependent properties through reference to material tables (TABLEMi (i = 1, 2, 3, 4)).

With heat transfer material properties:

- Although the surface characteristics for free convection (heat transfer coefficient) and radiation (emissivity and absorptivity) are not normally considered to be material properties, they are included in the bulk entries described here.
- Temperature dependence of the quantity of interest is ultimately defined on a TABLEMi entry. For example, the table is connected to the MAT4 through the MATT4 entry. Most material properties are directed toward a structural element and are referenced by the property entry for the element of interest.

Understanding MAT4 and MATT4

The MAT4 bulk entry lets you define thermal properties for temperature-independent isotropic materials. The format of the MAT4 bulk entry is:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	ρ	H	μ	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

where:

Symbol	Property	Consistent Units
K(T)	Thermal Conductivity*	W/m °C
CP(T)	Specific Heat*	J/kg °C
ρ	Density*	kg/m ³
H(T)	Free Convection Heat Transfer Coefficient	W/m ² °C
μ (T)	Dynamic Viscosity*	N sec/m ²
HGEN(T)	Volumetric Internal Heat Generation	W/m ³
REFENTH	Reference Enthalpy	J/kg
TCH	Lower Temperature Limit for Phase Change	°C
TDELTA	Temperature Range for Phase Change	°C
QLAT	Latent Heat	J/kg

* Thermal conductivity, specific heat, density, and dynamic viscosity cannot be time dependent. Thermal heat flux loads can reference explicit functions of time. Thermal boundary conditions for convection and radiation utilize control node techniques for specifying time dependent behavior.

The MATT4 entry lets you define temperature-dependent material properties in conjunction with MAT4. NX Nastran always multiplies the basic quantities you specify on the MAT4 entry by the corresponding tabular function (TABLEMi bulk entries) that you reference with the MATT4 entry.

See also

- “Convection and Radiation Boundary Conditions” in the *NX Nastran Thermal Analysis User's Guide*.

- “**MAT4**” in the *NX Nastran Quick Reference Guide*
- “**MATT4**” in the *NX Nastran Quick Reference Guide*

Understanding MAT5 and MATT5

The MAT5 bulk entry lets you define thermal properties for temperature-independent anisotropic materials. The MATT5 entry lets you define temperature-dependent material properties in conjunction with MAT5. NX Nastran always multiplies the basic quantities you specify on the MAT5 entry by the corresponding tabular function (TABLEMi bulk entries) that you reference with the MATT5 entry.

See also

- “**MAT5**” in the *NX Nastran Quick Reference Guide*
- “**MATT5**” in the *NX Nastran Quick Reference Guide*

Understanding RADM and RADMT

The RADM bulk entry lets you define the radiation properties of a boundary element for heat transfer analysis. The format of the RADM bulk entry is:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

The following table describes the fields for the RADM bulk entry.

Field	Description	Unit
ABSORP(T)	Absorptivity for Directional Heat Flux and Radiation Boundary Condition (QVECT and RADBC)	Nondimensional
EMIS(λ ,T)	Emissivity for Radiation Boundary Condition and Enclosure Radiation (RADBC and RADSET)	Nondimensional

The RADMT entry lets you specify table references for properties in a RADM entry that you want to be temperature-dependent.

See also

- “**RADM**” in the *NX Nastran Quick Reference Guide*
- “**RADMT**” in the *NX Nastran Quick Reference Guide*

7.5 Material Properties for Fluid Analysis

In NX Nastran, the MAT10 and MFLUID bulk entries allow you to define material properties for different types of fluid analyses.

Understanding MAT10

The MAT10 bulk entry allows you to define material properties for compressible fluids used in analyses that account for coupled fluid-structural interaction.

See also

- “**MAT10**” in the *NX Nastran Quick Reference Guide*
- “**Coupled Acoustic Analysis**” in the *NX Nastran Advanced Dynamic Analysis User's Guide*
- “**Overview of Coupled Fluid-Structure Interaction**” in the *NX Nastran User's Guide*

Understanding MFLUID

The MFLUID bulk entry allows you to define properties of an incompressible fluid used to generate a virtual mass matrix.

See also

- “**MFLUID**” in the *NX Nastran Quick Reference Guide*
- “Virtual Fluid Mass” in the *NX Nastran Advanced Dynamic Analysis User's Guide*

Chapter 8: Applied Loads

- *Overview of Load Sets*
- *Defining and Combining Subcases for Loading*
- *Defining Static Loads*
- *Point Loads*
- *Distributed Loads*
- *Inertia Loads (Acceleration Loads)*
- *Thermal Loads*
- *Enforced Element Deformation Loads*
- *Loads Due to Enforced Motion*
- *Frequency-dependent Loads*
- *Time-dependent Loads*
- *Using Static Loading Data in Dynamic Analysis*
- *Defining Loads for Cyclic Symmetry Analysis*
- *Combining Loads*
- *Bolt Preload*
- *External Force Fields*

8.1 Overview of Load Sets

Load sets in NX Nastran are collections, or lists, of loads. You define the loads using bulk data entries. You then use case control commands to specify which of the loading entries you want to use in a given analysis.

8.2 Defining and Combining Subcases for Loading

NX Nastran allows you to efficiently analyze multiple load cases in a single analysis using the SUBCASE Case Control command. Each subcase defines a unique loading condition. When you use create subcases:

- Each SUBCASE represents a separate, static loading condition, including different boundary conditions.
- You can select different constraints, loads, and output in each SUBCASE.
- You must list SUBCASE ids within the Case Control section in ascending order. However, the ids don't need to be sequential. For example, you could have SUBCASE 1, 14, 31, and 50.

Within the Case Control section, any title information, output requests, or Bulk Data selections that you make prior to defining the first SUBCASE command are the defaults for all subcases. These defaults apply to all of the subcases unless you specifically override them with a different request within a subcase.

If you don't include a SUBCASE command in the Case Control section of your input file, the software assumes that the model contains a single subcase with an ID of 1.

See also

- “[Defining SUBCASES for Loading](#)” in the *NX Nastran User's Guide*

Example: Creating Two Load Subcases for the Truss

For example, suppose we add a second load case to the truss structure as shown in [Figure 8-1](#) so that we can perform two separate analyses during a single run.

- Subcase 1 consists of the 1000-pound force acting at grid point 4 in the negative Y-direction.
- Subcase 2 consists of the 2000-pound force acting at grid point 3 in the X-direction.

- Subcase 2 consists of the 2000-pound force acting at grid point 3 in the X-direction.

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

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

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

$$\nu = 0.3$$

○ Elements

□ Grid points

• Pin joints

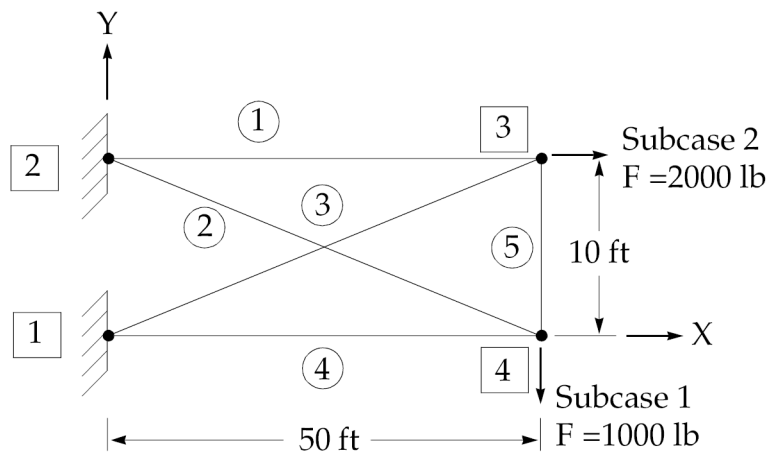


Figure 8-1. Truss Example with Two Loading Conditions

In the input file for this analysis (shown in [Listing 8-1](#)). In this input file, we placed a displacement request (DISPLACEMENT - ALL) above the first subcase but not within either of the two subcases. Therefore, NX Nastran outputs displacements for all the grid points for both subcases. Since we included an element force request only in Subcase 2, NX Nastran doesn't generate any element force output for Subcase 1. It only outputs the element forces for the grid points for Subcase 2.

```

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

```

Listing 8-1. Truss Input File with Two Loadings

Using REPCASE

For large models, you may need to further partition the results in the output files in a certain manner or repeat the same output in different subcases. You can do this with the REPCASE Case Control command. The REPCASE command lets you organize the output based on the element and/or grid point ID numbers.

Listing 8-2 shows how you can use REPCASE separate the results for the truss example into three different sections in the .f06 output file. In this example, the REPCASE 2 command provides for additional output from the solution generated in SUBCASE 1 without performing any additional analysis.

For the Case Control shown in **Listing 8-2**, SUBCASE 1 output consists of the displacement for all of the grid points and the element forces for elements 1 and 2. The output for REPCASE 2 consists of the element force for elements 3 through 5. Note that the default output request for the grid point displacements is overridden by the DISPLACEMENT = NONE command. SUBCASE 3 is a new loading condition with its own output requests. A second static solution is performed for this subcase. Also note that only two static solutions are performed for this input file.

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

Listing 8-2. Truss Input File with the REPCASE Command

See also

- “**REPCASE**” in the *NX Nastran Quick Reference Guide*

Using SUBCOM to Combine Subcases

When you're performing a linear statics analysis, you can use the SUBCOM command to create new subcases by combining other subcases. When you use SUBCOM to combine subcases, you must use the SUBSEQ command to specify the coefficients for combining the previous subcases. A SUBSEQ command is required for each SUBCOM subcase.

The use of the SUBCASE and SUBSEQ commands is best illustrated with an example. Returning to the truss structure, the two load cases were previously treated as separate loads, each with their

own subcase. Now, suppose a third load case is desired that is a linear combination of the first two subcases. The input file shown in [Listing 8-3](#) illustrates this.

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

Listing 8-3. SUBCOM Input File

For this Case Control section, the SUBCOM 3 output is as follows:

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

The output for SUBCOM 4 is as follows:

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

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

You can use SUBCOM commands to combine any number of previous subcases. However, you must define a coefficient on the SUBSEQ command for each subcase. If you want to exclude a subcase from the summation, set the coefficient to 0.0 for that particular subcase.

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

See also

- “**SUBCOM**” in the *NX Nastran Quick Reference Guide*

Using SYMCOM to Create a Symmetry Combination Subcase

In a static analysis, you can use the SYMCOM case control command to create a symmetry combination subcase. When you use the SYMCOM command to combine selected SUBCASES, you must also use a SYMSEQ command to specify the coefficients for combining the symmetry subcases.

See also

- “**SYMCOM**” in the *NX Nastran Quick Reference Guide*
- “**SYMSEQ**” in the *NX Nastran Quick Reference Guide*

Temperature Loads and Enforced Deformations in Subcase Combinations (SUBCOMS)

You can use the SUBCOM and SYMCOM case control commands to combine the results of two or more immediately preceding subcases or SYM cases respectively. In NX Nastran, the results for a combined loading condition are efficiently obtained by forming the linear combination of the displacement vectors obtained in the previous subcases. NX Nastran calculates the element forces and stresses in the usual manner from the displacement vector for the combined loading condition. This technique is quite straightforward when the several loading conditions are composed of forces and moments.

However, if any of the subcases that you're combining contain thermal loads and/or enforced element deformations, you must exercise some care to obtain correct element stresses and forces for the combined loading condition.

See also

- “**Using Temperature Loads with Subcases**” in the *NX Nastran User's Guide*

8.3 Defining Static Loads

In NX Nastran, you can apply static loads to geometric and scalar points in a variety of ways, including

- Loads applied directly to grid points
- Pressure loads on surfaces
- Distributed and concentrated loads on CBAR, CBEAM, and CBEND elements
- Gravity loads

- Centrifugal loads due to steady rotation
- Tangential loads due to angular acceleration
- Loads resulting from thermal expansion
- Loads resulting from enforced deformations of a structural element
- Loads resulting from enforced displacements at a grid point

While you can define multiple load sets in the bulk data section of your input file, the software only uses the sets that you explicitly select in the case control section to solve the model.

See also

- “[Defining SUBCASES for Loading](#)” in the *NX Nastran User's Guide*
- Section 3.6 of *The NASTRAN Theoretical Manual*

8.4 Point Loads

This section shows how to apply concentrated forces and concentrated moments at grid points.

- You can use the FORCE, FORCE1, and FORCE2 entries to apply concentrated forces directly to grid points.
- You can use the MOMENT, MOMENT1, and MOMENT2 entries to apply a concentrated force in the translational degrees of freedom (T1, T2, and T3) at a grid point.
- Use FORCEAX and MOMAX to define forces or moments on conical shell elements.
- Use SLOAD to define a concentrated static load at a scalar point.

FORCE, FORCE1, FORCE2

You can apply concentrated forces directly to grid points with the FORCE, FORCE1, and FORCE2 entries.

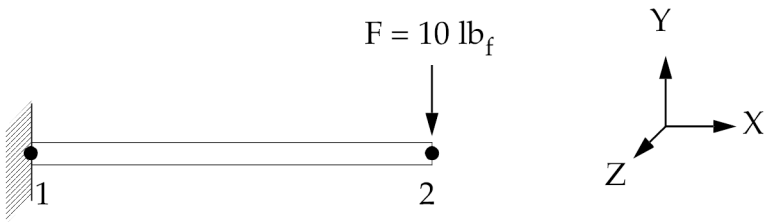
- Use a FORCE entry if you want to define a static, concentrated force at a grid point by specifying a vector.
- Use a FORCE1 entry if the direction is determined by a vector connecting two grid points.
- Use a FORCE2 entry if the direction is specified by the cross product of two such vectors.

Using FORCE

The FORCE entry lets you specify the magnitude and direction of a force vector in any coordinate system as shown below. You can use the FORCE entry to define a static load applied to a geometric grid point in terms of components defined by a local coordinate system. The orientation of the load components depends on the type of local coordinate system you used to define the load. The directions

of the load components are the same as those indicated on **Figure 5-2** of “Defining Grid Points and Scalar Points” for displacement components.

Consider a force F acting on a cantilever beam:



A concentrated force F is applied to a CBAR element connecting grid points 1 and 2 using the FORCE entry. Its format is as follows:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one Ni ≠ 0.0)

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

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

or, in free field format,

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

The load set identification number (SID in column 2) refers to a command defined in the Case Control Section (LOAD=2 in this example; the integer value 2 is arbitrarily chosen). Leaving column 4 blank means that the basic (default) coordinate system is used to specify the orientation of the load. The (0., -1., 0.) entries in columns 6, 7, and 8 refer to a vector in the -Y direction, defining the direction of application of the load. The force applied to the grid point is \vec{f} , given by

$$\vec{f} = F\vec{N}$$

Equation 8-1.

where $\vec{N} = (N1,N2,N3)$

Thus, the value of F in column 5 is the full value of the applied load of 10 lb_f because vector \vec{N} (in this example) is of unit length.

The magnitude of the applied force is the scale factor (field 5) times the magnitude of the vector defined in fields 6 through 8. For example, the force applied with the following two FORCE entries is the same.

1	2	3	4	5	6	7	8	9	10
FORCE	1	3	0	-100.	1.	0.	2.		

1	2	3	4	5	6	7	8	9	10
FORCE	1	3	0	2.	-50.	0.	-100.		

See also

- “**FORCE**” in the *NX Nastran Quick Reference Guide*

Using FORCE1 and FORCE2

- The FORCE1 entry lets you define a force by specifying a magnitude and two grid points (not necessarily the loaded grid point) to determine the direction.
- The FORCE2 entry lets you define a force by specifying a magnitude with the direction defined by the vector product of two other vectors.

The format of the FORCE1 and FORCE2 entries is as follows:

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

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Field	Contents
SID	Load set identification number.
G	Grid point identification number.
F	Magnitude of the force.
Gi	Grid point identification numbers.

FORCE1 and FORCE2 Example

While the FORCE1 and FORCE2 forms are not used extensively, they are useful for some modeling situations as shown in the following example.

Suppose you wish to model a curved structure with edge loads as shown in **Figure 8-2**. The edge loads consist of 10 N per element (acting normal to the elements) and 8 N per element along the edge of each element (acting as a shear load). The FORCE1 and FORCE2 entries are the ideal choice to apply these loads. You could use the FORCE entry, but it would be more difficult because

you would need to compute the normals for each of the loaded CQUAD4 elements. Using FORCE1 and FORCE2 entries for the shear and normal loads, respectively, makes the task much easier.

The 10 N normal force for each element is divided equally between the two grid points along the edge. The directions of the resulting 5 N forces are applied using the FORCE2 entry by taking the cross product of the vectors defined by the edges to which the grid points are attached. The result is a 5 N load acting normal to the plate. The 8 N shear force for each element is also divided up equally between the two grid points along the edge. The resulting 4 N loads are applied with the FORCE1 entry. The directions of these loads are determined by the line connecting the edge grid points.

The input file is shown in Listing 8-4. Note that the normal forces and the shear forces are applied as one load condition. If you want the forces to act as separate loadings, you can use different IDs for the FORCE1 and FORCE2 entries and call them out using two separate subcases in your input file.

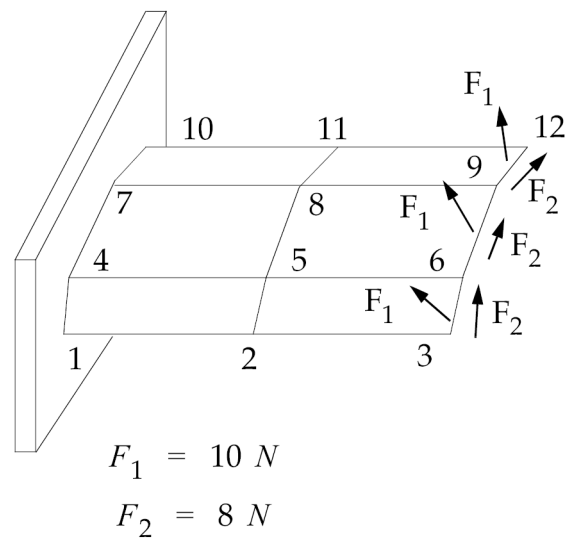


Figure 8-2. Edge Forces on a Plate Element

```
$ FILENAME - EDGE.DAT
$
LOAD=ALL
LOAD=3
.
.
.
BEGIN BULK
$
$ UNITS ARE CM AND N
$
FORCE1 3      3      4.      3      6
FORCE1 3      6      4.      3      6
FORCE1 3      6      4.      6      9
FORCE1 3      9      4.      6      9
FORCE1 3      9      4.      9      12
FORCE1 3      12     4.      9      12
FORCE2 3      3      5.      3      6      3      2
FORCE2 3      6      5.      3      6      3      2
FORCE2 3      6      5.      6      9      6      5
FORCE2 3      9      5.      6      9      6      5
FORCE2 3      9      5.      9      12     9      8
FORCE2 3      12     5.      9      12     9      8
```

```

$
GRID 1 0.0 -60. -2.
GRID 2 60. -60. -2.
GRID 3 120. -60. -2.
GRID 4 0.0 -30. 0.0
GRID 5 60. -30. 0.0
GRID 6 120. -30. 0.0
GRID 7 0.0 30. 0.0
GRID 8 60. 30. 0.0
GRID 9 120. 30. 0.0
GRID 10 0.0 60. -2.
GRID 11 60. 60. -2.
GRID 12 120. 60. -2.
$
PSHELL 1 4 .5 4
MAT1 4 7.1E6 .3
$
CQUAD4 1 1 1 2 5 4
CQUAD4 2 1 2 5 6 3
CQUAD4 3 1 4 5 8 7
CQUAD4 4 1 5 6 9 8
CQUAD4 5 1 7 8 11 10
CQUAD4 6 1 8 9 12 11
$
SPC1 1 123456 1 4 7 10
$
ENDDATA

```

Listing 8-4. Edge Loads

Note that the two loads are applied at the same grid points. This is perfectly acceptable. The total force acting at the grid points is the sum of all the applied loads. The OLOAD (applied load output request) output, which you request through the Case Control section, shows the applied force resultant acting at a set of user specified grid points. The OLOAD output is useful in verifying that the loads are applied correctly. A partial listing of the output file showing the OLOAD output is shown in [Figure 8-3](#).

LOAD VECTOR							
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
3	G	0.0	3.658545E+00	5.255002E+00	0.0	0.0	0.0
6	G	0.0	7.658545E+00	1.025500E+01	0.0	0.0	0.0
9	G	0.0	8.323735E+00	9.722850E+00	0.0	0.0	0.0
12	G	0.0	4.323736E+00	4.722850E+00	0.0	0.0	0.0

Figure 8-3. OLOAD Output

See also

- “**FORCE1**” in the *NX Nastran Quick Reference Guide*
- “**FORCE2**” in the *NX Nastran Quick Reference Guide*

Using **MOMENT**, **MOMENT1**, **MOMENT2**

MOMENT, MOMENT1, and MOMENT2 are similar to the three FORCE entries. The only difference is that you use force entries to apply concentrated force in the translational degrees of freedom (T1, T2, and T3), while you use moment entries apply concentrated moments in the rotational degrees of freedom (R1, R2, and R3).

- With MOMENT, you define a static, concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction
- With MOMENT1, the software uses two grid points to determine direction ($\vec{m} = M\vec{n}$, where \vec{n} is a unit vector parallel to the vector from grid 1 to grid 2).
- With MOMENT2, the software uses four grid points to determine direction ($\vec{m} = M\vec{n}$, where \vec{n} is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4).

The application of concentrated moments is analogous to forces. MOMENT has the following format:

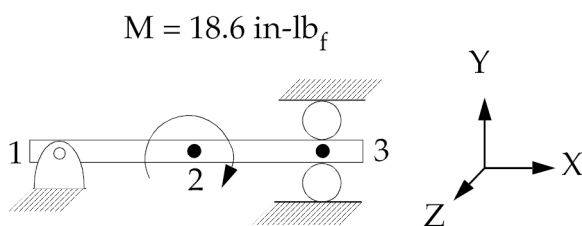
1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Field

Contents

SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni ≠ 0.0)

Consider moment M acting about the basic Z axis of the simply supported beam shown below:



In this case, the MOMENT entry could be written as

1	2	3	4	5	6	7	8	9	10
MOMENT	6	2		-18.6	0.0	0.0	1.		

or, in free field format,

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

The applied moment \vec{m} is given by

$$\vec{m} = M\vec{N}$$

Equation 8-2.

where \vec{N} is the vector (N1, N2, N3). As was the case with FORCE entry, M is the full magnitude of the moment since \vec{N} is a vector of unit length. The direction of the applied moment is given by the sign of M according to the right-hand rule because $\vec{N} = (0., 0., 1.)$ is a vector in the direction of the positive Z axis direction. Note that specifying M = 18.6 and $\vec{N} = (0., 0., -1.)$ would produce an equivalent result.

See also

- “**MOMENT**” in the *NX Nastran Quick Reference Guide*
- “**MOMENT1**” in the *NX Nastran Quick Reference Guide*
- “**MOMENT2**” in the *NX Nastran Quick Reference Guide*

Using FORCEAX, MOMAX

You can use the FORCEAX and MOMAX entries to define the loading of specified harmonics on rings of conical shell elements. You can use FORCE and MOMENT entries to apply concentrated loads or moments to conical shell elements if you defined such points with a POINTAX entry.

See also

- “**FORCEAX**” in the *NX Nastran Quick Reference Guide*
- “**MOMAX**” in the *NX Nastran Quick Reference Guide*

Using SLOAD

You can use the SLOAD entry to define a concentrated static load at a scalar point or a grid point. With SLOAD, you can only specify the magnitude since only one component of motion exists at a scalar point.

See also

- “**SLOAD**” in the *NX Nastran Quick Reference Guide*

8.5 Distributed Loads

There are three Bulk Data entries available to apply distributed loads to element surfaces.

- Use PLOAD define a uniform static pressure load on a triangular or quadrilateral surface.
- Use PLOAD1 and PLOAD2 to apply a distributed load to the CQUAD4, CTRIA3, and CSHEAR elements only.
- Use PLOAD4 to apply distributed loads on any of the two-dimensional elements and on the surfaces of three-dimensional elements.

PLOAD

You can use the PLOAD entry to define a uniform normal static pressure load on triangular or quadrilateral surfaces by using grid points. You can apply PLOAD to 2-D (surface) or 3-D (solid) elements.

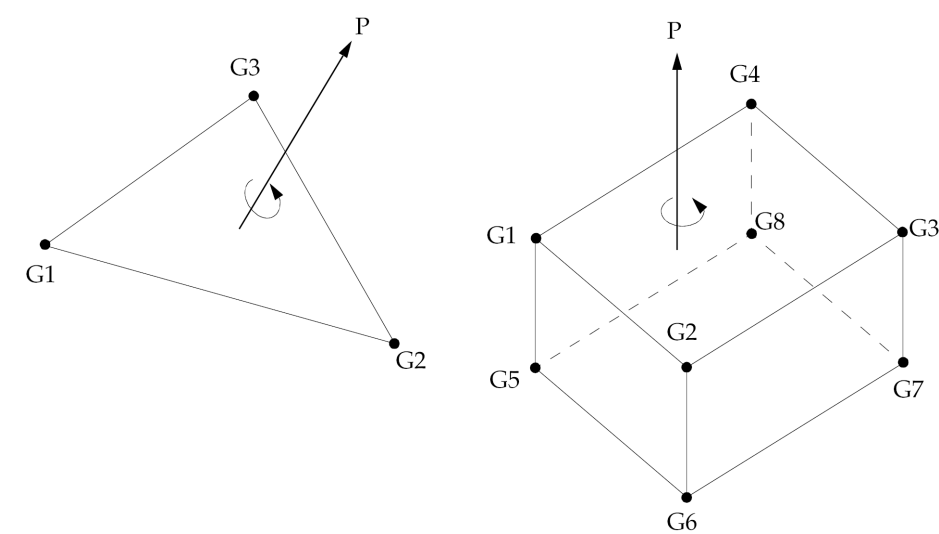
The PLOAD entry is different from the other PLOADi entries because it references three or four grid points rather than the element itself. When you use PLOAD to load a CTRIA3 element, the total force acting on the element is divided evenly among the three corner points. When you use PLOAD to load a CQUAD4 element, the total force acting on the element is distributed to corner grid points using the geometric shape of the element. The resultant of the applied corner loads acts through the centroid of the element. The direction of the pressure load is determined using the right-hand rule by the numbering sequence of the grid points on the PLOAD entry.

The PLOAD entry has the following form:

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

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure. (Real)
Gi	Grid point identification numbers. (Integer > 0; G4 may be zero or blank)

Grid Points G1, G2, G3, and G4 define either a triangular or quadrilateral surface; if G4 is zero or blank, the software assumes that the surface is triangular. The direction of the pressure load is determined by applying the right-hand rule to the grid point ordering sequence of the surface.



Pressure is applied in the opposite direction by making the value of P negative.

See also

- “PLOAD” in the *NX Nastran Quick Reference Guide*

PLOAD1

You can use the PLOAD1 entry to apply a distributed load to a CBAR, CBEAM, or CBEND element. You can use the PLOAD1 entry for both concentrated and linearly distributed forces.

- For the CBAR and CBEAM elements, you can apply the linearly distributed force between any two locations on the element (or off the element if you wish.)
- For the CBEND element, you can only apply the linearly distributed loads to linearly varying forces and moments between the end points.

You can use PLOAD1 entry to define either distributed or concentrated loads at intermediate points on CBAR and CBEAM elements. You define applied loads at the end points of CBAR and CBEAM elements with either FORCE or MOMENT entries. You can also use PLOAD1 to define only distributed loads for the CBEND element. For the CBEND element, you can only use PLOAD1 to define

You can use the CBARAO entry is used to define intermediate points on CBAR elements where you want to obtain stress and/or force output. You can also use the PLOAD1 to define only distributed loads for the CBEND element. The distributed load is always applied along the entire length of the element and does not define loads at intermediate points on the CBEND. The PLOAD1 entry cannot be used to define concentrated loads at intermediate stations on the CBEND element.

You can apply the load can be applied along the entire element length, a segment of the length, or at a point along the length. The form of the PLOAD1 entry is shown below. The meanings of X1, X2, P1, and P2 are shown in [Figure 8-4](#).

Format

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

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number.(Integer > 0)
TYPE	Load type. (Character: "FX", "FY", "FZ", "FXE", "FYE", "FZE", "MX", "MY", "MZ", "MXE", "MYE", "MZE")
SCALE	Determines scale factor for X1, X2. (Character: "LE", "FR", "LEPR", "FRPR")
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank, $0 \leq X1 \leq X2$)
P1, P2	Load factors at positions X1, X2. (Real or blank)

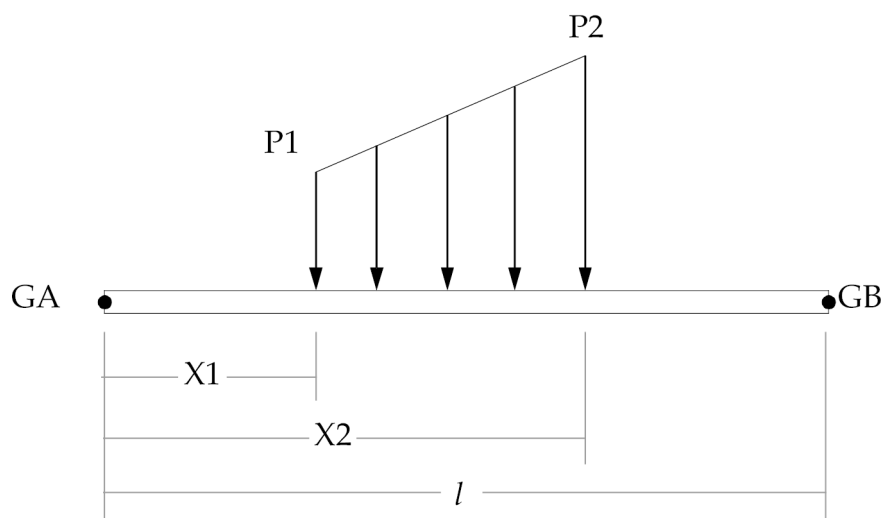


Figure 8-4. PLOAD1 Convention in CBAR or CBEAM Elements

The TYPE option in field 4 lets you specify the type of the load. The load can be a concentrated force or moment in the basic or element coordinate system. If the applied load is to be a concentrated load, leave fields 8 and 9 blank and the concentrated load will be applied at the X1 location. If both X1 and X2 are input, the load will be taken as a linearly varying load between X1 and X2.

The SCALE option in field 5 lets you specify the location of the load on the CBAR and CBEAM elements using field 5 of the PLOAD1 entry.

- With the “LE” or “LEPR” methods, you specify the actual start and end positions of the load as measured from end A of the element. When using this method, the distances X1 and X2, as shown in [Figure 8-5](#), are in the same units as the dimensions used for the model.
- With the “FR” or “FRPR” methods, you specify the percent (using “X/XB”) along the element where the load starts and ends. A value of 0.0 defines end A, while a value of 1.0 defines end B. Again, the start of the load is measured from end A.

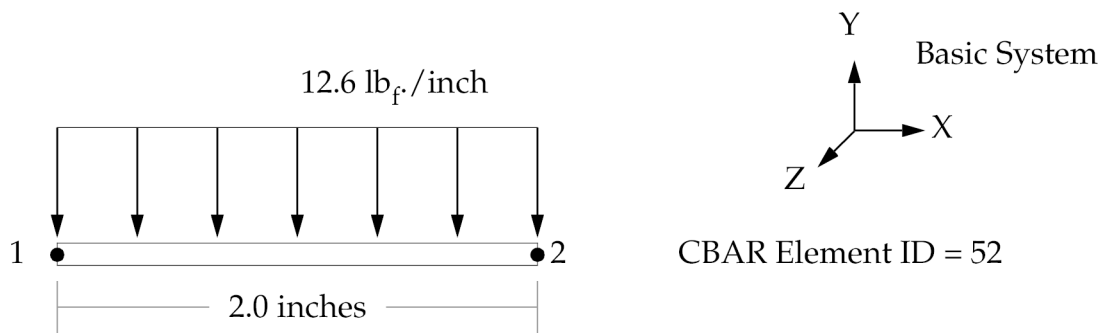
For both methods of describing the location of the loads, you can also specify whether the applied load is to be a direct load (scale “LE” and “FR”) or a projected load (scale “LEPR” or “FRPR”). For the projected loads, distributed loads are entered in terms of the projected length of the element as shown in [Figure 8-5](#). Remember that if you input the loads in terms of the basic coordinate system (“FX”, “FY”, “FZ”, “MX”, “MY”, or “MZ”), then the projected angle α is with respect to the basic coordinate system, not the element coordinate system.

PLOAD1Examples

This section contains a series of examples that illustrate different uses of the PLOAD1 entry.

PLOAD1 Example 1

In this example, we used PLOAD1 to apply a uniformly distributed load over the full length of a CBAR element using fractional (normalized) scaling.



Note that

P1	=	P2 = 12.6 lb _f /inch
X1	=	0.0 fractional scaling
X2	=	0.0

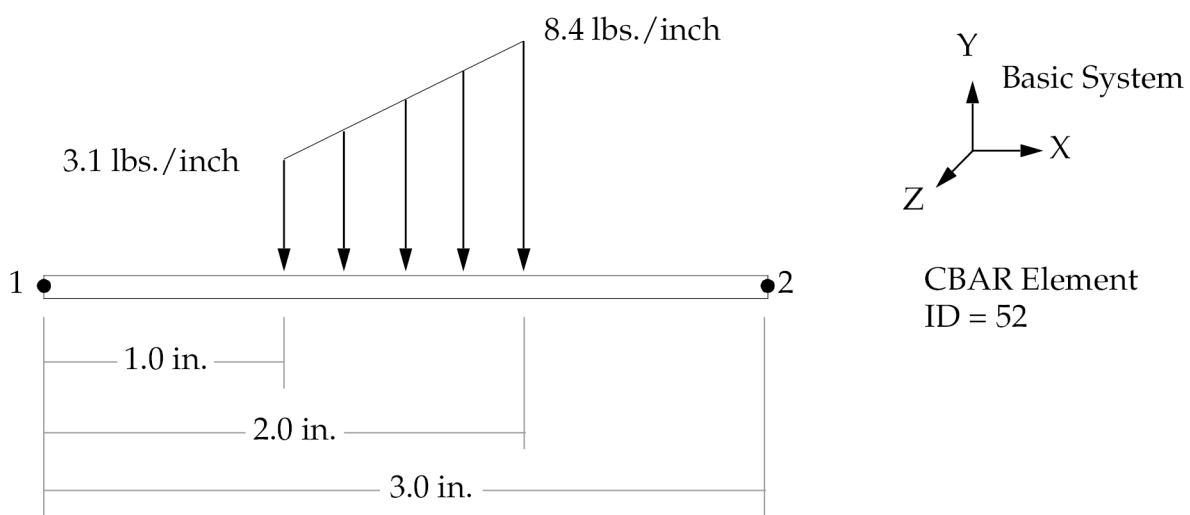
1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	FR	0.0	-12.6	1.0	-12.6	

Note: You can also use PLOAD1 to apply a uniformly distributed load over the full length of a CBAR element using length scaling.

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

PLOAD1 Example 2

In this example, we used PLOAD1 to apply a linearly varying distributed load to the interior of a CBAR element using length scaling:



1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	LE	1.0	-3.1	2.0	-8.4	

PLOAD Example 3

The example demonstrating the use of the PLOAD1 entry consists of applying a direct linearly varying load to the three-bar structure shown in [Figure 8-5](#).

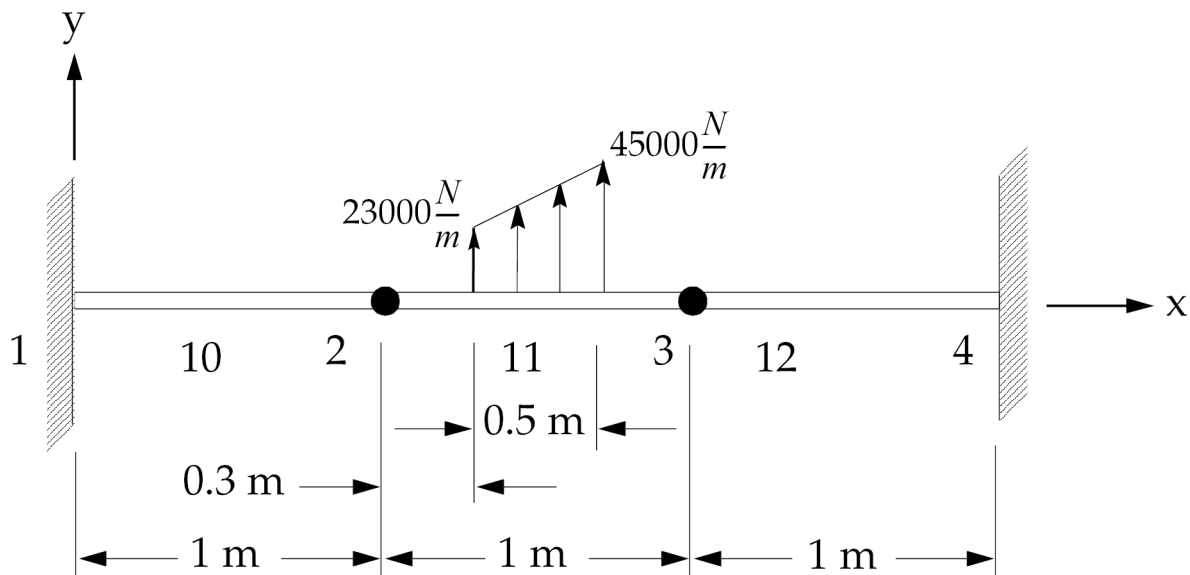


Figure 8-5. Distributed Load on CBAR Elements

The input file for this example is shown in [Listing 8-5](#).

```

$
$  FILENAME - DISTRIB.DAT
$
ID LINEAR,DISTRIB
SOL 101
TIME 2
CEND
LOAD = 1
FORCE = ALL
BEGIN BULK
$
$  UNITS ARE MM AND N
$
GRID    1          0.      0.0    0.0          123456
GRID    2        1000.    0.0    0.0          345
GRID    3        2000.    0.0    0.0          345
GRID    4        3000.    0.0    0.0          123456
$
CBAR    10         1      1      2      1.0     0.0     1.0
CBAR    11         1      2      3      1.0     0.0     1.0
CBAR    12         1      3      4      1.0     0.0     1.0
CBARAO  11         FR    19     .05     .05
$

```

```

PBAR    1      1      100.0  100.0  100.0  100.0
$
MAT1    1      4.E4      .3
$
PLOAD1  1      11      FY      LE      300.   23.   800.   45.
$
ENDDATA

```

Listing 8-5. PLOAD1 Loading

Note that the starting location for the linearly varying load starts and ends at 0.3 m (300 mm) and 0.8 m (800 mm), respectively. This is the distance measured from end A. One of the most common mistakes is to make these distances 1.3 m and 1.8 m, respectively, while thinking that it is the actual distance measured in the basic coordinate system. It is not; it is the distance measured from end A of the loaded element—in this case, grid point 2.

When you use the PLOAD1 entry to define a distributed load, the CBARAO (CBAR Additional Output) entry is helpful. You use the CBARAO entry to request stress and force output at intermediate locations along the CBAR element. The CBARAO output for element 2 is shown in [Figure 8-6](#).

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

Figure 8-6. CBARAO Output for the Center Element

The CBARAO entry is one of the best tools available for model checkout when you're using linearly varying loads on the CBAR elements. The CBARAO output lets you easily generate a shear and moment diagram as shown in [Figure 8-7](#) below.

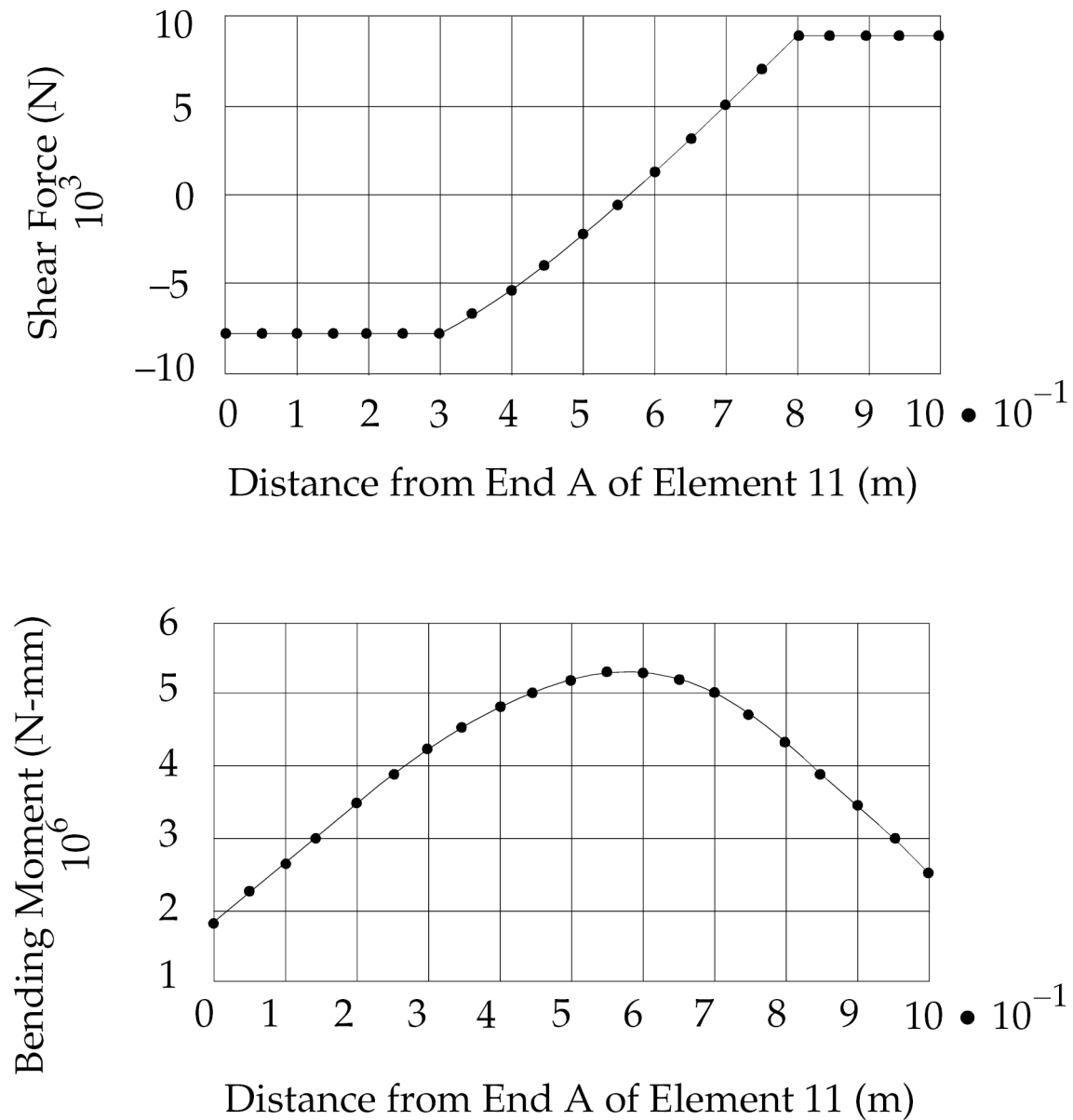
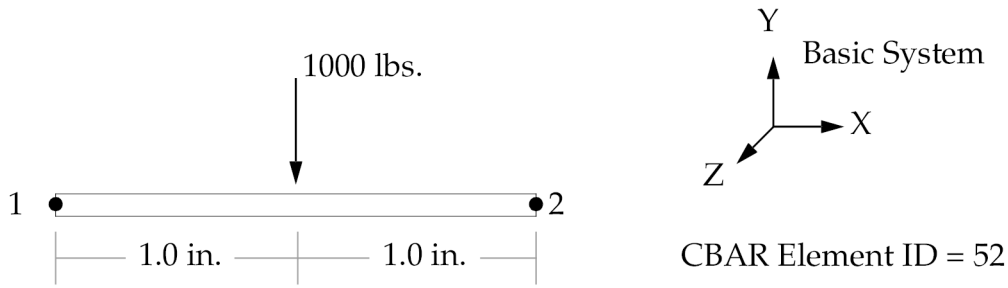


Figure 8-7. Shear and Moment Diagram Generated from CBARAO Output

For CBEAM elements, you can use the PBEAM entry to directly request intermediate output along the element.

PLOAD1 Example 4

You can also use PLOAD1 to apply a concentrated load at an interior point of a CBAR element using fractional scaling.



1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
PLOAD1	36	52	FY	FR	0.5	-1000.			

PLOAD1 Example 5

Another feature of the PLOAD1 entry is the ability to apply projected loads. To apply a load as a projected load, you choose a scale (field 5) of “LEPR” for the actual length from end A or “FRPR” for the fractional distance from end A. This example shows illustrates how you can use PLOAD1 to define a projected load using a snow load on a truss structure as shown in [Figure 8-8](#).

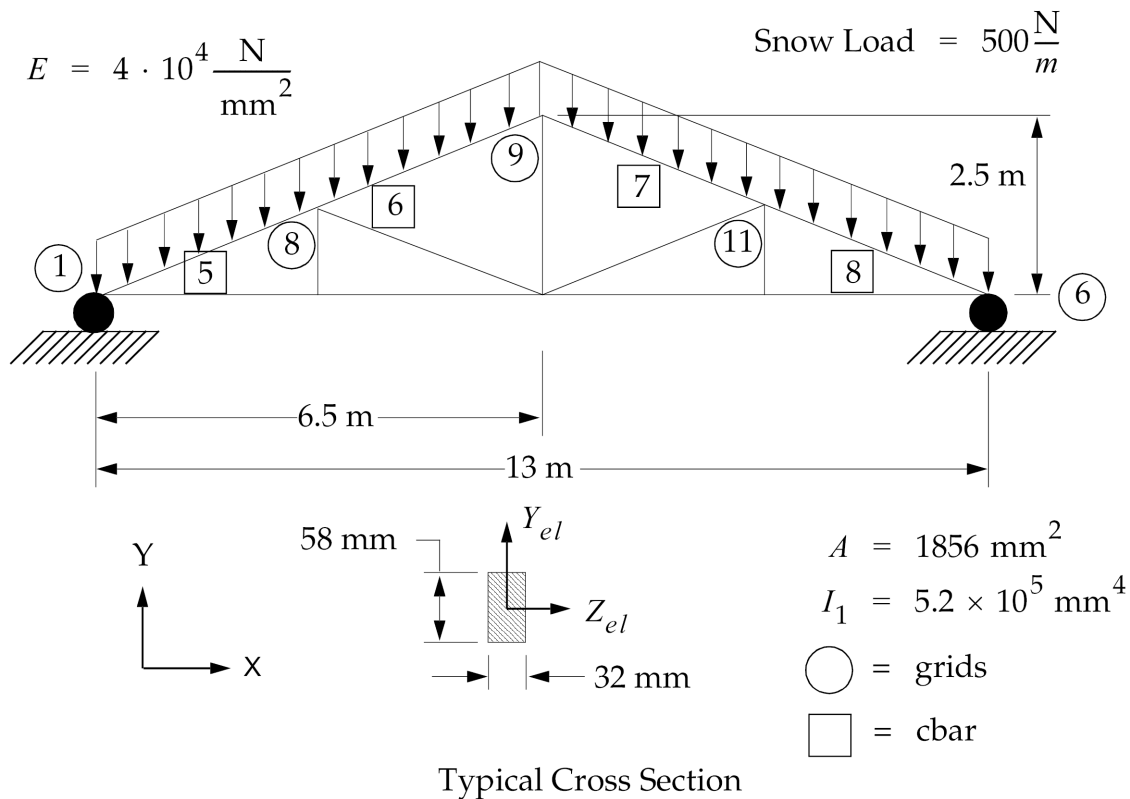


Figure 8-8. Planar Truss Structure with a Snow Loading

The snow load on the truss is 500 N/m acting in the basic Y-direction. The input file containing the grid points and PLOAD1 entries is shown in [Listing 8-6](#).

```

$
$ FILENAME - PLOAD1.DAT
$
$ UNITS ARE MM AND N
$
GRID      1      0.0      0.0      0.0      345
GRID      6     13000.    0.0      0.0      345
GRID      8     3250.    1250.    0.0      345
GRID      9     6500.    2500.    0.0      345
GRID     11     9750.    1250.    0.0      345
$
CBAR      5      1      1      8      1.0      1.      0.0
CBAR      6      1      8      9      1.0      1.      0.0
CBAR      7      1      9     11      1.0      1.      0.0
CBAR      8      1     11      6      1.0      1.      0.0
$
PLOAD1    1      5      FY      FRPR    0.0     -.5      1.     -.5
PLOAD1    1      6      FY      FRPR    0.0     -.5      1.     -.5
PLOAD1    1      7      FY      FRPR    0.0     -.5      1.     -.5
PLOAD1    1      8      FY      FRPR    0.0     -.5      1.     -.5

```

Listing 8-6. Truss with Snow Loading

To see the effect of using projected forces, the resulting SPC forces are shown in [Figure 8-9](#). The total reaction of the loads obtained by adding the SPC forces is 6500 N in the Y-direction. The total reaction load is equal to the projected length of 13 meters times the distributed load of 500 N/m. If the projected option had not been used, the applied load would have been 6964.2 N, which is the total length of the top members times the distributed load.

FORCES OF SINGLE-POINT CONSTRAINT								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	-2.511155E-05	3.250000E+03	.0	.0	.0	.0	
6	G	.0	3.250000E+03	.0	.0	.0	.0	

Figure 8-9. SPC Forces Due to the Snow Loading

Note: With PLOAD1, if you're using a concentrated load and don't need the additional features that the CBEAM element offers, you should use the CBAR element because of the way the equivalent end loads are generated.

- For the CBAR element, the equivalent end loads are generated explicitly.
- For the CBEAM element, the end loads are generated by numerical integration along the length of the element, which may not be as accurate as the method used for the CBAR element.

A linearly varying load applied with a PLOAD1 entry is accurate when used with either element.

PLOAD2

You can use the PLOAD2 entry to apply a normal uniform pressure load to CQUAD4, CSHEAR, or CTRIA3 2-D (surface) elements using element IDs. The positive direction of the loading is determined by the order of the grid points on the element connection entry, using the right hand rule. The software computes the magnitude and direction of the load from the value of the pressure and the coordinates of the connected grid points. NX Nastran applies the load to the connected grid points.

The PLOAD2 entry has two forms:

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

Alternate form:

PLOAD2	SID	P	EID1	"THRU"	EID2				
--------	-----	---	------	--------	------	--	--	--	--

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

The direction of the pressure is determined using the connected GRID points in the same right-hand rule sense as the PLOAD entry (i.e., with respect to the positive element z axis). In addition, you use the "THRU" option, all referenced elements must actually exist.

The PLOAD2 entry is similar to the PLOAD entry except that the PLOAD2 references the element ID instead of the grid points where the element is attached. The PLOAD2 is usually preferred over the PLOAD entry because it is easier to use. The PLOAD2 entry, similar to the PLOAD entry, is limited to pressure acting normal to the element surface. The direction of the pressure load is determined by the numbering sequence of the grid points on the connectivity entry (CQUAD4, CTRIA3, etc.).

See also

- "PLOAD2" in the *NX Nastran Quick Reference Guide*

PLOAD4

The PLOAD4 entry lets you create the most general pressure definition. You can use PLOAD4 to apply pressures and/or tractions to any of the two-dimensional elements and the surfaces of the three-dimensional elements. With PLOAD4, you can create a pressure load that's either normal to the surface or that contains a traction (not normal to the surface) component. In addition, you can define a different value of pressure at each corner.

You can use PLOAD4 with CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, and CTRIAR, CHEXA, CPENTA, CPYRAM and CTETRA elements. Since the surface hyperelastic elements CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6 are plane strain elements, you can't apply pressure loads on them.

The software automatically computes the magnitude and direction of the equivalent grid point forces using the isoparametric shape functions of the element to which the load has been applied.

Load intensities P1, P2, P3, (and P4) act at corner points G1, G2, G3, (and G4) for triangular (and quadrilateral) elements. The default direction of positive pressure for faces of solid elements is inward.

You can set the direction of the pressure can be set by one of two methods.

- By default, the software uses the element normal to determine the direction of the pressure. For two-dimensional elements, the direction of positive pressure is in the direction of the outward normal as is determined by applying the right-hand rule to the ordering sequence of the grid points on the connectivity entry. For surfaces of solid elements, the direction of positive pressure is inward toward the center of the element. The face of the solid to which you are applying the pressure is determined by specifying the appropriate corner grid points.
- Alternatively, you can input the direction of the pressure defined by the PLOAD4 entry using an optional coordinate system and a vector entered on the continuation line. Using a local coordinate system, you can define a pressure acting at any angle to the surface. You can also apply loads acting parallel to the surface (tractions).

Another feature unique to the PLOAD4 entry is the ability to apply a nonuniform pressure. You can enter the pressure at each of the corner grid points to create a linearly varying pressure load. If you enter the pressure for the first grid point only, the software assumes that the pressure is constant over the element.

The format of the PLOAD4 entry is:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

The alternate format is

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID1	P1	P2	P3	P4	"THRU"	EID2	
	CID	N1	N2	N3					

Field	Contents
SID	Load set identification number.
EID	
EID1	Element identification number.
EID2	
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element.
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only.

Field**Contents**

	For CHEXA, CPYRAM, or CPENTA quadrilateral faces, G3 is the identification number of a grid point connected to a corner diagonally opposite to G1. Required for quadrilateral faces of CHEXA, CPYRAM and CPENTA elements only.
G3	For CPYRAM element triangle faces, G1 and G3 are adjacent corner nodes on the quadrilateral face, and the load is applied on the triangular face which includes those grids. For CPENTA element triangle faces, G3 must be omitted.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only.
CID	Coordinate system identification number.
N1, N2, N3	Components of the vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity.

If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1; P4 is left blank for a triangular face. In addition, for pressure that acts normal to the face, the continuation entry is not used.

See also

- “**PLOAD4**” in the *NX Nastran Quick Reference Guide*

PLOAD4 Example 1

Consider the curved plate shown in **Figure 8-10**. PLOAD4 entries are to be used to apply a normal pressure to each of the six CQUAD4 elements.

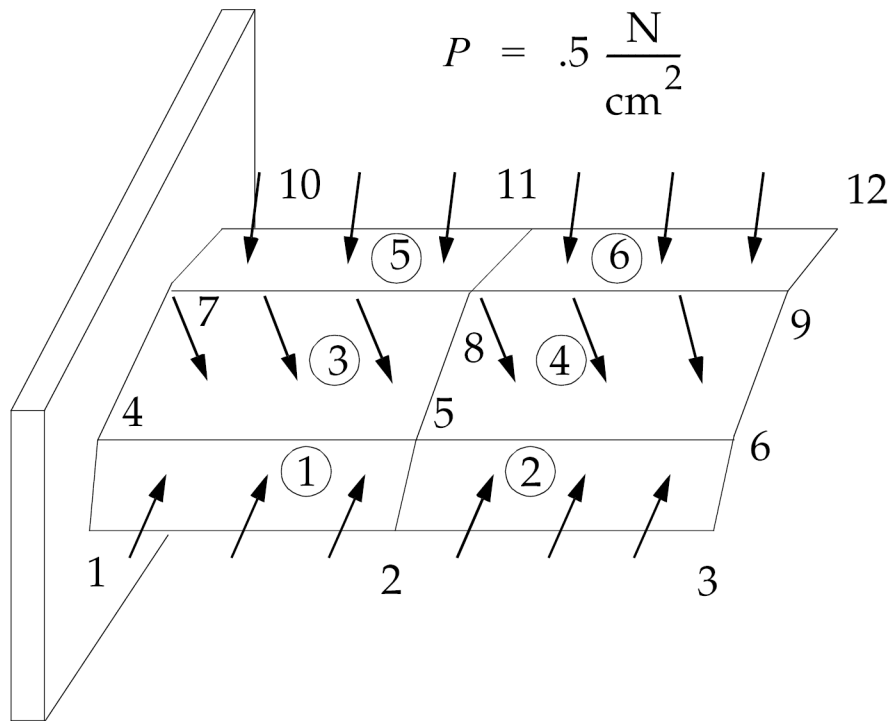


Figure 8-10. Pressure Loads on the CQUAD4s

This input file is given in [Listing 8-7](#). When using the PLOAD4 entry, you may specify a PLOAD4 entry for every element or use the alternate method of specifying several elements that have the same pressure. The alternate method is used for this example.

```

$ FILENAME - PRESS.DAT
$ UNITS ARE CM, N
PLOAD4 3 1 -.5 THRU 6
$
GRID 1 0.0 -60. -2.
GRID 2 60. -60. -2.
GRID 3 120. -60. -2.
GRID 4 0.0 -30. 0.0
GRID 5 60. -30. 0.0
GRID 6 120. -30. 0.0
GRID 7 0.0 30. 0.0
GRID 8 60. 30. 0.0
GRID 9 120. 30. 0.0
GRID 10 0.0 60. -2.
GRID 11 60. 60. -2.
GRID 12 120. 60. -2.
$
CQUAD4 1 1 1 2 5 4
CQUAD4 2 1 2 5 6 3
CQUAD4 3 1 4 5 8 7
CQUAD4 4 1 5 6 9 8
CQUAD4 5 1 7 8 11 10
CQUAD4 6 1 8 9 12 11
$
SPC1 1 123456 1 4 7 10
$
PSHELL 1 4 .5 4
MAT1 4 7.1E6 .3

```

Listing 8-7. CQUAD4 Elements with Pressure Loads

The resulting deflection is shown in [Figure 8-11](#). The deflected shape is not what you would expect. Element 2 appears to bend up. The problem occurs because the normal for element 2 is reversed (see the ordering of the grid point IDs on the CQUAD4s). The pressure on this element is acting upward. This type of mistake is quite common.

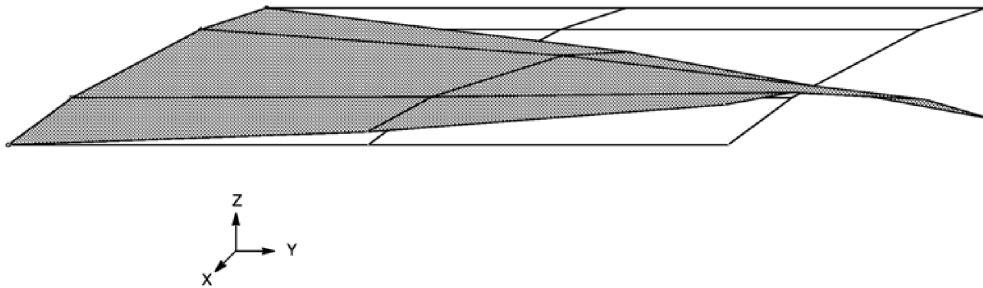
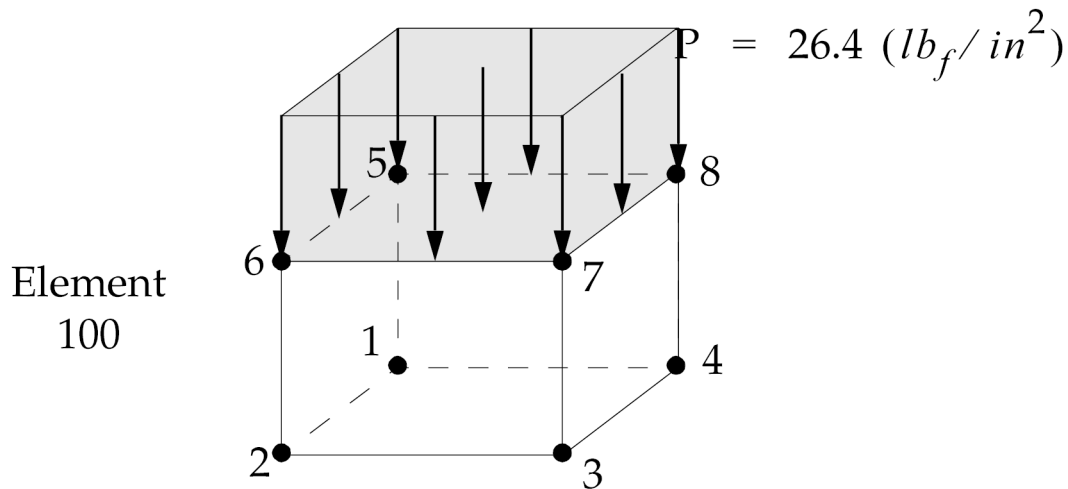


Figure 8-11. Displacements Due to Pressure Load

PLOAD4 Example 2

Specify the PLOAD4 entry for a uniform normal pressure load applied to the CHEXA solid element shown in the following figure:



1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

PLOAD4	12	100	26.4				5	7	
--------	----	-----	------	--	--	--	---	---	--

This load is selected in the Case Control Section with the command `LOAD = 12`. Leaving P2, P3, and P4 blank assigns a uniform pressure value of 26.4 lb/in².

PRESAX, PLOADE1, and PLOADX1

- You can use the `PRESAX` entry to define pressure loading on the conical shell element.
- You can use the `PLOADE1` entry to define an in-plane distributed loading on the edge of `CPLSTN3`, `CPLSTN4`, `CPLSTN6`, and `CPLSTN8` plane strain elements and `CPLSTS3`, `CPLSTS4`, `CPLSTS6`, and `CPLSTS8` plane stress elements.
- You can use the `PLOADX1` entry to define pressure loading on the `CTRIAX6` element and on the hyperelastic `CTRIAX` and `CQUADX` elements.

See also

- “PRESAX”** in the *NX Nastran Quick Reference Guide*.
- “PLOADE1”** in the *NX Nastran Quick Reference Guide*.
- “PLOADX1”** in the *NX Nastran Quick Reference Guide*.

8.6 Inertia Loads (Acceleration Loads)

This section describes how to apply “static” acceleration loads to your model, such as gravity (the response of a structure to its own weight) and vehicle maneuver loads. The acceleration causes a

static load and not a dynamic one - the structural response is steady state (static) with the transient part being zero. With an acceleration load, you must specify both the magnitude of the acceleration and the direction in which it acts.

Using GRAV

You can use the GRAV entry to define the direction and magnitude of a gravity vector in any user-defined coordinate system. The software multiplies components of the gravity vector by the mass matrix to obtain the components of the gravity force at each grid point. Since the mass matrix is used to compute the forces, you must have mass in your model. Typically, you would define that mass by density you specify on a material bulk data entry. The software doesn't calculate the gravitational acceleration at scalar points. You must introduce gravity loads at scalar points directly.

Gravity loading is one the best loadings to use when you're performing model checkout because the software applies the force at every point in the model that has mass. If you have modeling errors where elements or grids points aren't attached to the structure, their displacement is unbounded, and the job fails. If this occurs, the software prints a diagnostic message indicating the grid point and component, where the singularity is detected.

The format of the GRAV Bulk Data entry is:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
MB	Used only in superelement analysis.

The direction and magnitude of acceleration are given by \vec{a} where the vector $\vec{N} = (N1, N2, N3)$ gives the direction. The magnitude of \vec{a} is equal to A times the magnitude of \vec{N} . Note, for example, that entering the value of A in in/sec² indicates that other specifications of the model that involve length units (element length, moments of inertia, modulus of elasticity, etc.) must also be in inches to preserve unit consistency.

For models that have only GRAV loads, the load set ID (SID) is selected in the Case Control Section with a LOAD = SID Case Control command. For models that also contain other types of loads, such as FORCE or PLOAD entries, you must use the LOAD entry to combine them with the GRAV entry.

Finally, the model must include mass density information to use acceleration loads. You enter mass density on a material property bulk data entry. For example, for a model of typical structural steel using English units, the mass density might be

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

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

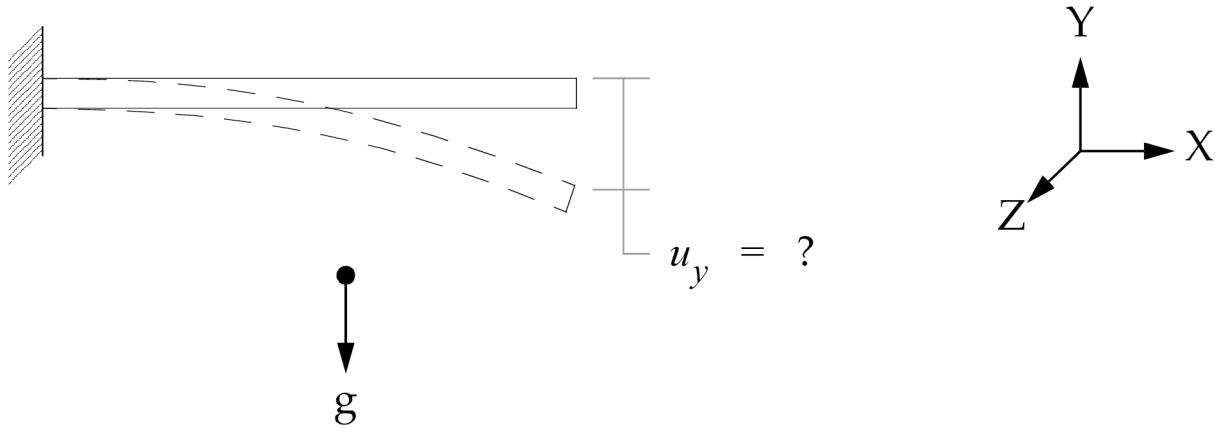
1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	12	30.E6		0.3	7.0E-4				

See also

- “Combining Loads” in the *NX Nastran User's Guide*

GRAV Example 1

This example shows how to calculate the tip deflection of the cantilever beam due to its own weight.



First, notice that the force of gravity (g) acts in the $-Y$ direction (the direction of the center of the earth). Thus, the vector \vec{N} can be written as:

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

Equation 8-3.

The acceleration due to gravity on the Earth's surface is approximately 32.2 ft/sec² (386.4 in/sec²) or 9.8 m/sec². Units must be consistent throughout the model (this example uses inches).

Thus, the GRAV entry is as follows:

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

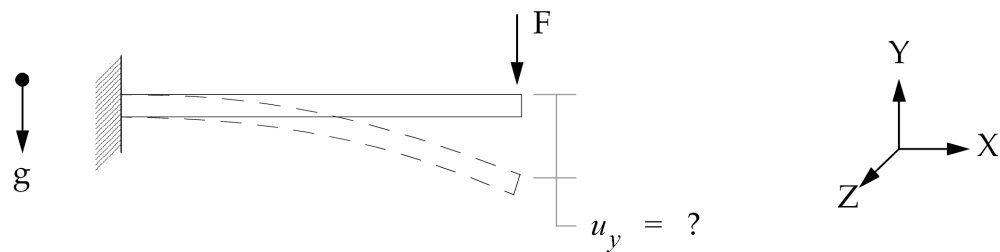
or, in free field format,

GRAV,15,,386.4,0.,-1.,0.

The Case Control command required to apply this load is `LOAD = 15`.

GRAV Example 2

Next, we want to calculate the tip deflection of the cantilever beam due to its own weight and a concentrated force. Assume that a concentrated force is added to the beam as shown:



To combine gravity loading with the concentrated force, the following approach must be used:

In Case Control: `LOAD = 15`

LOAD	15	1.0	1.0	12	1.0	13			
------	----	-----	-----	----	-----	----	--	--	--

and

In Bulk Data:

GRAV	12
FORCE	13

Importantly, the set ID on a GRAV entry can't be the same as the Set ID on any other load entry.

GRAV Example 3

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

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

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

0.3 g in the x-direction ($0.3\text{ g} = 0.3 \cdot 9.8\text{ m/sec}^2 = 2.9\text{ m/sec}^2$)

Three separate GRAV entries can be written as:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		
GRAV	15		12.7	1.	0.	0.			
GRAV	15		27.4	0.	1.	0.			
GRAV	15		2.9	0.	0.	1.			

Using RFORCE

You can use the RFORCE entry to define a static loading condition due to angular velocity and/or acceleration. These loads are specified by the designation of a grid point that lies on the axis of rotation and by the components of rotational velocity or angular acceleration in any defined coordinate system.

There are two methods to compute these loads.

- In the first method, the mass matrix is regarded as pertaining to a set of distinct rigid bodies connected to grid points. Deviations from this viewpoint, such as the use of scalar points or the use of mass coupling between grid points, can result in errors.
- The second method allows mass coupling, but does not allow offsets. The second method is the only method used for angular acceleration.

On the RFORCE entry, you input the components of a spin vector that are used internally to compute centrifugal forces. Each component of the spin vector is multiplied by the same scale factor. The format of the RFORCE entry is as follows:

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

Field	Contents
SID	Load set identification number.
G	Grid point identification number through which the rotation vector acts.
CID	Cartesian coordinate system defining the components of the rotation vector.
A	Scale factor of the angular velocity in revolutions per unit time.
R1, R2, R3	Rectangular components of rotation vector \vec{R} . The vector defined passes through point G.
METHOD	Method used to compute centrifugal forces due to angular velocity.
RACC	Scale factor of the angular acceleration in revolutions per unit time squared.

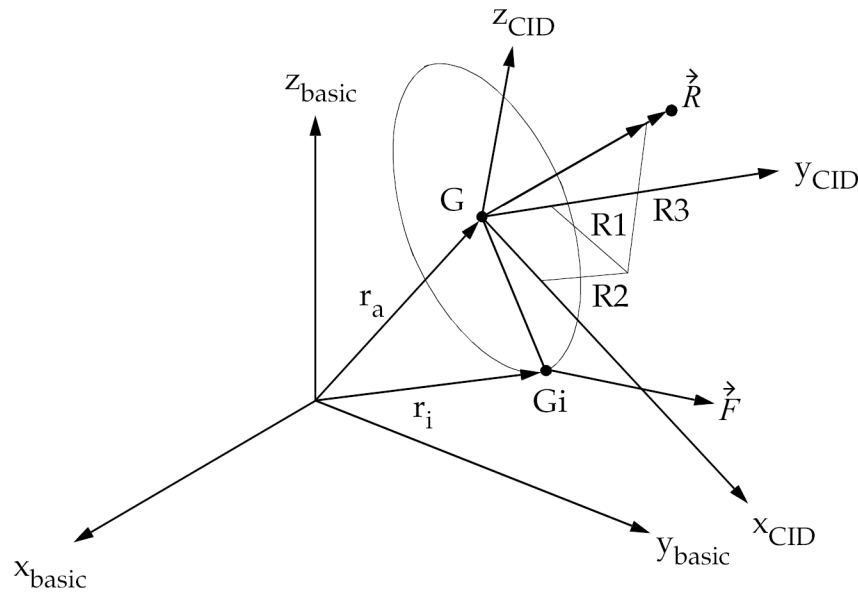


Figure 8-12. RFORCE Vector at Grid Point Gi

$$\{\vec{F}\}_i = [m]_j[\vec{\omega}_X(\vec{\omega}_X(\vec{r}_i - \vec{r}_a)) + \vec{\alpha}_X(\vec{r}_i - \vec{r}_a)]$$

where:

angular velocity = $\vec{\omega} = 2\pi A \cdot \vec{R}$ (radians/unit time)

angular acceleration = $\dot{\vec{a}} = 2\pi \text{RACC} \cdot \dot{\vec{R}}$ (radians/unit time squared)

$[m]_i$ = 3 × 3 translational mass matrix at grid point Gi

Note

The equation for \vec{F}_i has additional terms if the mass is offset and METHOD = 1 is selected.)

You must select one of two methods for the internal calculation of the loading vector (field 9).

- Method=1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS).
- Method=2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with method=2.

All possible combinations of mass matrices and offset and the correct method to be used are shown in [Table 8-1](#).

Table 8-1. Restrictions When Using the RFORCE Entry

	No Offset	Offset
Lumped	Method=1 or Method=2	Method=1
Coupled	Method=2	Neither

In addition, for problems with elements that have edge grid points (CQUAD8, CTRIA6, CTRIAX6, CHEXA, CPENTA, CPYRAM and CTETRA), the software only produces correct centrifugal loads if you include the parameter PARAM,COUPMASS,x (where x is greater than 1) in the input file and you select Method 2.

See also

- “**RFORCE**” in the *NX Nastran Quick Reference Guide*
- “COUPMASS” in the *NX Nastran Quick Reference Guide*

Defining Mass in Your Model

There are several different ways that you can define mass in your model in NX Nastran:

- The concentrated mass elements and scalar points (CMASSi and CONMi)
- The mass density option on the material entries (MAT1, MAT2, etc.)
- Nonstructural mass defined on the property entries

You define mass density on the material entries in terms of mass/unit volume. The mass unit must be consistent with the other units in the model. For example, in the English system (in, lb, sec), the mass density of steel is approximately 7.32×10^{-4} lb-sec²/in⁴.

The software adds the nonstructural mass you define on the property entries to the structure in addition to the structural mass from the elements.

- For one-dimensional elements, the units are mass/unit length.
- For two-dimensional and three-dimensional elements, the units are mass/unit area and mass/unit volume, respectively.

Using the WTMASS Parameter to Express Mass in Weight Units

In some cases, you may want to use the WTMASS parameter to express the mass in terms of weight units instead of mass units. The WTMASS parameter multiplies the assembled mass matrix by the scale factor you specify with the WTMASS parameter.

For the steel example, you can enter the mass density a weight density of 0.283 lb/in³ with a WTMASS parameter of 0.00259 (which is 1/386.4). However, if you enter any of the mass in terms of weight, you must enter all the mass in terms of weight. The WTMASS multiplies all the mass in the model by the same scale factor, with the only exception being the mass you enter in Direct Matrix Input.

See also

- “WTMASS” in the *NX Nastran Quick Reference Guide*

WTMASS Example

For example, consider the two rotating masses shown in [Figure 8-13](#). The masses are accelerated at constant angular acceleration of 20 rev/sec^2 from 0 to 200 rev/min. The goal is to determine the axial force in the CBAR as a function of angular velocity.

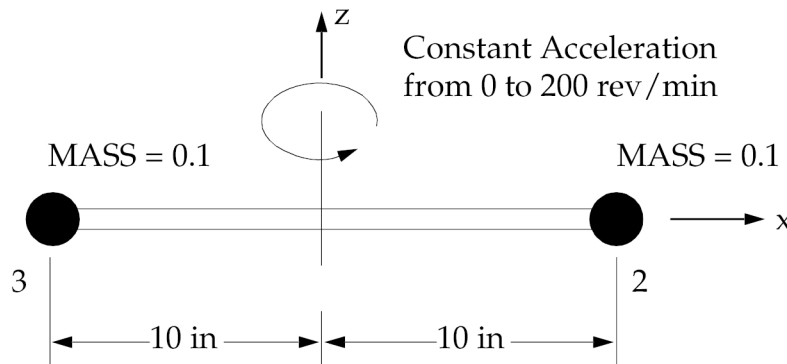


Figure 8-13. Rotating Concentrated Masses

The input file is shown in [Listing 8-8](#). Multiple load cases are used to determine axial forces at six rotational speeds. Using multiple load cases for this problem is much more efficient than submitting separate runs because NX Nastran doesn't have to regenerate the stiffness and mass matrices. Another interesting note is that the model doesn't actually rotate; it is fixed at the center. If the constraints permitted the model to rotate, the analysis would fail.

```

$
$ FILENAME ROTATE.DAT
$
ID LINEAR,ROTATE
SOL 101
TIME 2
CEND
$
  FORCE   = ALL
SUBCASE 1
  LABEL = 0 REV/min
  LOAD   = 1
SUBCASE 2
  LABEL = 40 REV/min
  LOAD   = 2
SUBCASE 3
  LABEL = 80 REV/min
  LOAD   = 3
SUBCASE 4
  LABEL = 120 REV/MIN
  LOAD   = 4
SUBCASE 5
  LABEL = 160 REV/MIN
  LOAD   = 5
SUBCASE 6
  LABEL = 200 REV/MIN
  LOAD   = 6
BEGIN BULK
PARAM  GRDPNT  0
$
GRID   1          0.0      0.0      0.0          123456
GRID   2          10.      0.0      0.0
GRID   3         -10.      0.0      0.0
$
CBAR   10         1        1        2        1.0      0.0      1.0
CBAR   11         1        1        3        1.0      0.0      1.0
$
PBAR   1          1        1.0      1.0      1.0      1.0
$
MAT1   1          30.E6          .3
$
RFORCE 1          1          0.      0.0      0.0      1.0      1
20.
RFORCE 2          1          .666667 0.0      0.0      1.0      1
20.
RFORCE 3          1          1.333333 0.0      0.0      1.0      1
20.
RFORCE 4          1          2.0      0.0      0.0      1.0      1
20.
RFORCE 5          1          2.666667 0.0      0.0      1.0      1
20.
RFORCE 6          1          3.333333 0.0      0.0      1.0      1
20.
$
CONM2  101        2          .1
CONM2  102        3          .1
$
ENDDATA

```

Listing 8-8. Axial Force in the Rotating Arms

Figure 8-14 shows a plot summarizing the axial force. The plot isn't linear because centrifugal forces aren't linear with angular velocity. Consequently, using a SUBCOM command to combine the subcases with RFORCE entries can lead to misleading results. In other words, the element axial forces for Subcase 3 can't be obtained by merely doubling the element axial forces of Subcase 2

by using a SUBCOM/ SUBSEQ combination. This occurs because the axial force is a function of ω^2 , and not 2ω .

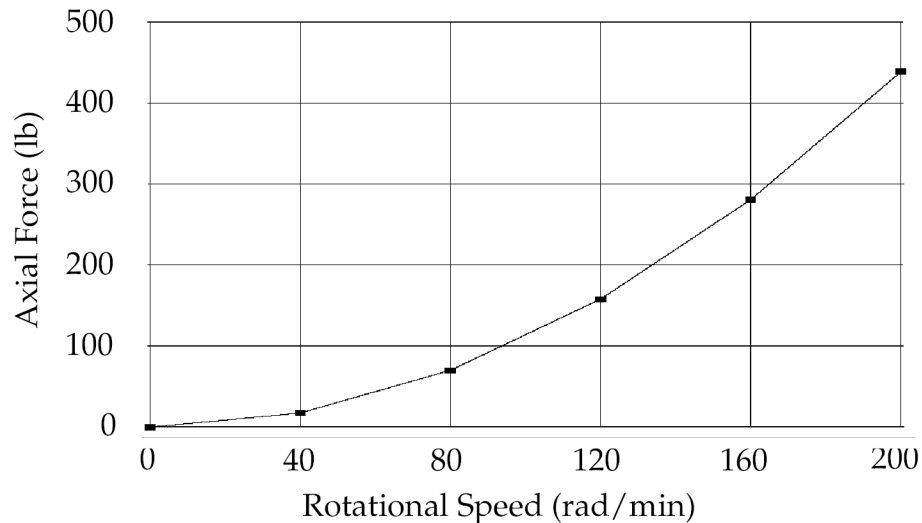


Figure 8-14. Axial Force in the Bar as a Function of Angular Velocity

8.7 Thermal Loads

You can define thermal loads on a structure to perform stress analysis or to determine thermal expansion. In NX Nastran, you must define a temperature distribution via TEMPIj Bulk Data entries and thermal expansion coefficients.

- Thermal expansion coefficients are specified on the material Bulk Data entries.
- Temperatures can be specified at grid points (TEMP and TEMPD entries) and interpolated to grid points within elements.

You can also specify temperature data on an element-by-element basis as shown in [Table 8-2](#).

Table 8-2. Bulk Data Entries Used for Temperature Definition on Elements		
Elements	Temperature Data	Bulk Data Entry
CROD, CONROD, CTUBE	Average temperature at ends A and B.	TEMPRB
CBAR, CBEAM, CBEND	Average temperature and cross-sectional temperature gradients at ends A and B.	TEMPRB
CQUAD4, CTRIA3CQUAD8, CTRIA6	Average temperature and gradient in the thickness direction.	TEMPP1

Any average temperatures you specify directly for an element take precedence over the temperatures that the software interpolates from the element's connected grid points. Solid elements obtain their temperatures only by interpolation from connected grid points. Note that interpolated grid point

temperatures provide temperature gradients over the neutral surface of shell elements, whereas the TEMPPi entries do not.

NX Nastran uses the temperature data and thermal expansion coefficients to calculate equivalent forces and moments acting at the grid points.

The TEMP(INIT) and TEMP(LOAD) Case Control commands specify the initial temperature and applied temperature, respectively. The TEMP(INIT) command must appear either above the first subcase or inside the first subcase. As with other types of loads, NX Nastran only applies a thermal load if you specifically request it in the case control section.

Defining Temperatures for Grid Points

You can use the TEMP and TEMPD bulk data entries to define grid point temperatures.

- TEMP lets you define a scalar value of temperature at selected grid points.
- TEMPD lets you define a default temperature for all grid points whose temperatures aren't defined by a TEMP entry.

See also

- “TEMP” in the *NX Nastran Quick Reference Guide*
- “TEMPD” in the *NX Nastran Quick Reference Guide*

Defining Temperatures for Elements

You can define temperatures for selected elements.

- You can specify temperatures for CROD, CBAR, CBEAM, CBEND, CONROD, or TUBE elements on the TEMPRB bulk data entry. TEMPRB specifies the average temperature on both ends. With CBAR, CBEAM and CBEND elements, you can use TEMPRB to define temperature gradients over the cross section.
- You can specify temperatures for two-dimensional plate and membrane elements on a TEMPP1, TEMPP2, or TEMPP3 bulk data entry. The user-defined average temperature over the volume is used to produce in-plane loads and stresses. You can use thermal gradients over the depth of the bending elements, or the resulting moments to produce bending loads and stresses.

If you don't define any thermal element data for an element, the software averages temperatures of the connected grid points from the TEMP, TEMPD, or TEMPAX entries to calculate an average temperature for the element. The thermal expansion coefficients are defined on the material definition entries. Regardless of the type of thermal data, if the material coefficients for an element are temperature dependent by use of the MATTi entry, the software always calculates them from the “average” temperature of the element.

Understanding Coefficient of Thermal Expansion (CTE)

It's important to understand how the coefficient of thermal expansion (CTE) are defined and how they are used in both linear and nonlinear solutions. There are two typical forms of thermal expansion data, tangent data and secant data. The CTE value can vary considerably depending on which of these definitions is used.

General Definition of CTE

The coefficient of thermal expansion is defined as the normalized change in length for a given change in temperature:

$$\alpha(T) = \frac{1}{\Delta T} \left(\frac{\Delta L}{L} \right)$$

Equation 8-4.

such that the thermal strain is given by:

$$\varepsilon = \alpha \cdot \Delta T$$

Equation 8-5.

Tangent CTE Data

The tangent data for the coefficient of thermal expansion is defined by the following derivative approximation:

$$\alpha_1 = \frac{\Delta L_1}{L} \left(\frac{1}{T_1} \right)$$

Equation 8-6.

such that the thermal strain is given by:

$$\varepsilon = \int_{T_{init}}^{T_{load}} \alpha(T) dT$$

Equation 8-7.

Secant Data

ΔT for secant CTE data is the temperature range $T - T_{ref}$:

$$\alpha(T) = \frac{1}{T - T_{ref}} \left(\frac{\Delta L}{L} \right)$$

Equation 8-8.

The secant data always begins at some reference temperature (T_{ref}), which is assumed to be the zero thermal strain condition. The axial strain is then measured at the consecutive temperatures, and divided by the temperature change ($T - T_{ref}$). For example, to calculate the secant CTE at temperatures T_1 , T_2 , and T_3 :

$$\alpha_1 = \frac{\Delta L_1}{L(T_1 - T_{ref})}$$

Equation 8-9.

where ΔL_1 is the axial deflection going from Tref to T1

$$\alpha_2 = \frac{\Delta L_2}{L(T_2 - T_{ref})}$$

Equation 8-10.

where ΔL_2 is the axial deflection going from Tref to T2

$$\alpha_3 = \frac{\Delta L_3}{L(T_3 - T_{ref})}$$

Equation 8-11.

where ΔL_3 is the axial deflection going from Tref to T3.

The secant data is then applied such that when a model has an initial temperature of T2 and a final temperature of T3, the resulting thermal strain is equal to the thermal strain from “Tref to T3” minus the thermal strain from “Tref to T2”. The thermal strain is accurately calculated from secant data using:

$$\varepsilon(T) = \alpha_{load}(T_{load} - T_{ref}) - \alpha_{init}(T_{init} - T_{ref})$$

Equation 8-12.

CTE in Linear Solutions

The linear solution thermal strain calculation depends on the case control and the material definition. The linear solutions assume a tangent CTE data described in equation 8-6. However, secant data can be used as a good approximation if the variation of the CTEs as a function of temperature is small:

When neither TEMP(MAT) nor TEMP(INIT) is specified:

$$\varepsilon = \alpha(T_{load} - T_{ref})$$

Equation 8-13.

When using TEMP(INIT) with no MATTi:

$$\varepsilon = \alpha(T_{load} - T_{init})$$

Equation 8-14.

When using TEMP(MAT) with MATTi:

$$\varepsilon = \alpha_{mat}(T_{load} - T_{ref})$$

Equation 8-15.

where $\alpha(mat)$ is the CTE evaluated at TEMP(MAT).

When using TEMP(INIT) with MATTi:

$$\varepsilon = \alpha_{init} (T_{load} - T_{init})$$

Equation 8-16.

where α_{init} is the CTE evaluated at TEMP(INIT).

CTE in Nonlinear Solutions

Solution 106 uses the secant equation 8-12 when the CTEs are temperature dependent:

$$\varepsilon(T) = \alpha_{load} (T_{load} - T_{ref}) - \alpha_{init} (T_{init} - T_{ref})$$

where α_{load} is the CTE evaluated at TEMP(LOAD) and α_{init} is the CTE evaluated at TEMP(INIT).

For solution 106 temperature dependent CTEs (using MATTi), TEMP(INIT) is required and TEMP(MAT) is not allowed. If TEMP(LOAD) is not specified, the thermal load set will default to TEMP(INIT). The reference temperature is defined on the MATi card or PCOMP card for composites.

When the CTE is constant in solution 106, equations 8-13 and 8-14 apply.

Rigid Element Thermal Strain

The RBAR, RBE1, RBE2, RBE3, RROD and RTRPLT elements support a thermal strain calculation when the RIGID case control command is assigned to LAGRAN (default), and alpha is defined on the rigid element bulk data entry.

The rigid element thermal strain is calculated as follows:

The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

The rigid element thermal strains are calculated by
 $\varepsilon_{thermal} = \alpha(AVGTEMP(LOAD) - AVGTEMP(INIT)).$

If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

See the RIGID case control command in the *NX Nastran Quick Reference Guide* for more details.

Using Temperature Loads with Subcases

If temperature loads are present in one or more of the subcases in your input file, the software must take thermal strain into account in the calculation of element stresses and forces. That is, a body subjected to a temperature field will deform in a stress free manner in the absence of constraints. However, if the deformation is resisted through boundary conditions and/or the presence of applied loads, the total actual strain, at a point must be reduced by the thermal strain, $\alpha(T - T_o)$, to obtain the stress at the point. For example, the stress-strain relation for a rod that undergoes thermal expansion and also sustains an axial force is written in the form

$$\varepsilon - \alpha(T - T_o) = \frac{\sigma}{E}$$

Equation 8-17.

NX Nastran calculates element stresses and forces on an element-by-element basis from the displacement vector and the temperature field. Therefore, if you're using SUBCOM or SYMCOM to combine selected subcases, you must supply a definition of the temperature field whenever element stresses and forces are required.

Consider the following example where a temperature load is specified in SUBCASE 2.

```

      .
      .
      .
SUBCASE 1
  LOAD=2
SUBCASE 2
  TEMP (LOAD) =4
SUBCASE 10
  LOAD=10
SUBCOM 20
  SUBSEQ=.5,1.,1.6
  TEMP (LOAD) =4
      .
      .
      .

```

Here, the multiplier for SUBCASE 2 is unity so that the temperature distribution required to calculate element forces and stresses for the combined loading case is precisely TEMP(LOAD)=4. If, on the other hand, the SUBSEQ entry is modified as indicated below, an additional TEMP(LOAD) must be defined.

```

      .
      .
      .
SUBCASE 1
  LOAD=1
SUBCASE 2
  TEMP (LOAD) =4
SUBCASE 10
  LOAD=10
SUBCOM 20 SUBSEQ=.5,.75,1.6
  TEMP (LOAD) =40

```

Here, TEMP(LOAD)=40 must reflect a temperature distribution of the form

$$(T_{40} - T_o) = .75(T_4 - T_o)$$

Equation 8-18.

$$T_{40} = T_o + .75(T_4 - T_o)$$

Equation 8-19.

You must supply this new TEMP(LOAD) through standard Bulk Data entries such as TEMPD, TEMP, TEMPP1, and TEMPP3 and then call them out in the Case Control section with the SUBCOM command.

In general, with n temperature loading conditions in the subcases immediately preceding a SUBCOM, the temperature distribution that must be defined under the SUBCOM if you want element force or stress output is of the form

$$T_{SUBCOM} = T_o + \sum_{i=1}^n a_i(T_i - T_o)$$

Equation 8-20.

where the coefficients a_i are defined on the SUBSEQ entry under the SUBCOM in question. The reference temperature T_o , defined on MATi Bulk Data entries, can conveniently be defined as zero in order to simplify the calculation of the temperature distribution for the combined loading condition of the SUBCOM.

The same approach is used with combined loading condition defined through a SYMCOM Case Control entry. However, the use of SYM subcases and SYMCOM entries implies the use of the concepts of reflective symmetry, in which case you should use the DIH option of cyclic symmetry. NX Nastran internally generates the appropriate boundary conditions, and you simply define the actual temperature distribution on each portion of the structure without regard to decomposing the temperatures and/or mechanical loads into symmetric and antisymmetric components.

This is best illustrated with the example problem shown in [Listing 8-9](#).

```

.
.
.
CEND
$
TEMP (INIT) = 1
$
SUBCASE 1
TEMP (LOAD) = 10
$
SUBCASE 2
TEMP (LOAD) = 15
$ THE FOLLOWING SUBCOM WILL COMBINE
$ 100% OF THE RESULTS FROM SUBCASE 1
$ WITH 50% OF THE RESULTS FROM SUBCASE 2
SUBCOM 3
SUBSEQ = 1.0,0.5
TEMP (LOAD) = 20
$
BEGIN BULK
$
TEMPD,1, 100.
TEMPD,10,150.
TEMPD,15,250.
$ "TEMPD,20" IS CALCULATED AS FOLLOW:
$ 100 + 1.0*(150-100) + 0.5*( 250 - 100 ) = 225.
$
TEMPD,20,225.
$ REST OF THE BULK DATA ENTRIES
ENDDATA

```

Listing 8-9. Using SUBCOMs with Temperature Loads

As an example of thermal loads, consider the umbrella tent shown in [Figure 8-15](#). The tent is subjected to thermal radiation from partial exposure to the sun. The ground is held at 40 degrees Fahrenheit and the air temperature is held at 50 degrees Fahrenheit. Can the tent withstand the thermal stresses imposed by solar radiation?

This problem involves conduction, convection, and radiation, which is beyond the scope of this user's guide, but the stress analysis of the resulting temperature profile is not. For further discussion on the subject of heat transfer, refer to the *NX Nastran Thermal Analysis User's Guide*.

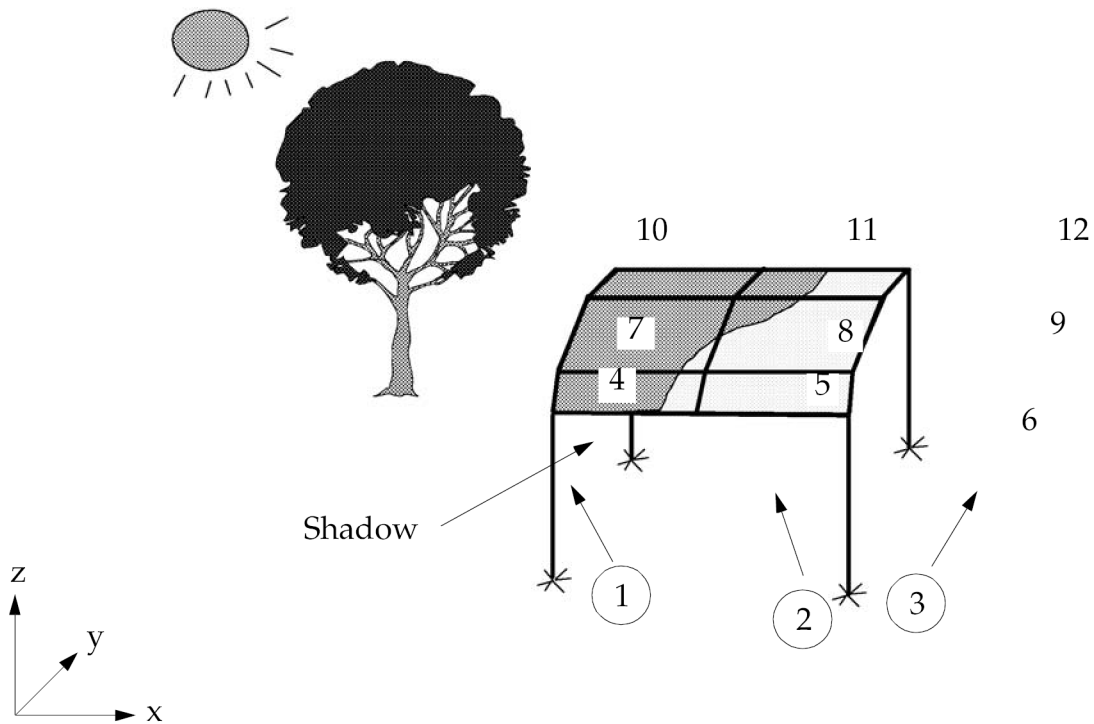


Figure 8-15. Umbrella Tent With Thermal Loads

To analyze this type of problem, you need to run the heat transfer Solution 153 to produce a temperature profile for all of the grid points in the model. A Solution 153 input file is generated for this model and is included in the Test Problem Library with the name 'heat.dat'.

In Solution 153, you have the option of creating a temperature file that can be included directly into the Solution 101 input file. The Solution 101 input file used for the stress analysis is shown in [Listing 8-10](#). The temperature file from the Solution 153 run is shown in [Listing 8-11](#). Note the use of the INCLUDE statement, which inserts the temperature input file at that location when you submit the job.


```

$ FILENAME - THERMAL.DAT
$
ID LINEAR,THERMAL
SOL 101
TIME 10
CEND
TITLE = UMBRELLA TENT WITH THERMAL LOADING
$ DEFINE INITIAL TEMPERATURE
TEMP(INIT) = 2
$ DEFINE THERMAL LOADING
TEMP = 1
STRESS = ALL
SPC = 15
BEGIN BULK
$
INCLUDE 'heat.pch'
$
TEMPD 2 40.0
$
MAT1 1 10.E6 .3 2.59E-3 21.E-6
25000. 25000. 15000.
MAT1 4 2.E6 .3 2.59E-5 21.E-6
25000. 25000. 15000.
$
$ GRID, CBAR, CQUAD4, PBAR, PSHELL, AND SPC ENTRIES
$ REMOVED TO SAVE SPACE
ENDDATA

```

Listing 8-10. Umbrella Tent Thermal Analysis

```

$
$FILENAME - HEAT.PCH
$
$TITLE = TENT THERMAL ANALYSIS SUBJECTED TO RADIATION 1
$SUBTITLE=THE BASE TEMPERATURE AT 40 DEGREE F 2
$LABEL = 3
$DISPLACEMENTS 4
$REAL OUTPUT 5
$SUBCASE ID = 1 6
TEMP* 1 1 4.011359E+01 7
TEMP* 1 2 5.999791E+01 8
TEMP* 1 3 1.194475E+02 9
TEMP* 1 4 4.001704E+01 10
TEMP* 1 5 4.705626E+01 11
TEMP* 1 6 6.816640E+01 12
TEMP* 1 7 4.000030E+01 13
TEMP* 1 8 4.004408E+01 14
TEMP* 1 9 6.814966E+01 15
TEMP* 1 10 4.005281E+01 16
TEMP* 1 11 4.012182E+01 17
TEMP* 1 12 1.193867E+02 18
TEMP* 1 99 4.000000E+01 19
TEMP* 1 101 7.500000E+01 20
TEMP* 1 103 7.500000E+01 21
TEMP* 1 110 7.500000E+01 22
TEMP* 1 112 7.500000E+01 23
TEMP* 1 999 0.000000E+00 24

```

Listing 8-11. Temperature Distribution Generated with Solution 153

The OLOAD output shows the applied loads generated by the imposed temperature. The thermal stresses for this example are quite small, so the structure should have no difficulty surviving the thermal loading.

See also

- “[Understanding Cyclic Symmetry](#)” in the *NX Nastran User's Guide*

8.8 Enforced Element Deformation Loads

NX Nastran includes a limited capability to define enforced axial deformations. You can use the DEFORM Bulk Data entry to apply an enforced extension (units of length) to CBAR, CBEAM, CONROD, CROD, and CTUBE elements.

Enforced deformations, for example, are useful in the simulation of misfit or misalignment in engineering structures. As in the case of thermal expansion, the software calculates the equivalent loads by separate subroutines for each type of structural element and are applied to the connected grid points. The magnitude of the axial deformation is specified on a DEFORM entry.

You can only use DEFORM in SOLs 101, 105, 114, and 200. To apply enforced displacements in other solution sequences, use SPCD.

With DEFORM, the enforced deformation is analogous to a turnbuckle that can be adjusted to provide a desired axial deformation in an element. The appropriate stress-strain relation for a rod element of length that undergoes an enforced axial deformation, δ is given by the following equation:

$$\varepsilon - \frac{\delta}{l} = \frac{\sigma}{E}$$

Equation 8-21.

Consider this example:

```

      .
      .
      .
SUBCASE 1
  LOAD=1
SUBCASE 2
  DEFORM=5
SUBCASE 10
  LOAD=10
SUBCOM 20
SUBSEQ=.5, .75, 1.6
  DEFORM=50
      .
      .
      .

```

Here DEFORM = 50 reflects a set of enforced deformations of the form

$$\delta_{50} = .75\delta_5$$

Equation 8-22.

For n DEFORM loading conditions in the subcases that immediately precede a SUBCOM, the DEFORM loading condition that must be defined under the SUBCOM, if element force and stress output is desired, is of the form

$$\delta_{SUBCOM} = \sum_{i=1}^n a_i \delta_i$$

Equation 8-23.

The format of the DEFORM entry is shown below:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

Field	Contents
SID	Deformation set identification number. (Integer > 0)
EIDi	Element number. (Integer > 0)
Di	Deformation. (Real; positive value represents elongation; negative value represents contraction)

You must use the DEFORM=SID to select the DEFORM entry in the Case Control section.

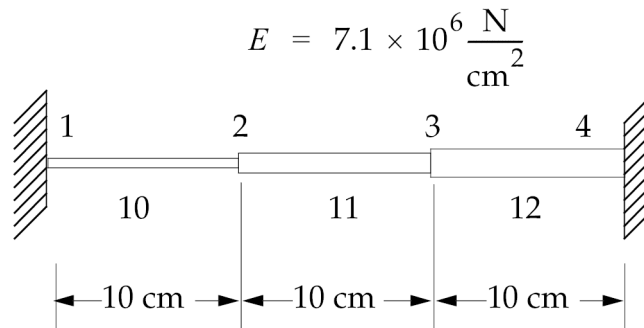
When you use DEFORM, you must remember that you're not enforcing a strain or an actual extensional length to the element. Rather, you're applying a force to the element that produces the specified extension if the element is free to expand without internal forces being generated. The software adds this computed force to the other forces in the model. Since most elements in your model aren't free to expand, the extension value you specify may not be achieved. This feature is best illustrated with an example.

See also

- “**DEFORM**” in the *NX Nastran Quick Reference Guide*
- “**SPCD**” in the *NX Nastran Quick Reference Guide*

DEFORM Example

Suppose you want to use the DEFORM entry to enforce a strain on the three-member structure shown in **Figure 8-16**. The goal is to impose an initial strain equal to 100 $\mu\text{cm}/\text{cm}$ in the center member.

**Figure 8-16. Three-Bar Structure with Initial Strain**

Consider: (1) the DEFORM entry can only impose a force, (2) from static equilibrium, if no forces are imposed on grid points 2 and 3, the force in all three members must be the same. With this in mind, the following procedure is used.

1. Determine the force required in member 11 that produces the 100 mcm/cm strain.

$$\begin{aligned}\sigma_{11} &= E\varepsilon \\ &= (7.1 \times 10^6)(100 \times 10^{-6}) = 710 \frac{\text{N}}{\text{cm}^2} \\ P &= A\sigma_{11} \\ &= (1.5)(710) = 1065 \text{ N}\end{aligned}$$

2. Since there are no external forces acting on the elements, the force of 1065 N is same in all the members. Using the 1065 N force, the stresses computed in the other members.

$$\begin{aligned}\sigma_{10} &= \frac{P}{A_{10}} = \frac{1065}{1} = 1065 \frac{\text{N}}{\text{cm}^2} \\ \sigma_{12} &= \frac{P}{A_{12}} = \frac{1065}{2} = 532.5 \frac{\text{N}}{\text{cm}^2}\end{aligned}$$

3. From the stresses found in part 2, the element strains are computed for elements 10 and 12. Multiplying the strain within each element by its length yields the extensional change of the element. These extensional changes are the deform values required to produce a 100 $\mu\text{cm}/\text{cm}$ strain in element 11.

$$\begin{aligned}\varepsilon_{10} &= \frac{\sigma_{10}}{E} = 150 \cdot 10^{-6} \\ \varepsilon_{12} &= \frac{\sigma_{12}}{E} = 75.0 \cdot 10^{-6}\end{aligned}$$

Therefore,

$$\Delta L_{10} = L_{10} \varepsilon_{10} = (10)(150 \cdot 10^{-6}) = 1.5 \cdot 10^{-3} \text{ cm}$$

$$\Delta L_{11} = L_{11} \varepsilon_{11} = (10)(150 \cdot 10^{-6}) = 1.0 \cdot 10^{-3} \text{ cm}$$

$$\Delta L_{12} = L_{12} \varepsilon_{12} = (10)(75 \cdot 10^{-6}) = 0.75 \cdot 10^{-3} \text{ cm}$$

These values are entered on the DEFORM Bulk Data entry as shown in [Listing 8-12](#). To help understand the implementation of the DEFORM entry, two configurations are being analyzed. The first configuration (Subcase 1) specifies the boundary conditions as shown in [Figure 8-17\(a\)](#). For this case, the structure is free to expand without inducing any element forces. The results of this subcase should be the extension of the members as entered on the DEFORM entry. As mentioned previously, a force is applied to the element that produces the specified extension if the element is free to expand.

The second subcase, shown in [Figure 8-17\(b\)](#), is the constrained configuration that is of interest. If the calculations are correct, the strain in the center element should be the required 100 $\mu\text{cm/cm}$.

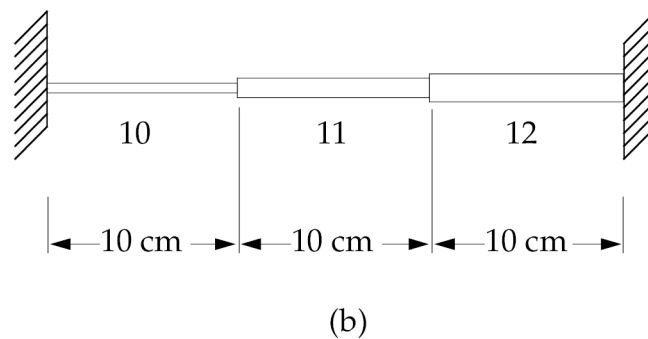
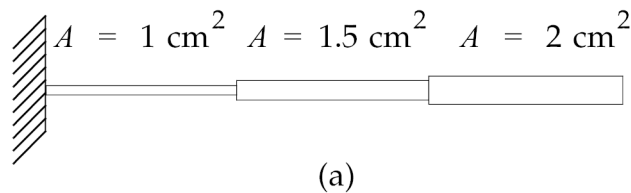


Figure 8-17. Three-Bar Structures with Initial Strain

```

$ FILENAME - STRAIN.DAT
ID LINEAR,STRAIN
SOL 101
TIME 2
CEND
$
DISP      = ALL
STRESS    = ALL
DEFORM    = 1
SUBCASE 1
SUBCASE 2
  SPC = 1
BEGIN BULK
$
SPC      1      4      1
$
GRID     1      0.0    0.0    0.0      123456
GRID     2      10.    0.0    0.0      23456
GRID     3      20.    0.0    0.0      23456
GRID     4      30.    0.0    0.0      23456
$
CBAR     10     1      1      2      1.0    0.0    1.0
CBAR     11     2      2      3      1.0    0.0    1.0
CBAR     12     3      3      4      1.0    0.0    1.0
$
PBAR     1      1      1.0
PBAR     2      1      1.5
PBAR     3      1      2.0
$
MAT1     1      7.1E6      .3
$
DEFORM   1      10      1.5E-3  11      1.E-3  12      .75E-3
ENDDATA

```

Listing 8-12. Initial Strain Input File

The stresses and displacements for the two subcases are shown in [Figure 8-18](#). As can be seen for the free end configuration, the extension of all of the members is consistent with the DEFORM entry.

Note that the extensions for the elements aren't the same as the values specified on the DEFORM entry. The stress in member 11 is 710 N/cm². Since $E = 7.1 \times 10^6$, the strain in element 11 is equal to $\epsilon = E/\sigma = 100 \mu\text{m}/\text{cm}$ strain. The stress output is requested for this example because strain output is not available for one-dimensional elements.

D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	1.500000E-03	.0	.0	.0	.0	.0	
3	G	2.500000E-03	.0	.0	.0	.0	.0	
4	G	3.250000E-03	.0	.0	.0	.0	.0	
SUBCASE 2								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	-1.791005E-11	.0	.0	.0	.0	.0	
3	G	8.955025E-12	.0	.0	.0	.0	.0	
4	G	.0	.0	.0	.0	.0	.0	
SUBCASE 1								
S T R E S S E S I N B A R E L E M E N T S (C B A R)								
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
10	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0		.0	.0	
11	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0		.0	.0	
12	.0	.0	.0	.0	.0	.0	.0	
	.0	.0	.0	.0		.0	.0	
SUBCASE 2								
ELEMENT ID.	SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
10	.0	.0	.0	.0	-1.065000E+03	-1.065000E+03	-1.065000E+03	
	.0	.0	.0	.0		-1.065000E+03	-1.065000E+03	
11	.0	.0	.0	.0	-7.100000E+02	-7.100000E+02	-7.100000E+02	
	.0	.0	.0	.0		-7.100000E+02	-7.100000E+02	
12	.0	.0	.0	.0	-5.325000E+02	-5.325000E+02	-5.325000E+02	
	.0	.0	.0	.0		-5.325000E+02	-5.325000E+02	

Figure 8-18. Stress and Displacement Output of the Three Bar Structure

8.9 Loads Due to Enforced Motion

Statics

You can specify enforced zero displacements on GRID, SPC, or SPC1 entries.

- If you use GRID entries, the constraints become part of the structural model and modifications cannot be made at the subcase level.
- SPC or SPC1 entries are usually used for defining zero displacements, which result in nonzero forces of constraint.

You can specify nonzero enforced displacements on SPC or SPCD entries.

- The SPC entry lets you specify both the component to be constrained and the magnitude of the enforced displacement.

- The SPCD entry lets you specify only the magnitude of the enforced grid point displacement. When you use an SPCD entry, you must specify the component to be constrained on either an SPC or SPC1 entry. Using SPCD avoids the decomposition of the stiffness matrix when changes are only made in the magnitudes of the enforced displacements.

NX Nastran calculates the equivalent loads resulting from enforced displacements of grid points and adds them to the other applied loads. You specify the magnitudes of the enforced displacements on SPC entries (SPCAX in the case of conical shell problems) in the global coordinate system. NX Nastran automatically applies the load when you select the associated SPC set in the Case Control section.

See also

- “**SPC**” in the *NX Nastran Quick Reference Guide*
- “**SPCD**” in the *NX Nastran Quick Reference Guide*
- “**SPCAX**” in the *NX Nastran Quick Reference Guide*

Dynamics

In NX Nastran, you can use the:

- TLOAD1 and TLOAD2 entries to define a time-dependent dynamic load or enforced motion for use in a transient response analysis
- RLOAD1 and RLOAD2 entries to define a frequency-dependent dynamic load for use in a frequency response analysis

You can use the TYPE field (field5 on TLOAD1/TLOAD2 and field 8 on RLOAD1 and RLOAD2) to specify the type of dynamic excitation you're creating.

- If the TYPE field on the TLOAD1/2 entries indicates an enforced motion, the software first assumes that the EXCITEID field points to SPC-type data. If not present, NX Nastran then assumes the excitation is enforced motion using large masses and will then look for DAREA and various static loading data, just as in the case of applied loads.
- With RLOAD1 and RLOAD2 entries, the software only look for SPC-type data in the case of enforced motion.

For all entries (TLOAD1/2, RLOAD1/2), if the TYPE field indicates an applied load, the program will search only for static loading data.

See also

- “**TLOAD1**” in the *NX Nastran Quick Reference Guide*
- “**TLOAD2**” in the *NX Nastran Quick Reference Guide*
- “**RLOAD1**” in the *NX Nastran Quick Reference Guide*
- “**RLOAD2**” in the *NX Nastran Quick Reference Guide*

Specifying Enforced Motion Data

In NX Nastran, you specify enforced motion data on the SPC/SPC1/SPCD Bulk Data entries. (Note that components specified in SPCD data must be also be referenced on SPC or SPC1 entries.)

You can define enforced motion with SPC data (without the use of SPCDs) alone. However, such usage is discouraged in favor of SPC/SPC1 data in connection with SPCDs. In this way, you select SPCD entries via the EXCITEID field in the dynamic load data, while the software uses the SPC/SPC1 sets to identify the constrained displacement set in the Case Control section.

With SPC data alone, enforced motion is applied solely via the Case Control section. The entries referenced by the EXCITEID field on the TLOAD1/2 or RLOAD1/2 entries do not have to exist. This method of specifying enforced motion is less elegant and somewhat more difficult to interpret.

See also

- “**SPC**” in the *NX Nastran Quick Reference Guide*
- “**SPC1**” in the *NX Nastran Quick Reference Guide*
- “**SPCD**” in the *NX Nastran Quick Reference Guide*

Using Residual Vectors (Modal Augmentation) in Modal Dynamic Response Analysis

When enforced motion is employed in modal dynamic response analysis, using modal augmentation vectors is highly recommended in order to improve solution accuracy. Without these vectors, your obtained results may be unacceptable.

Qualitatively, the enforced motion shapes, or “modes,” may not necessarily be represented by the component modes of the structure, leading to a truncation of the enforced motion shapes during projection to the modal space. To obtain modal residual vectors from applied loads using modal augmentation vectors, specify PARAM,RESVEC,YES in the Bulk Data section. NX Nastran automatically derives the necessary static loads from the enforced motion data; you don't need to specify any additional loads.

In summary, the procedure for applying enforced motion in a dynamic analysis:

1. Specify the appropriate type of enforced motion to be applied via the TYPE field in RLOAD1/RLOAD2 (field 8) or TLOAD1/TLOAD2 (field 5) Bulk Data entry, as appropriate.
2. Define the desired enforced motion using the SPCD Bulk Data entry. The set IDs of these SPCD data must match the IDs appearing in the EXCITEID fields of the corresponding dynamic load data in step 1.
3. Ensure that the components referenced in the SPCD Bulk Data entry above are also specified in the SPC1 Bulk Data entry and this SPC1 is also selected in the Case Control Section.
4. Ensure PARAM,RESVEC is set to YES in the Bulk Data Section if a modal dynamic solution approach is used.

See also

- “**RESVEC**” in the *NX Nastran Quick Reference Guide*

8.10 Frequency-dependent Loads

You use the DLOAD entry to define linear combinations of frequency dependent loads that are defined on RLOAD1, RLOAD2, or ACSRCE entries. The RLOAD1 entry defines a frequency dependent load of the form

$$\{P(f)\} = A[C(f) + iD(f)]e^{i(\theta - 2\pi f\tau)}$$

Equation 8-24.

where A is defined on a load entry, for example DAREA, $C(f)$ and $D(f)$ are defined on TABLEDi entries, θ is defined on a DPHASE entry, τ is defined on a DELAY entry, and f is defined on a FREQ, FREQ1 or FREQ2 entry.

The RLOAD2 entry defines a frequency-dependent load of the form

$$\{P(f)\} = AB(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}$$

Equation 8-25.

where A is defined on a DAREA entry, $B(f)$ and $\phi(f)$ are defined on TABLEDi entries, θ is defined on a DPHASE entry, and τ is defined on a DELAY entry. DAREA, DELAY, and DPHASE entries may be different for each loaded degree-of-freedom. The loads are applied to the specified components in the global coordinate system.

The ACSRCE entry defines a frequency-dependent acoustic source of the form

$$\text{Source Strength} = Q(f) = A \left[\frac{1}{2\pi f} \sqrt{\frac{8\pi CP(f)}{\rho}} \right] e^{i(\theta + 2\pi f\tau)}$$

$$C^2 = B / \rho$$

where A is defined on a load entry, for example DAREA, $P(f)$ is defined on TABLEDi entries, θ is optionally defined on a DPHASE entry, τ is optionally defined on a DELAY entry, ρ and B are defined on the ACSRCE entry, and f is defined on a FREQ, FREQ1 or FREQ2 entry.

The RANDPS entry defines load set power spectral density factors for use in random analysis of the form

$$S_{jk}(f) = (X + iY)G(f)$$

Equation 8-26.

where $G(f)$ is defined on a TABRNDi entry. The subscripts j and k define the subcase numbers of the load definitions. If the applied loads are independent, only the diagonal terms ($j = k$) need be defined. You use the RANDT1 entry to specify the time lag constants for use in the computation of the auto correlation functions.

See also

- “Frequency Response Analysis” in the *NX Nastran Basic Dynamic Analysis User's Guide* (for a discussion of frequency response calculations)
- Section 12.2 of *The NASTRAN Theoretical Manual* (for a discussion of random response calculations)

8.11 Time-dependent Loads

You can use the DLOAD entry to define linear combinations of time-dependent loads that are defined on TLOAD1 and TLOAD2 entries. The TLOAD1 entry defines a time-dependent load of the form

$$\{P(t)\} = \{AF(t - \tau)\}$$

Equation 8-27.

where A is defined on a DAREA entry, τ is defined on a DELAY entry, and $F(t - \tau)$ is defined on a TABLEDi entry. The TLOAD2 entry defines a time-dependent load of the form

$$\{P(t)\} = \begin{cases} 0 & , t < (T_1 + \tau) \text{ or } t > (T_2 + \tau) \\ A\tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , (T_1 + \tau) \leq t \leq (T_2 + \tau) \end{cases}$$

Equation 8-28.

where T_1 and T_2 are time constants, $\tilde{t} = t - T_1 - \tau$, A and τ are the same as on the TLOAD1 entry, B is the growth coefficient, C is the exponential coefficient, F is the frequency, and P is the phase angle.

The coefficients on the DAREA and DELAY entries may be different for each loaded degree-of-freedom. The loads are applied to the specified components in the global coordinate system.

Two methods are available to automatically create DAREA values from static load data.

- Simple static loads that supply load values directly to grid points (such as FORCE, MOMENT, SLOAD, and PLOAD Bulk Data entries) will add to the DAREA factors when the load identification matches the requested set.
- More complicated loads may be generated using the LOADSET Case Control command. The requested LSEQ Bulk Data entry matches DAREA sets to static loading sets. Any type of static load (e.g., LOAD, GRAV, RFORCE, and SPCD Bulk Data entries) may be generated for either superelement or residual only (non-superelement) models.

Nonlinear effects are treated as an additional applied load vector, for which the components are functions of the degrees-of-freedom. This additional load vector is added to the right side of the equations of motion and treated along with the applied load vector during numerical integration. It is required that the points to which the nonlinear loads are applied and the degrees-of-freedom on which

they depend be members of the solution set; i.e., that they cannot be degrees-of-freedom eliminated by constraints. It is further required that, if a modal formulation is used, the points referenced by the nonlinear loads be members of the set of extra scalar points introduced for dynamic analysis.

There are four different types of nonlinear transient forcing functions:

- Arbitrary function generators
- Multiplies
- Positive power functions
- Negative power functions

The NOLIN1 entry defines a nonlinear load of the form

$$P_i(t) = S_i T(u_j)$$

Equation 8-29.

where P_i is the load applied to u_i , S_i is a scale factor, $T(u_j)$ is a tabulated function defined with a TABLEDi entry, and u_j is any permissible displacement component or velocity component.

The NOLIN2 entry defines a nonlinear load of the form

$$P_i(t) = S_i u_j u_k$$

Equation 8-30.

where u_j and u_k are any permissible pair of displacement components and/or velocity components. They may be the same.

The NOLIN3 entry defines a nonlinear load of the form

$$P_i(t) = \begin{cases} -S_i(u_j)^A & , u_j > 0 \\ 0 & , u_j \leq 0 \end{cases}$$

Equation 8-31.

where A is an exponent and u_j is any permissible displacement component or velocity component.

The NOLIN4 entry defines a nonlinear load of the form

$$P_i(t) = \begin{cases} -S_i(u_j)^A, & u_j < 0 \\ 0, & u_j \geq 0 \end{cases}$$

Equation 8-32.

where A is an exponent and u_j is any permissible displacement component or velocity component.

See also

- “Transient Response Analysis” in the *NX Nastran Basic Dynamic Analysis User's Guide* (for a discussion of transient response calculations)
- Section 11.2 of *The NASTRAN Theoretical Manual*
- For a discussion of nonlinear forcing functions, see Section 11.2 of *The NASTRAN Theoretical Manual*.

8.12 Using Static Loading Data in Dynamic Analysis

To include static loading data in a dynamic analysis, all static loads and thermal loads that have the same ID as the DAREA ID on the dynamic load entry are automatically selected, as shown in the example below.

```
.
CEND
.
$ THE FOLLOWING REQUEST SELECTS RLOAD1 ENTRY 150
DLOAD = 150
.
BEGIN BULK
$ THE FOLLOWING PLOAD4 IS AUTOMATICALLY SELECTED
$ BECAUSE ITS ID OF 100 MATCHES THE EXCITEID ID OF THE
$ SELECTED RLOAD1 ENTRY
PLOAD4,100,...
RLOAD1,150,100,...
$ THE ABOVE USAGE IS EQUIVALENT TO THE USER
$ SELECTING AN LSEQ BULK DATA OF THE FORM:-
$ LSEQ,SET_ID,100,100,100
.
```

The DAREA Bulk Data entry is used to specify point loads in dynamic analysis. In the case of grid points, these loads are implicitly assumed to be in the displacement (or local) coordinate systems of those points. The software automatically converts all DAREA Bulk Data entries for grid and scalar points into the appropriate equivalent FORCE/MOMENT/SLOAD Bulk Data entries.

Two additional, important advantages of the above enhancement follow.

- When you perform dynamic analysis using the modal approach, you may want to employ residual vectors to improve the quality of the solution. In addition to specifying PARAM,RESVEC,YES in the Bulk Data Section, you must also specify static loads at those points that are dynamically excited. However, with the above automatic conversion feature, it isn't necessary to explicitly

specify static loads for the purpose of residual vector calculations. Such loads are automatically generated by NX Nastran.

- With the automatic conversion feature, you can use the DAREA entry to be used in static analysis as well. This is particularly advantageous when you want to apply loads at grid points in the displacement (or local) coordinate systems of those grid points.

When the software performs the automatic conversion, it issues you a message at the end of the Preface module that contains an image of each converted DAREA Bulk Data entry as well as the corresponding FORCE/MOMENT/SLOAD Bulk Data entry into which it has been converted.

See also

- “DAREA” in the *NX Nastran Quick Reference Guide*

8.13 Defining Loads for Cyclic Symmetry Analysis

In NX Nastran, there are several different bulk data entries that you can use to define loading for cyclic symmetry analyses.

- You can use the LOADCYH entry to define the harmonic coefficients of a load in cyclic symmetry analysis.
- You can use the LOADCYN entry to define a physical load in cyclic symmetry analysis.
- You can use the LOADCYT entry to specify a tabular load input for axisymmetric cyclic symmetry problems.

See also

- “LOADCYH” in the *NX Nastran Quick Reference Guide*
- “LOADCYN” in the *NX Nastran Quick Reference Guide*
- “LOADCYT” in the *NX Nastran Quick Reference Guide*
- “Understanding Cyclic Symmetry” in the *NX Nastran User's Guide*

8.14 Combining Loads

You can use the LOAD case control command to combine the loads you specified on other entries to form new loading conditions. You can use LOAD to define a static loading condition that's a linear combination (superposition) of load sets consisting of loads applied directly to grid points, pressure loads, gravity loads, and centrifugal forces.

Once you've used the loading bulk data entries to define the load, you can use case control commands to specify which of the loading entries you want to use in a given analysis. The case control command you use depends on the type of load or loads you're applying:

LOAD=n

DEFORM=n

TEMP (LOAD) =n

SPC=n

where n is a unique, user-defined set identification number (SID) in field 2 of the load Bulk Data entry.

For a given analysis, the total load the software applies is the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and displacement (SPC) loads.

See also

- “LOAD” in the *NX Nastran Quick Reference Guide*
- “TEMPERATURE” in the *NX Nastran Quick Reference Guide*
- “DEFORM” in the *NX Nastran Quick Reference Guide*
- “SPC” in the *NX Nastran Quick Reference Guide*

Using LOAD

You must use LOAD if you want to combine gravity loads with loads applied directly to grid points, pressure loads, or centrifugal forces, even if there is only one loading condition.

You can define up to 300 (Si, Li) pairs with a single LOAD entry.

For any particular solution, the total static load will be the sum of the applied loads (grid point loading, pressure loading, gravity loading, and centrifugal forces) and the equivalent loads.

The resulting combined load is determined by:

$$\text{LOAD} = S \sum_i S_i \cdot \{L_i\}$$

Equation 8-33.

Where:

- (S) is the overall scale factor you define on the LOAD entry
- (Si) is the scale factor you define on the LOAD entry for each individual load set
- {Li} is the applied load vector corresponding to load set ID Li

You request subcases, which may or may not include different loading conditions, in the Case Control Section. Three Case Control commands (LOAD, TEMP(LOAD), and DEFORM) are used in selecting loading conditions for subcases. Each of these commands identifies a unique set whose SID is the same as the SID of one or more Bulk Data entries. **Figure 8-19** shows the Bulk Data entries you can select with each of the three Case Control commands.

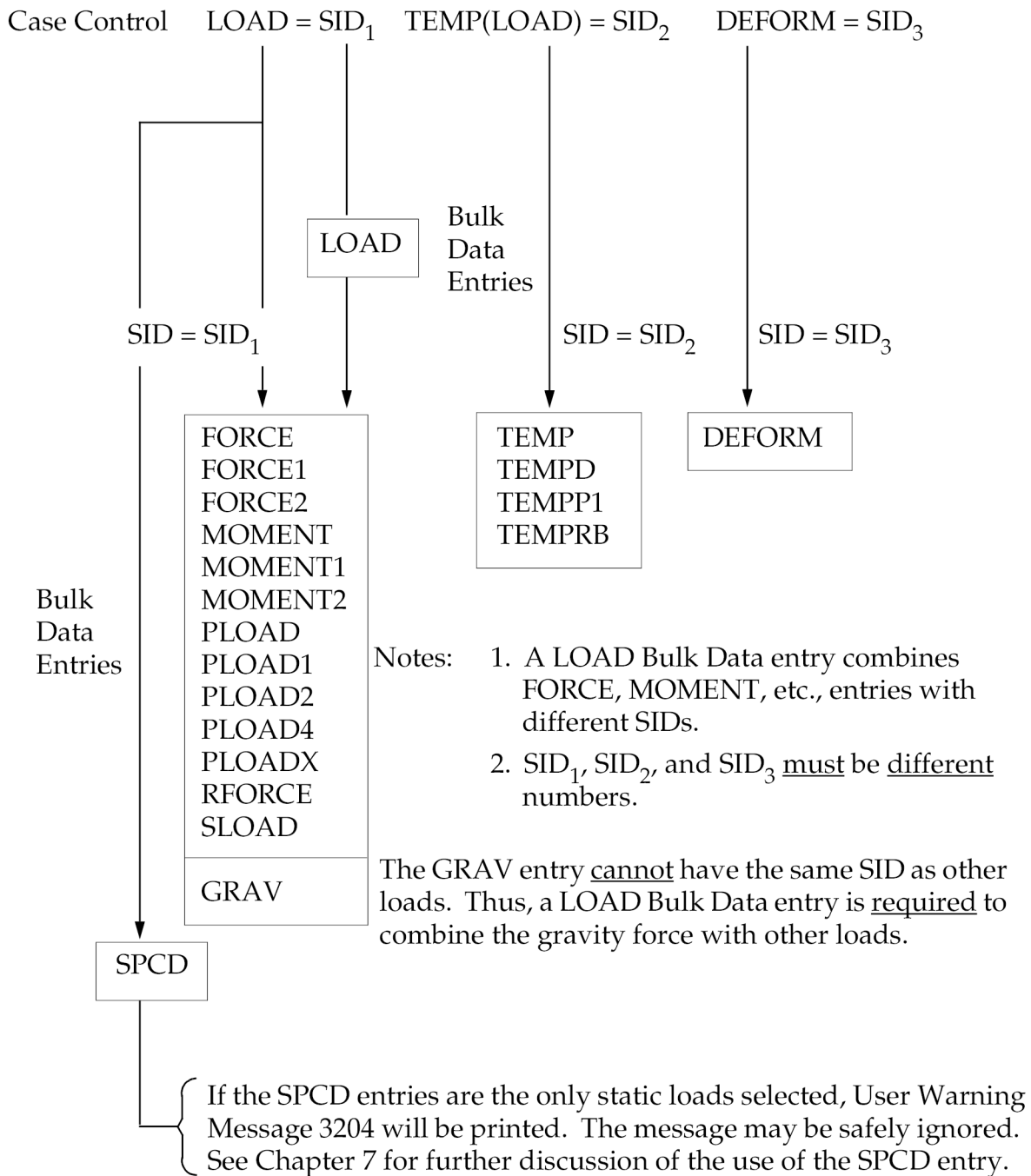


Figure 8-19. Load Combinations

LOAD Example 1

As an example of the LOAD entry combining two loads, consider the umbrella tent shown in [Figure 8-20](#).

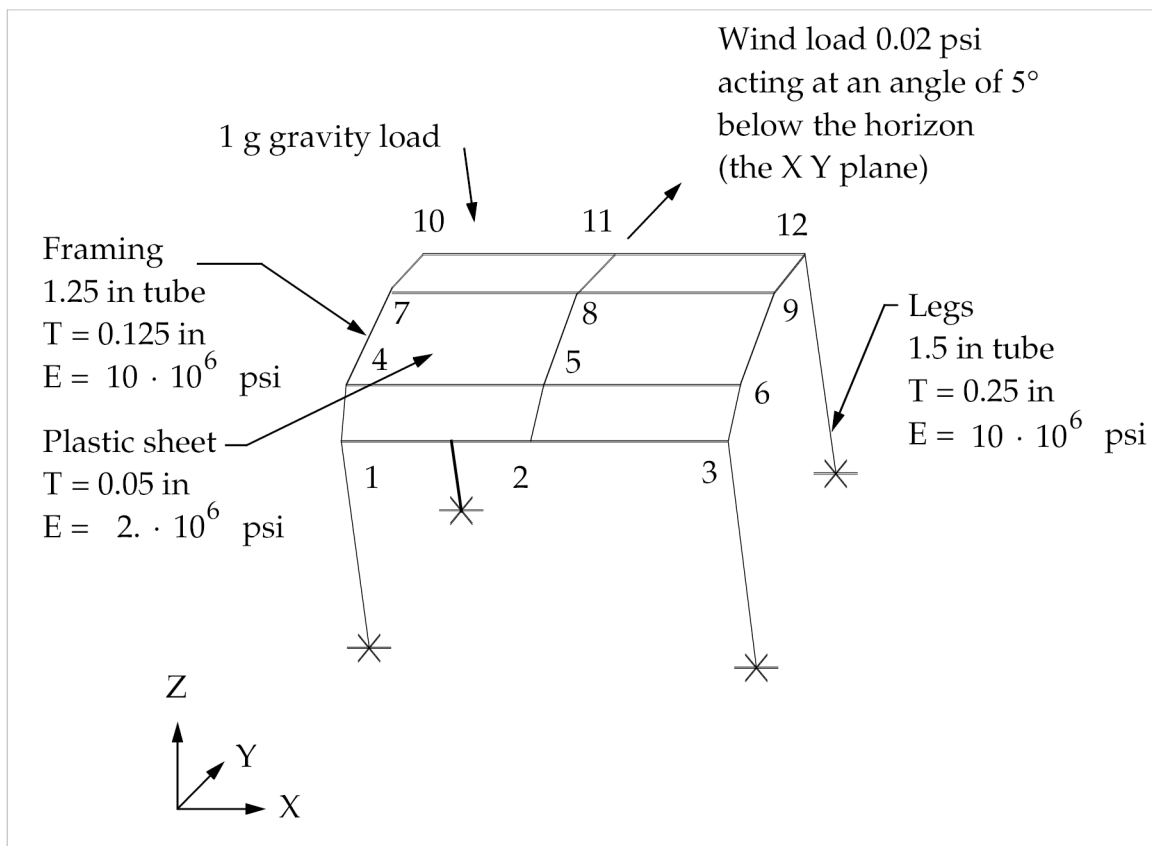


Figure 8-20. Umbrella Tent Subjected to Gravity and Wind Loads

The input file containing the combined loading is shown in [Listing 8-13](#). To apply the wind pressure in the correct direction, a local coordinate system is defined so that the local Y axis makes an angle of 5 degrees with the ground. This coordinate system is referenced on the PLOAD4 continuation entry. Since all of the elements in the model are exposed to the wind load, the simplest method is to use the thru option on the PLOAD4 entry (fields 8 and 9).

The gravity load of 386.4 entered on the GRAV entry corresponds to a 1 G load in the English system. Whenever you are working with an acceleration vector in terms of Gs, make sure that you convert the load such that its units are consistent with your structure.

```

$
$  FILENAME - TENT.DAT
$
:
LOAD=5
:
BEGIN BULK
$
CORD2R  1      0      0.      0.      0.      0.      .08716  1.
        1.      .08716  1.
$
GRAV    1      0      386.4  0.0      0.0      -1.0
PLOAD4  2      1      .02      THRU      6
        1      0.      1.      0.
LOAD    5      1.      1.      1      1.      2
$
PARAM   POST      0
PARAM   AUTOSPC   YES
PARAM   GRDPNT    0
$
PSHELL  1      4      .05
$
MAT1    1      10.E6      .3      2.59E-3
        25000.  25000.  15000.
$
MAT1    4      2.E6      .3      2.59E-5
        25000.  25000.  15000.
$
$  GRID, ELEMENT, AND CONSTRAINTS ARE NOT SHOWN
$
ENDDATA

```

Listing 8-13. Umbrella Tent with Wind and Gravity Loads

The gravity load is defined by the GRAV entry with a loading set ID of 1. The wind loading is defined using the PLOAD4 entry with a loading set ID of 2. The LOAD entry (ID of 5) is used to combine the gravity and wind loadings. The LOAD entry is selected by the LOAD = 5 command in the Case Control Section. If the GRAV and the PLOAD4 entries were assigned the same ID, a fatal error would have resulted.

LOAD Example 2

Assume that your model has one concentrated force of 15.2 lbs in the y direction applied to grid point 12, and one concentrated moment of 6.4 inch-lbs about the x-axis applied to grid point 127. It is required to double the value of force and triple the value of moment for your next analysis.

The LOAD Bulk Data entry may be written with an overall scale factor (S) of 1.0 and loadset scale factors (Si) of 2.0 for force and 3.0 for moment. Thus,

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

In Case Control:

```
LOAD = 22
```

In Bulk Data:

1	2	3	4	5	6	7	8	9	10
LOAD	22	1.0	2.0	30	3.0	40			
FORCE	30	12		15.2	0.	1.	0.		
MOMENT	40	127		6.4	1.	0.	0.		

8.15 Bolt Preload

Bolts (and certain types of threaded fasteners) are commonly tightened to levels producing very high preload forces. Preloading bolts to about 75% of their proof strength is typical. The bolt preload capability in NX Nastran allows you to predict stresses in the bolts and the bolted medium that arise from bolt preload forces alone or bolt preload forces and service loads.

Historically, bolt preload was modeled using either an equivalent thermal load approach or a multipoint constraint (MPC) approach. Both methods are capable of providing accurate results. However, both methods are labor intensive requiring multiple solutions, manual capture of data, and hand calculations.

The NX Nastran approach is much more efficient because the entire run is automated and allows for direct entry of the bolt preload forces. During the run, the model is solved twice. The first solution calculates the deformed shape of the bolted medium resulting from bolt preload forces. The software then performs an intermediate calculation. The appropriate compressive forces to apply to the bolts during the second solution are calculated correcting for differences in the length of the bolt model and the loaded length of the actual bolt. Solving the model a second time gives the stress state resulting from bolt preload forces and, optionally, service loads.

Bolts can be modeled using CBEAM and CBAR line elements or CHEXA, CPENTA, and CTETRA solid elements.

Bolt preload is supported in SOLs 101, 103, 105, 107 through 112, and 601. The description provided here applies to SOLs 101, 103, 105, 107 through 112. For information regarding bolt preload in SOL 601, see the *Advanced Nonlinear Theory and Modeling Guide*.

Related Inputs

- BOLT
The BOLT bulk entry is used to select:
 - o CBEAM and CBAR elements to treat as bolts when the bolt is modeled with line elements.
 - o GRID entries that form a cross section through the bolt when the bolt is modeled with CHEXA, CPENTA, and CTETRA solid elements.
- BOLTFOR
The BOLTFOR bulk entry is used to define the bolt preload force for BOLT bulk entries.
- BOLTLD (bulk entry)
The BOLTLD bulk entry is optionally used to define combinations of BOLTFOR bulk entries and scale the corresponding preload forces.

- **BOLTLD** (case control command)
The BOLTLD case control command is used to select a BOLTLD bulk entry or BOLTFOR bulk entries or to include in either the global case or subcases.
- **BOLTFAC**
During the first solution, the deformed shape of the bolted medium is obtained by solving the model with the axial stiffness of the bolts effectively eliminated from the model. This is accomplished by reducing the bolt axial stiffness by the BOLTFAC parameter (default value of 1×10^7). Additional information regarding this parameter is provided below.

An input file can contain BOLTLD case control commands selecting both BOLTLD and BOLTFOR bulk entries. For such a case, if the set identification number (SID) referenced by a BOLTLD case control command exists on both a BOLTLD bulk entry and BOLTFOR bulk entries, the BOLTLD bulk entry is selected.

Bolt Preload Procedure

Modeling with Line Elements

CBEAM and CBAR elements are used to represent a bolt.

- A bolt can be modeled as a single CBEAM or CBAR, or as several.
- The material properties and cross section of the CBEAM or CBAR elements should be representative of the corresponding bolt.
- The grids connected to CBEAM or CBAR elements representing bolts should be unconstrained.

BOLT bulk entries are used to define CBEAM or CBAR elements as bolts when ETYPE = 1.

- A single BOLT bulk entry should be created for each bolt incidence.
- If a bolt is modeled with multiple CBEAM or CBAR elements, only one of the elements needs to be included in a BOLT bulk entry. If additional CBEAM and CBAR elements are included in the BOLT bulk entry, the first element listed is used and all the other elements are ignored.

BOLTFOR bulk entries are used to assign bolt preload forces to bolts.

- At least one BOLTFOR bulk entry is required for each bolt preload force level.

BOLTLD bulk entries are used to define combinations of BOLTFOR bulk entries and scale the corresponding bolt preload forces.

- The use of BOLTLD bulk entries is optional. BOLTLD bulk entries provide flexibility in creating combinations of bolts and bolt preloads.

BOLTLD case control commands are used to select BOLTLD or BOLTFOR bulk entries to include in either the global case or subcases.

- Use a BOLTLD case control command before the subcases to select a BOLTLD bulk entry or BOLTFOR bulk entries to include in the global case.
- Use a BOLTLD case control command inside the subcases to select a BOLTLD bulk entry or BOLTFOR bulk entries to include in the subcases.

The BOLTFAC parameter is the stiffness reduction factor.

- For most cases, the default value of 1×10^7 is appropriate.
- For cases where the bolts are very stiff relative to the bolted medium, the BOLTFAC parameter can be increased to 1×10^{10} or so.
- For cases where constraints are sufficient to equilibrate the bolted medium if the bolts were not present, the BOLTFAC parameter can be arbitrarily large.
- For other cases, caution should be exercised when specifying the BOLTFAC parameter. Inappropriately large values can make the stiffness matrix nearly singular and potentially cause numerical problems.

RBE2 spider elements are typically used to connect the end grids of CBEAM or CBAR elements that represent bolts to the surrounding mesh.

Modeling with Solid Elements

The structure of an input file for modeling bolts with solid elements is identical to that for modeling bolts with line elements except that:

- A mesh of CHEXA, CPENTA, and CTETRA elements are used to represent a bolt. Create the solid element mesh such that at an intermediate position along the bolt axis the element edges and element faces of the mesh form a cross section through the bolt. As a best practice, make the cross section planar and normal to the axis of the bolt. Doing so will facilitate interpretation of the results. The cross section of the mesh and the material properties associated with the mesh should also be representative of the corresponding bolt.
- ETYPE = 2 should be specified on the BOLT bulk entries. When ETYPE = 2, the fields on the BOLT entry are used to define the axis of the bolt and list the grid points that lie in a cross section of the bolt.

Output

Use a FORCE case control command to obtain element force output for the CBEAM and CBAR elements that represent bolts. Use GPFORCE and MPCFORCES case control commands to obtain force output for bolts modeled with solid elements.

Subcases

Subcases can be used to examine different combinations of bolts, bolt preload forces, and service loads, among others.

- Use the BOLTLD case control command to select a BOLTLD bulk entry or BOLTFOR bulk entries to include in a subcase. All BOLTFOR bulk entries used in a subcase should have the same SID.
- For the special case where the sparse solver is used, the model does not contain contact, and the keyword SCRATCH = YES, the BOLTLD case control commands in the subcases must be ordered. For example, if a model contains three subcases with BOLTLD = 1 used twice and BOLTLD = 2 used once, the BOLTLD case control commands should be ordered with:
 - o The first and second subcases containing BOLTLD = 1 and the third subcase containing BOLTLD = 2.

- o The first subcase containing BOLTLD = 2 and the second and third subcases containing BOLTLD = 1.

Otherwise, the run will terminate without results. If the iterative solver is used or contact is used or the keyword SCRATCH = NO, ordering the subcases in this way is not necessary, but is recommended. Doing so minimizes the number of matrix decompositions required during the solution.

- Subcases without BOLTLD case control commands should be placed at either the beginning or end of the subcases.

Superelements and Bolt Preloads

Models with bolt preload can contain superelements. However, the elements used to define the bolts along with the bolt preload forces must be in the residual.

Contact and Bolt Preloads

When a contact definition exists, the contact conditions are included in both solutions. To decrease the solution time, the second solution begins with the contact status from the end of the first solution. As a result, the contact element creation steps and some of the initial contact iterations are avoided in the second solution.

Enforced Displacements and Bolt Preloads

Enforced displacements defined with non-zero SPC or SPCD* bulk entries can be included with a bolt preload. As described in the [Bolt Preload introduction](#), when a bolt preload is defined, the software automatically solves the model twice; the bolt preload solution step and the consecutive solution step. The software behaviour is different during these solution steps for the following input scenarios.

* NOTE: When an SPCD is defined with a bolt preload, system cell 581 must be set to 1 to override an error trap that exists in the software. Also note the general, NX Nastran requirement that DOF included on an SPCD must also be fixed on an SPC or SPC1 bulk entry, and selected by the SPC case control.

- Contact is not defined. A bolt preload and non-zero SPC or SPCD* entries are defined.

When contact conditions are not included with the bolt preload and enforced displacements, for example, glue conditions are defined at the bolted interface, the bolt preload solution step is performed with all SPC and SPCD DOF fixed, including the DOF where non-zero SPC or SPCD specifications are defined. In other words, enforced displacements are not applied during the bolt preload solution step.

The resulting preload is then carried forward for the consecutive solution step, which includes the service loads and the enforced displacements.

Case control example without contact conditions:

```
...
$CASE CONTROL
SUBCASE 1
SPC=1 (Selects fixed*, and non-zero SPC entries)
LOAD=2 (Selects service loads including SPCD* entries)
BOLTLD=4 (Selects bolt preload)
...
```

- Contact, a bolt preload, and non-zero SPCD* entry are defined.

When the bolt preload, the SPC (fixed DOF), the service loads (including the SPCD), and contact are all selected in a single subcase, the bolt preload solution step includes the enforced displacements (non-zero SPCD specifications). In most instances, this is not desired.

Single subcase case control example with undesired results:

```
...
$CASE CONTROL
SUBCASE 1
SPC=1 (Selects fixed* DOF)
LOAD=2 (Selects service loads including SPCD* entries)
BOLTLD=4 (Selects bolt preload)
BCSET=3 (Selects contact conditions)
...
```

To apply the non-zero SPCD definition for the consecutive solution step only, two subcases are required. The bolt preload, the contact set, and the SPC set are defined globally (above the subcases). The first subcase can be left empty, but must be defined. The second subcase selects the service loads, which includes the SPCD entry.

Two subcase case control example with desired results:

```
...
$CASE CONTROL
BOLTLD=4 (Selects bolt preload for all subcases)
BCSET=3 (Selects contact conditions for all subcases)
SPC=1 (Selects fixed* DOF only for all subcases)
SUBCASE 1
SUBCASE 2
LOAD=2 (Selects service loads, which includes the SPCD* entries)
...
```

- Contact, a bolt preload, and a non-zero SPC entry are defined.

The software always applies the non-zero SPC definition for both the bolt preload solution step, and the consecutive solution, regardless of the number of subcases. Since this is likely undesired behaviour, it is recommended that you use the two subcase option with the SPCD entry described above.

Bolt Preload Example — Line Element Model

A structure contains two bolts. One bolt has a M10 x 1.5 thread and the other has a M20 x 2.5 thread. Both bolts are modeled using CBAR elements. Four subcases are considered:

- Bolt preload forces at 70% of proof strength and the service loads excluded.
- Bolt preload forces at 70% of proof strength and the service loads included.
- Bolt preload forces at 80% of proof strength and the service loads excluded.
- Bolt preload forces at 80% of proof strength and the service loads included.

The following table contains bolt and proof strength data for use in the model. The equivalent radius is calculated from the tensile stress area.

Bolt size	Grade	Tensile stress area (mm²)	Equivalent radius (mm)	Proof strength (MPa)	Preload force at 70% of proof strength (kN)	Preload force at 80% of proof strength (kN)	Preload force at 100% of proof strength (kN)
M10 x 1.5	5.8	58	4.30	380	15.4	17.6	22.0
M20 x 2.5	5.8	245	8.83	380	65.2	74.5	93.1

Three input files are provided that model this situation:

- The first input file does not include a BOLTLD bulk entry. In this case, the BOLTLD case control commands select the BOLTFOR bulk entries having the same SID.
- The second input file includes BOLTLD bulk entries. In this case, BOLTLD case control commands select the BOLTLD bulk entry having the same SID.
- The third input file demonstrates how both BOLTLD bulk entries and BOLTFOR bulk entries can be selected by BOLTLD case control commands in the same input file. In this case, the BOLTLD case control commands in two subcases select a BOLTLD bulk entry and the BOLTLD case control commands in the other two subcases select BOLTFOR bulk entries.

Input file without BOLTLD bulk entries

```

SOL 101
.....
$   SUBCASES
$
SUBCASE = 1
  SUBTITLE   = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE   = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE   = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE   = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK
.....
.....
CBAR      2237      2      8001      8000      1.0      0.0      0.0
CBAR      2238      3      8003      8002      1.0      0.0      0.0
$
$   BOLT ENTRIES
$
BOLT      1      1      2237
BOLT      2      1      2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR    1  15.4+3      1
BOLTFOR    1  65.2+3      2
BOLTFOR    2  17.6+3      1
BOLTFOR    2  74.5+3      2
.....
.....
ENDDATA

```

Input file with BOLTLD bulk entries

```

SOL 101
.....
.....
$   SUBCASES
$
SUBCASE = 1
  SUBTITLE   = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE   = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE   = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE   = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK
.....
.....
CBAR      2237      2      8001      8000      1.0      0.0      0.0
CBAR      2238      3      8003      8002      1.0      0.0      0.0
$
$   BOLT ENTRIES
$
BOLT      1      1      2237
BOLT      2      1      2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR    1  22.0+3      1
BOLTFOR    2  93.1+3      2
$
$   SCALE BOLT PRELOAD ENTRIES
$
BOLTLD      1      0.70      1.0      1      1.0      2
BOLTLD      2      0.80      1.0      1      1.0      2
.....
.....
ENDDATA

```

Input file with BOLTLD case control commands selecting both BOLTLD and BOLTFOR bulk entries

```

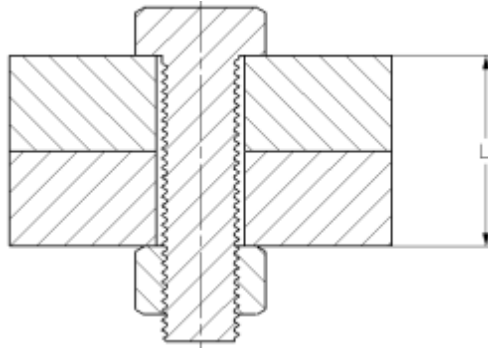
SOL 101
.....
.....
$   SUBCASES
$
SUBCASE = 1
  SUBTITLE   = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE   = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE   = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE   = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK
.....
.....
CBAR      2237      2      8001      8000      1.0      0.0      0.0
CBAR      2238      3      8003      8002      1.0      0.0      0.0
$
$   BOLT ENTRIES
$
BOLT      1      1      2237
BOLT      2      1      2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR   2  17.6+3      1
BOLTFOR   2  74.5+3      2
BOLTFOR   3  22.0+3      1
BOLTFOR   3  93.1+3      2
$
$   SCALE BOLT PRELOAD ENTRIES
$
BOLTLD      1      0.70      1.0      3
.....
.....
ENDDATA

```

Other sample input files are available at *nastran_installation\nxn\nast\tpl*. Files with bolt preload are typically named *boltld*.dat*.

Bolt Preload Formulation

Figure 8–21 depicts a bolted connection where L denotes the undeformed thickness of the medium being bolted together. Suppose the bolt is modeled using a CBEAM or CBAR element that is positioned along the centerline of the actual bolt and has an undeformed length L . Typically, the element representing the bolt is connected to the mesh of the bolted medium using RBE2 spider elements.

**Figure 8-21.**

Because the bolt head is in static equilibrium, a free body diagram of the bolt head gives

$$F_b - F_m = 0$$

where F_b denotes the force the body of the bolt exerts on the bolt head and F_m denotes the force the bolted medium exerts on the bolt head.

In the absence of service loads, these forces arise exclusively because of bolt preload. Therefore

$$F_b = F_m = F_p$$

where F_p denotes bolt preload force.

When the bolted joint is tightened, the loaded portion of the bolt elongates and the bolted medium compresses. However, as the joint is tightened, the actual length of the bolt under load decreases. Therefore, the undeformed length of the bolt L defined by the model geometry is larger than the actual length of the bolt under load. The difference in length is termed the preload length and is denoted as Δ_p .

The method outlined below accounts for the difference in length by applying an appropriately sized compressive force $F_{applied}$ to the ends of the element representing the bolt. An expression for this force can be obtained by first noting that

$$\Delta_p = \Delta_b + \Delta_m$$

Equation 8-34.

where Δ_b is the elongation of the bolt due to the bolt preload force, and Δ_m is the compression of the bolted medium due to the bolt preload force.

If the elongation of the bolt and compression of the bolted medium are small in comparison to L , the elongation of the bolt due to the bolt preload force is given by

$$\Delta_b = \frac{F_p L}{AE}$$

Equation 8-35.

where A and E are the tensile stress area and modulus of the bolt, respectively.

To find Δ_m , the software automatically solves the model with:

- The bolt preload force F_p applied compressively to the ends of the bolt.
- The bolt axial stiffness reduced by the BOLFACT parameter.
- The service loads excluded from the solution.

Because the bolt is extremely compliant in the axial direction during this solution, nearly all of the preload force compresses the bolted medium. As a result, the nodal deflections at each end of the bolt (U_i and U_j) due solely to bolt preload are obtained.

Because Δ_m is a positive quantity when the bolted medium is compressed and the positive sense for nodal deflections is directed from node i to node j

$$\Delta_m = U_i - U_j$$

Equation 8-36.

Substituting Equation 8-35 and Equation 8-36 into Equation 8-34 gives

$$\Delta_p = \frac{F_p L}{AE} + (U_i - U_j)$$

Equation 8-37.

Recalling that the force producing this deflection is denoted by $F_{applied}$, we have

$$F_{applied} = \left(\frac{AE}{L} \right) \Delta_p$$

Equation 8-38.

Substituting Equation 8-37 into Equation 8-38 yields

$$F_{applied} = F_p + \left(\frac{AE}{L} \right) (U_i - U_j)$$

Equation 8-39.

The software calculates $F_{applied}$ and then solves the model a second time with:

- $F_{applied}$ applied compressively to each end of the bolt.
- The bolt axial stiffness not reduced.
- The service loads optionally included in the solution.

Therefore, the second solution of the run predicts the stress state resulting from both the bolt preload and, if applicable, the service loads.

Frequently, a model contains multiple bolts. These bolts might have a variety of sizes, and be fabricated from different materials, and have differing preload forces, and connect mediums of various thickness and material type. For such a case, all the bolts are included during the first solution with:

- The appropriate preload forces applied compressively to each bolt.
- The axial stiffness of each bolt reduced by either a factor specified using the BOLTFAC_T parameter or the default value of 1×10^7 .
- The service loads excluded from the solution.

The software then calculates $F_{applied}$ for each bolt from Equation 8–39 using F_p , A , E , L , U_i and U_j corresponding to each bolt. During the second solution:

- Each $F_{applied}$ is applied compressively to the appropriate bolt.
- The axial stiffness of each bolt is not reduced.
- The service loads are optionally included in the solution.

Therefore, the second solution of the run predicts the stress state resulting from both the bolt preload and, if applicable, the service loads. In so doing, the software accounts for interactions between the stress fields arising from each bolt preload force and, if applicable, the service loads.

When a bolt is modeled with CHEXA, CPENTA, and CTETRA elements the formulation is different. The software splits the bolt mesh by:

- Duplicating each grid point listed on the BOLT entry. The grid IDs for the duplicated grid points start at the highest user-defined grid ID in the model plus one and continue sequentially higher. The duplicate grid points are written to the .op2 file and are visible when post-processing results.
- Modifying the element connectivity on one side of the cross section to use the duplicate grid points. The modified connectivity is written to the .op2 file and is visible when post-processing results.

The software adds a compliant spring between each pair of coincident grid points. The springs have stiffness in the axial direction of the bolt. The spring stiffness is calculated from:

$$EA / (L \times \text{BOLTFAC}_T \times N)$$

where E is the modulus of the bolt material, A is the projected area of the bolt cross section, L is the undeformed length of the bolt, and N is the number of grid points selected on the BOLT entry. The value for BOLTFAC_T is determined from the BOLTFAC_T parameter. For additional information on the BOLTFAC_T parameter, see the *NX Nastran Quick Reference Guide*.

The projected area of the cross section is calculated using the grid points listed on the BOLT entry, and the bolt axis definition on the BOLT entry (the CSID and IDIR fields). This area is used to apply the equivalent bolt preload as two equal and opposite surface tractions in the bolt axial direction.

A solution is run with only these surface tractions applied. The deflection at the bolt cross section due to only bolt preload is obtained. These deflections are then used to calculate the initial strain in the bolt.

The software solves the model a second time with:

- The initial strain applied.
- The springs removed.

- The service loads applied if applicable.

8.16 External Force Fields

You can include forces from an external force field in your NX Nastran model. Although this capability could be adapted to a wide variety of problems, the initial application is to structural problems where electromagnetic fields are present. NX Nastran can convert the electromagnetic field surface loads computed by a third-party electromagnetic simulation product like MAXWELL, to NX Nastran structural loads. You can then examine the mechanical response of a structural component to an electromagnetic field both alone and in combination with other structural loads. This capability is useful for analyzing structural components in motors or other electromechanical devices.

External field surface loads are the components of force that act at discrete points on the free faces of the external field simulation model. These discrete points are referred to as *external force points*. *Surface patches* are either planar surfaces or cylindrical surfaces defined on free faces of the NX Nastran model that correspond to free faces of the external field simulation model. You define surface patches by selecting sets of structural grids in the NX Nastran model.

NX Nastran uses a surface spline technique to convert the external field surface loads acting at external force points that you select to mechanically equivalent structural loads acting at structural grids on the surface patch. The surface spline technique allows you to select external force points whose locations do not exactly coincide with the locations of the structural grids that define the surface patch. However, to obtain the most accurate surface load to structural load conversion, the projections of all the selected external force points in the direction normal to the surface patch should fall within the envelope of the surface patch.

The surface spline technique used by NX Nastran is also used in aeroelastic analysis. For more information regarding the surface spline technique, see [“Interconnection of the Structure with Aerodynamics”](#) in the *NX Nastran Aeroelastic Analysis User’s Guide*.

NX Nastran can convert the surface loads from an external field solution to structural loads for the linear static (SOL 101), normal modes (SOL 103), direct frequency response (SOL 108) solution sequences, and for a rotor dynamic analysis with the direct complex eigenvalue solution sequence (SOL 107). For a linear static solution, the converted surface loads represent static loads that contribute to the overall loading of the structural component. For a normal modes solution or a rotor dynamic analysis with the direct complex eigenvalue solution, the converted surface loads represent static loads that prestress the structural component. For a direct frequency response solution, the converted surface loads represent scaling factors for the dynamic loads that excite the structural component.

EFPOINTS and EFFORCE Matrices

For NX Nastran to convert surface loads from an external field solution to NX Nastran structural loads, you must use direct matrix input (DMI) to define an external force points (EFPOINTS) matrix and a corresponding external field force (EFFORCE) matrix for each surface patch.

- EFPOINTS matrices contain the spatial coordinates of external force points that you select. To follow the geometry conveniently, the coordinate basis for this data can be either rectangular or cylindrical.

- EFFORCE matrices contain external field surface loads that act at external force points defined in the corresponding EFPOINTS matrices. The coordinate basis of these force components must be rectangular.

For information on how to create EFPOINTS and EFFORCE matrices, see *Creating an EFPOINTS Matrix* and *Creating an EFFORCE Matrix*.

NX Nastran Input Requirements

A single NX Nastran input file and solution is required for a linear static (SOL 101) or direct frequency response (SOL 108) analysis. In the case control section of the input file, you must define a separate subcase for each surface patch for which you want to convert the external field surface loads to structural loads. Each of these subcases must include:

- A P2G case control command to select the applicable EFPOINTS and EFFORCE matrices from the bulk section.

```
P2G = EFPOINTS, EFFORCE
```

- A SET case control command to select structural grids.

```
SET 3 = 25, 28, 30 THRU 42
```

- A PARTN case control command to define the set of selected structural grids as a surface patch.

```
PARTN 3
```

- An EFLOAD case control command to request that the external field solution surface loads are converted to structural loads and to define the direction normal to the surface patch.

```
EFLOAD (NDIR = 1, SCID = 7) = YES
```

The final subcase in the input file can only include output requests and other structural loads. You cannot include an EFLOAD case control command in the final subcase. Thus, for the case where external field surface loads are converted to structural loads for only one surface patch, two subcases are required.

In the bulk section of the input file, you must include either DMI bulk entries defining the EFPOINTS and EFFORCE matrices or INCLUDE bulk entries that reference external files that contain these DMI bulk entries. For instructions on how to use DMI bulk entries to define EFPOINTS and EFFORCE matrices, see *Creating an EFPOINTS Matrix* and *Creating an EFFORCE Matrix*.

Normal modes (SOL 103)

For a normal modes (SOL 103) analysis, external field surface loads represent a static loading that prestresses the structure. To account for static prestress due to external field surface loads, two solves are required. The first solve is a linear static (SOL 101) analysis where the external field surface loads are converted to structural loads. The input file requirements for this solve are identical to those outlined previously. During this solve, the presence of an EFLOAD case control command signals NX Nastran to automatically write a punch (.pch) file that contains the structural loads that were converted from the external field surface loads.

The second solve is a normal modes (SOL 103) analysis. During this solution, the structural loads converted from the external field surface loads during the linear static analysis are used to statically prestress the structure. The input file is similar to that used to perform a normal modes analysis in which differential stiffness is computed from static loads, except that you must include:

- EFLOAD = YES and no more in the first subcase, which is the static subcase. In the second subcase, which is the normal modes subcase, you must include STATSUB = 1 and can optionally define output requests and so on. The second subcase cannot contain an EFLOAD case control command.

```
SUBCASE 1
EFLOAD = YES
SUBCASE 2
STATSUB = 1
.....
.....
```

- An INCLUDE bulk entry that references the punch file created during the linear static solve. NX Nastran then uses the structural loads in the punch file as static preload forces during the normal modes solution.

```
INCLUDE 'filename.pch'
```

Because the EFPOINTS and EFFORCE matrices are not used during the normal modes solution, the DMI bulk entries that define them can optionally be omitted from the normal modes solution input file.

Rotor dynamic analysis (SOL 107)

An external force field can be included in a rotor dynamic analysis with the direct complex eigenvalue solution sequence (SOL 107). The three solution steps are described as follows.

Run 1:

A linear static analysis (SOL 101) is executed with the standard external force inputs. The result of this analysis is a punch file that contains the combined load vector representing the external field. This simplified input example shows the external force field inputs required for this step.

The complete input file **rotefa.dat** is available in *install_directory\nxnr\nast\tp\l* to demonstrate this step.

```
SOL 101
CEND
...
$* CASE CONTROL
...
Case control inputs for the external field
SET 1 = 5, 6, 7, 45, 46, 47
PARTN 1
p2g = ep2, ef2
efload (nDIR=3, SCID=9)=yes
....
BEGIN BULK

This coordinate system is referenced by the SCID case control.
CORD2C, 9, , 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, +C1
+C1, 20.0, 0.0, 1.0
....
The EP2 matrix defines locations in the force field.
dmi,ep2,0,2,1,0,,3,2
dmi,ep2,1,1, 20.0, 180.0, 80.0
dmi,ep2,2,1, 20.0, 270.0, 80.0
....
The EF2 matrix defines the force components
at the locations defined in the EP2 matrix.
dmi,ef2,0,2,1,0,,3,2
dmi,ef2,1,1, -100.0, 0.0, 0.0
dmi,ef2,2,1, 0.0, -100.0, 0.0
```

```
....
ENDDATA
```

Run 2:

This SOL 107 run uses the force matrix stored in the punch file from run 1 to compute the differential stiffness matrix due to the external field loads. Run 2 stores the differential stiffness matrix into a new punch file, and then exits. It is an incomplete run since it does not compute the complex eigenvalues. This computation will occur in run 3.

The input file for this run is defined with two subcases. Subcase 1 contains an EFLOAD=yes case control command with no other data. Subcase 2 contains a STATSUB entry referring to the first subcase. The bulk data section must include the punch file generated by run 1. The execution of this abbreviated run is triggered by the presence of the EFLOAD case control command in the first subcase.

The complete input file **rotefb.dat** is available in *install_directory\nxnr\nast\tp* to demonstrate this step.

```
SOL 107
CEND
...
$* CASE CONTROL
...
The EFLOAD command is defined in subcase 1
SUBCASE 1
efload = yes
...
The STATSUB command is defined in subcase 2
SUBCASE 2
statsub = 1
...
BEGIN BULK
...
The punch file generated by run 1 is included here
include 'OUTDIR:rotefa.pch'
...
ENDDATA
```

Run 3:

This run is the complete rotor dynamic analysis, and includes the differential stiffness matrix written to the punch file created in run 2. Your SOL 107 input file for this run will include any typical rotor dynamic inputs, it will include EFLOAD=yes in the case control, and the contents of the run 2 punch file in the bulk data section. The EFLOAD command will trigger the software to add the differential stiffness matrix to the standard stiffness matrix before the rotor dynamic matrices are generated.

The complete input file **rotefc.dat** is available in *install_directory\nxnr\nast\tp* to demonstrate this step.

```
SOL 107
$
CEND
...
$* CASE CONTROL
The EFLOAD command will trigger the software to
include the differential stiffness matrix
...
```

```

efload = yes
...
spc = 1
cmethod = 1
rmethod = 2
...
OUTPUT
DISPLACEMENT(PRINT) = ALL
...
BEGIN BULK
...
Typical rotor dynamic analysis inputs
EIGC      1      CLAN      5
ROTORD    2      0.0      250.0  3      ROT      0.0      RPM      HZ      +ROT0
$          ZSTEIN  ORBEPS  ROTPRT
+ROT0     NO      1.0E-5  3
$          RID1   RSET1   RSPEED1 RCORD1  W3-1    W4-1    RFORCE1
+ROT1     1      3      1.0      1      0.      0.
...
ROTORG    3      1      THRU    64
...
The punch file generated by run 2 is included here
include 'OUTDIR:rotefb.pch'
...
ENDDATA

```

Using the EFLOAD Case Control Command

The EFLOAD case control command has the following format:

EFLOAD (NDIR = *n*, SCID = *m*) = YES/NO

If you specify the YES describer, the external field surface loads are converted to structural loads. Conversion does not occur if you specify the NO describer.

The use of the describers NDIR = *n* and SCID = *m* depends on the shape and orientation of the surface patch that is defined in the same subcase. You use the describer NDIR = *n* to specify the normal to the surface patch. You use the describer SCID = *m* to specify that the surface patch is cylindrical.

- For a planar surface patch lying on either an xy-, yz-, or zx-plane, you should specify the describer NDIR = *n* and omit the describer SCID = *m*. If the surface patch normal is the z-direction, the describer NDIR = *n* can be omitted because *n* = 1 is the default.
- For a cylindrical surface patch or a planar surface patch lying in either an rθ- or rz-plane, you should specify both the describer NDIR = *n* and the describer SCID = *m*. Regardless of the local coordinate system referenced by describer SCID = *m*, NX Nastran uses a cylindrical coordinate system whose:
 - o Origin coincides with that of the basic coordinate system.
 - o Cylindrical axis coincides with the z-axis of the basic coordinate system.
 - o Tangential coordinate, θ , is measured from the x-axis of the basic coordinate system. ($0 \leq \theta \leq 360$ degrees)

Table 8-3 lists the allowable values for *n* and the corresponding normal directions in rectangular and cylindrical coordinate systems.

Table 8-3. Normal Direction for Surface Patches

<i>n</i>	Rectangular CSYS	Cylindrical CSYS
1 (default)	z	z
2	y	θ
3	x	r

Creating an EFPOINTS Matrix

An external force points (EFPOINTS) matrix specifies the spatial coordinates of external force points corresponding to a surface patch. You can express the spatial coordinates in terms of either a rectangular or cylindrical coordinate system.

- For a rectangular coordinate system, the spatial coordinates must be expressed in terms of the basic coordinate system of the NX Nastran model.
- For a cylindrical coordinate system, the spatial coordinates must be expressed in terms of a cylindrical system whose:
 - Origin coincides with that of the basic coordinate system of the NX Nastran model.
 - Cylindrical axis coincides with the z-axis of the basic coordinate system of the NX Nastran model.
 - Tangential coordinate, θ , is measured from the x-axis of the basic coordinate system of the NX Nastran model. ($0 \leq \theta \leq 360$ degrees)

The first row of the DMI bulk entry specifies parameters for the EFPOINTS matrix. For real, single precision input of spatial coordinates, the format is as follows:

```
dm1,name,0,2,1,0,,3,N
```

where the EFPOINTS matrix can be named as you like and *N* is the number of external force points. The remaining rows in the DMI bulk entry contain the spatial coordinates. One row is required for each external force point. The format is as follows:

```
dm1,name,1,1,X1,Y1,Z1
.....
.....
dm1,name,n,1,Xi,Yi,Zi
.....
.....
dm1,name,N,1,XN,YN,ZN
```

where *name* must correspond with the first DMI entry, and X_i , Y_i , and Z_i are the spatial coordinates of the *i*th external force point in the x-, y-, and z-direction, respectively.

For cylindrical coordinates, the format is identical except X_i , Y_i , and Z_i are replaced by R_i , θ_i , and z_i , respectively. R_i , θ_i , and z_i are the spatial coordinates of the *i*th external force point in the r-, θ -, and z-direction, respectively.

Creating an EFFORCE Matrix

An external field force (EFFORCE) matrix specifies the components of force at the external force points of the corresponding EFPOINTS matrix. The force components must be expressed in terms of

the basic coordinate system of the NX Nastran model. The first row of the DMI bulk entry specifies parameters for the EFFORCE matrix. For real, single precision input of force components, the format is as follows:

```
dm1,name,0,2,1,0,,3,N
```

where the EFFORCE matrix can also be named as you like and N is the number of external force points in the corresponding EFPOINTS matrix. The remaining rows of the DMI bulk entry contain the components of force. One row is required for each external force point. The format is as follows:

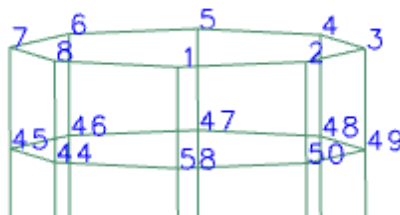
```
dm1,name,1,1,FX1,FY1,FZ1
.....
.....
dm1,name,n,1,FXi,FYi,FZi
.....
.....
dm1,name,N,1,FXN,FYN,FZN
```

where `name` must correspond with the first DMI entry, and FX_i , FY_i , and FZ_i are the components of force that act at the i^{th} external force point in the x-, y-, and z-direction, respectively.

Linear Static and Normal Modes Example

In this example, a linear static analysis is performed on a cylindrical tube that has one end fixed to ground and the other end subjected to an external force field that produces external field surface loads. Then, the converted external field surface loads from the linear static analysis are used to prestress the cylinder during a normal modes analysis.

The cylinder is modeled with CQUAD4 elements. The bottom edge of the NX Nastran model is fixed to ground. The other end of the tube is subjected to an external force field. The external field surface loads act on a ring of external force points that lie midway between the top two rings of structural grids in the NX Nastran model.



Structural grids used to define the surface patch

The relevant portions of the NX Nastran input file are:

```
ASSIGN PUNCH = static_loads.pch,NEW,UNIT=7
$
SOL 101
$
CEND
$
$ CASE CONTROL
$
ECHO = NONE
SPC = 1
$
SUBCASE 1
$
$ The P2G case control command selects the EFPOINTS and EFFORCE matrices from the bulk
```

```

$ section. The EFPOINTS matrix for this surface patch is named 'ep1'. The EFFORCE matrix
$ for this surface patch is named 'ef1'.
$
P2G = ep1, ef1
$
$ The SET and PARTN case control commands define the surface patch
$
SET 1 = 1 thru 8, 44 thru 50, 58
PARTN 1
$
$ The EFLOAD case control command requests that external field surface loads be converted
$ to structural loads. SCID=9 specifies that the surface patch is a cylindrical surface.
$ NDIR=3 specifies that the normal to the surface patch is in the radial direction.
$
$
EFLOAD (NDIR=3, SCID=9) = YES
$
SUBCASE 2
OUTPUT
DISPLACEMENT(PRINT) = ALL
STRESS(PRINT) = ALL
$
BEGIN BULK
$
$ The CORD2C bulk entry specifies a cylindrical local coordinate system used to specify
$ the structural grids.
$
CORD2C, 9, , 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, +C1
+C1, 20.0, 0.0, 1.0
$
$ PARAM CARDS
$
PARAM      K6ROT      100.0
PARAM      POSTEXT     YES
PARAM      UNITSYS     MN-MM
$
$ GRID CARDS
$
GRID        1          9      20.07.016-15      80.0      0
.....
GRID        64         9      20.0      0.0      0.0      0
$
$ ELEMENT CARDS
$
CQUAD4      1          1          9      16      17      10
.....
CQUAD4      56         1          57      9      10      56
$
$ PROPERTY CARDS
$
PSHELL      1          1          1.8      1      1.0      1.8333333      0.0
$
$ MATERIAL CARDS
$
MAT1        1      2.0+5      .25 7.872-9
$
$ LOAD AND CONSTRAINT CARDS
$
SPC          1          51 123456      0.0
.....
SPC          1          64 123456      0.0
$
$ The EFPOINTS and EFFORCE matrices are constructed from the external field solution.
$ The following DMI bulk entries define an EFPOINTS matrix named 'ep1'.
$
dmi,ep1,0,2,1,0,,3,8

```

```

dmi,ep1,1,1, 20. , 0.0, 74.286
dmi,ep1,2,1, 20. , 45., 74.286
dmi,ep1,3,1, 20. , 90.0, 74.286
dmi,ep1,4,1, 20. , 135.0, 74.286
dmi,ep1,5,1, 20. , 180.0, 74.286
dmi,ep1,6,1, 20. , 225.0, 74.286
dmi,ep1,7,1, 20. , 270.0, 74.286
dmi,ep1,8,1, 20. , 315.0, 74.286
$
$ The following DMI bulk entries define an EFFORCE matrix named 'ef1'. Because external field
$ surface loads are always expressed relative to a rectangular coordinate basis, the DMI bulk
$ entries define an external field surface load of FX = 100, FY = 0, and FZ = 100 acting at
$ each external force point.
$
dmi,ef1,0,2,1,0,,3,8
dmi,ef1,1,1, 100.0, 0.0, 100.
dmi,ef1,2,1, 100.0, 0.0, 100.
dmi,ef1,3,1, 100.0, 0.0, 100.
dmi,ef1,4,1, 100.0, 0.0, 100.
dmi,ef1,5,1, 100.0, 0.0, 100.
dmi,ef1,6,1, 100.0, 0.0, 100.
dmi,ef1,7,1, 100.0, 0.0, 100.
dmi,ef1,8,1, 100.0, 0.0, 100.
$
ENDDATA

```

Note

The use of an ASSIGN statement to specify the name of the punch file that is written during the solve is optional.

A second run is used to perform a normal modes analysis on the cylinder. During this run, the converted external field surface loads are used to prestress the cylinder. The relevant components of the input file are:

```

SOL 103
$
CEND
$
$ CASE CONTROL
$
ECHO = NONE
SPC = 1
$
SUBCASE 1
$
EFLOAD = YES
$
SUBCASE 2
STATSUB = 1
METHOD = 99
OUTPUT
DISPLACEMENT(PRINT) = ALL
STRESS(PRINT) = ALL
$
BEGIN BULK
$
EIGRL, 99, , , 9
$
$ The CORD2C bulk entry specifies a cylindrical local coordinate system used to specify
$ the structural grids.
$
CORD2C, 9, , 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, +C1
+C1, 20.0, 0.0, 1.0

```

```

$
$ PARAM CARDS
$
PARAM      K6ROT    100.0
PARAM      POSTEXT   YES
PARAM      UNITSYS  MN-MM
$
$ GRID CARDS
$
GRID        1        9    20.07.016-15    80.0        0
.....
GRID        64        9    20.0        0.0        0.0        0
$
$ ELEMENT CARDS
$
CQUAD4       1        1        9        16        17        10
.....
CQUAD4       56        1        57        9        10        56
$
$ PROPERTY CARDS
$
PSHELL       1        1        1.8        1        1.0        1.8333333    0.0
$
$ MATERIAL CARDS
$
MAT1         1    2.0+5            .25 7.872-9
$
$ LOAD AND CONSTRAINT CARDS
$
SPC          1        51 123456    0.0
.....
SPC          1        64 123456    0.0
$
$ The linear static solve wrote a punch file that contains the preload forces at the structural
$ grids. The INCLUDE bulk entry references this file.
$
INCLUDE static_loads.pch
$
ENDDATA

```


Chapter 9: Constraints

- *Introduction to Constraints*
- *Single-point Constraints*
- *Automatically Applying Single-point Constraints*
- *Enforced Displacements at Grid Points (SPCD, SPC)*
- *Multipoint Constraints*
- *Rigid Body Supports*
- *Surface-to-Surface Gluing*
- *Linear Contact Using Constraints*

9.1 Introduction to Constraints

A constraint is the enforcement of a prescribed displacement (i.e., component of translation or rotation) on a grid point or points. There are two basic types of constraints in NX Nastran: single point constraints (SPCs) and multipoint constraints (MPCs).

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

The boundary conditions of a static structure (fixed, hinged, roller support, etc.) typically require that various degrees of freedom be constrained to zero displacement. For example, consider a grid point fixed in a rigid wall. All six displacement degrees of freedom—three translational directions and three rotational directions—must be constrained to zero to mathematically describe the fixed boundary condition.

Real world structures often don't have simple or ideal boundary conditions. Because a model's constraints greatly influences its response to loading, you must try to constrain your model as accurately as possible.

This chapter describes how you apply constraints. To understand how constraints are processed, you need to be familiar with the NX Nastran set notation and matrix operations.

9.2 Single-point Constraints

A single point constraint (SPC) applies a fixed value to a translational or rotational component at an individual grid point or to a scalar point. Common uses of single point constraints are to specify the boundary conditions of a structural model by fixing the appropriate degrees-of-freedom and to eliminate unwanted degrees-of-freedom with zero stiffness. Multiple sets of single point constraints can be provided in the Bulk Data Section, with selections made at execution time by using the subcase structure in the Case Control Section. This procedure is particularly useful in the solution of problems having one or more planes of symmetry.

You can use single-point constraints to:

- To constrain a structure to ground.
- To apply symmetric or antisymmetric boundary conditions by restraining the degrees of freedom that must have a zero value to satisfy symmetry or antisymmetry.
- To remove degrees of freedom that are not used in the structural analysis (i.e., are not connected to any structural elements or otherwise joined to the structure).
- To remove degrees of freedom that are very weakly coupled to the structure. This condition can occur, for example, to the rotations about the normal of a slightly curved shell. In this case, a judgment must be made whether to remove the degree of freedom using an SPC (in which case the structure may be over-constrained), or to leave it in the problem (in which case the stiffness matrix is nearly singular). A reasonable rule is to constrain the degree of freedom if its stiffness is less than 10^{-8} as large as the stiffness in another direction at the same grid point.

The elements connected to a grid point may not provide resistance to motion in certain directions, causing the stiffness matrix to be singular. Single point constraints are used to remove these degrees-of-freedom from the stiffness matrix.

For example, consider a planar structure composed of membrane and extensional elements. The translations normal to the plane and all three rotational degrees-of-freedom must be constrained since the corresponding stiffness matrix terms are all zero. If a grid point has a direction of zero stiffness, the single point constraint doesn't need to be exactly in that direction. It only needs to have a component in that direction. This allows the use of single point constraints for the removal of such singularities regardless of the orientation of the global coordinate system. Although the displacements will depend on the direction of the constraint, the internal forces will be unaffected.

Defining Single point Constraints in NX Nastran

There are several different ways that you can define single point constraints:

- You can define single point constraints on SPC, SPC1, SPCADD and SPCAX bulk data entries.
- You can define single point constraints on the GRID and GRDSET entries.
- You can use the PARAM,AUTOSPC,YES option to have the software automatically create SPCs to remove degrees of freedom that are either unconnected or very weakly coupled.

You can obtain structural reaction forces called single point constraint forces (SPCFs) at grid points constrained by SPCs. You can request printed output for single point forces in the Case Control section. The software prints all nonzero forces, whether they originate from SPCi entries, the PS field on GRID entries, or PARAM,AUTOSPC,YES.

When you apply a single-point constraint to remove a singularity, the restrained component of motion to doesn't need to be aligned exactly with the singular direction of motion (however, it is highly recommended). Consider the pair of co-linear pin-connected rods, shown in [Figure 9-1](#), that permit unrestrained motion at point G in any direction perpendicular to the axis of the rods.

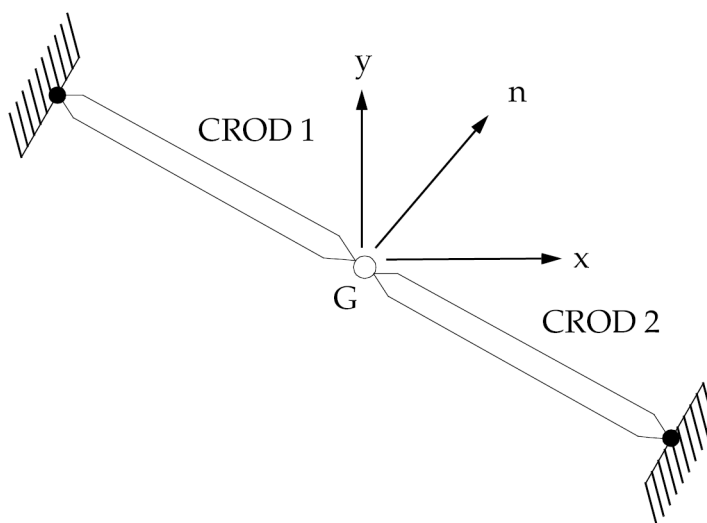


Figure 9-1. Pin-Connected Rods

After SPCs are applied to the translation in the z-direction and to all three components of rotation, unrestrained motion is still possible in the n-direction. An SPC in either the x- or the y-direction removes the remaining singularity without disturbing the axial forces in the rods (in this special case). Note, however, that the motion at point G has a zero component in the restrained direction, which may produce incorrect results for the displacements. For this reason, you should select a local coordinate system whose coordinate axes align with the singular components of motion.

Using GRID Entries

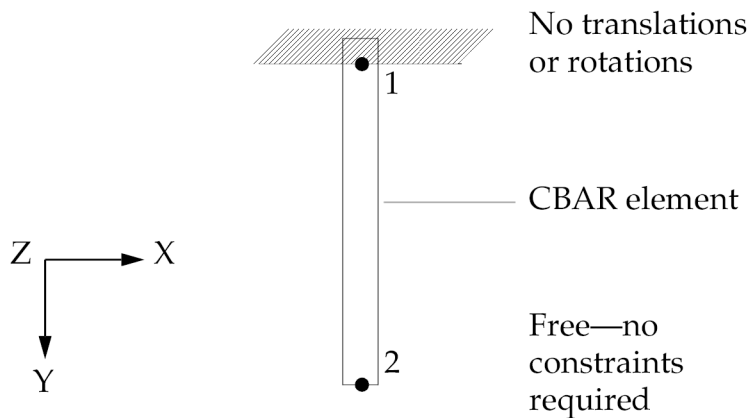
You can also define single point constraints using the PS option (field 8) on the GRID and GRDSET bulk data entries. When you define constraints this way, the constraints are permanent and are constrained for all subcases; you can't select them in the Case Control section. Their primary use, in addition to applying permanent supports, is to remove unconnected or very weakly connected degrees of freedom.

The default value for enforced displacement on points that you constrain on GRID entries is zero. You can override the default value at the subcase level with SPC entries.

See also

- “Specifying Permanent Single Point Constraints (Field 8)” in the *NX Nastran User's Guide*
- “GRID” in the *NX Nastran Quick Reference Guide*
- “GRDSET” in the *NX Nastran Quick Reference Guide*

Grid Example 1: Cantilever Beam



In this example, we constrained Grid 1 in all six DOFs using field 8 (permanent single point constraint) of the GRID Bulk Data entry:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.		123456		

Or, in free field format,

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

Note that with this option, you don't need to specify a constraint set in the Case Control section.

Using SPC and SPC1

You can define single point constraints on SPC, SPC1, SPCADD and SPCAX bulk data entries.

- The SPC entry is the most general way of specifying single point constraints.
- The SPC1 entry is a less general entry that is more convenient when several grid points have the same components constrained to a zero displacement.
- The SPCADD entry defines a union of single point constraint sets specified with SPC or SPC1 entries.
- The SPCAX entry is used only for specifying single point constraints in problems using conical shell elements.

You should use the SPC and SPC1 Bulk Data entries if you need to apply different sets of boundary conditions in different subcases. The constraints specified on the SPC and SPC1 entries belong to sets identified by set identification numbers (SIDs) that must be selected in the Case Control Section to be used. You can use the SPCADD entry in the Case Control section to merge individual SPC and SPC1 sets into combined sets.

The format of the SPC entry is as follows:

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

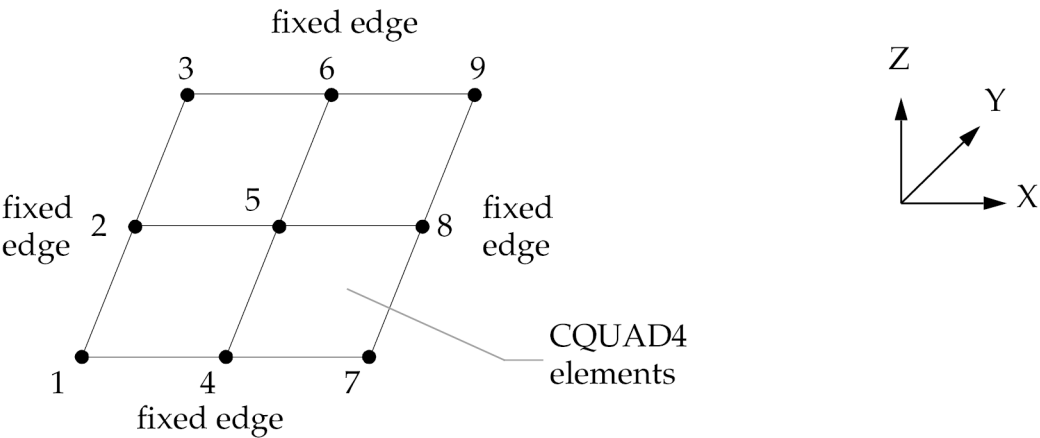
Field	Contents
SID	Identification number of the single-point constraint set.
Gi	Grid or scalar point identification number.
Ci	Component number.
Di	Value of enforced displacement for all degrees of freedom designated by Gi and Ci.

The SPC entry can also be used to specify a nonzero displacement constraint. The values specified in fields 5 and 8 are enforced displacements for the components specified in fields 4 and 7, respectively. When using an enforced constraint within multiple subcases, it is more efficient to use the SPCD entry.

See also

- “**SPC**” in the *NX Nastran Quick Reference Guide*
- “**SPC1**” in the *NX Nastran Quick Reference Guide*
- “**SPCADD**” in the *NX Nastran Quick Reference Guide*

SPC Example 1



Grids 1, 2, 3, 4, 6, 7, 8, and 9 must be constrained in all six DOFs. Assume that a single point constraint selection is defined in the Case Control Section (SPC = 42). Applying the same constraint on numerous individual GRID Bulk Data entries is not particularly convenient-an SPC1 entry is a better choice:

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

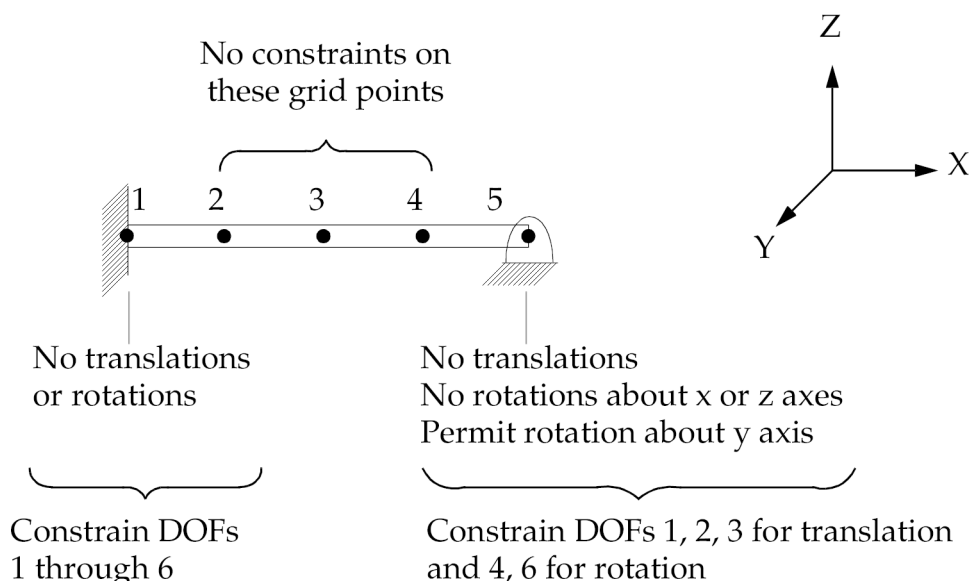
SPC1	42	123456	1	2	3	4	6	7	
	8	9							

Or, in free field format,

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

Note that if the “THRU” (alternate format) option were used (1 THRU 9), grid point 5 would have been incorrectly fixed, and no displacement would occur anywhere in the model.

SCP Example 2: Fixed Hinged Beam



Here, an SPC entry is used to constrain both ends of the beam. Assume that a single point constraint set selection is defined in the Case Control Section (SPC = 42). The values of enforced displacement (Di) are left blank since zero displacement is the default.

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

```
SPC      42      1      123456      5      12346
```

Or, in free field format,

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

9.3 Automatically Applying Single-point Constraints

NX Nastran automatically identifies and constrains singularities in the stiffness matrix using a parameter called AUTOSPC. When you add PARAM,AUTOSPC,YES to either the bulk data or case control sections of your input file, NX Nastran automatically removes degrees of freedom that are either unconnected or very weakly coupled to the finite element model.

With AUTOSPC, you use the PARAM,EPZERO option to specify the minimum stiffness ratio the software should use to determine a singularity. The stiffness ratio is the ratio of the stiffness in the weakest direction to the stiffness in the strongest direction at the grid point (considering all possible directions rather than just the coordinate directions). The software considers translational and rotational stiffness separately.

See also

- “AUTOSPC” in the *NX Nastran Quick Reference Guide*
- “EPZERO” in the *NX Nastran Quick Reference Guide*

Grid Point Singularity Processor

When you use the AUTOSPC parameter, the software uses the Grid Point Singularity Processor (GPSP) module to identify potential singularities. When you set the AUTOSPC parameter to YES, the GPSP constrains the degrees of freedom associated with the obvious singularities. The GPSP identifies local singularities in the stiffness matrix after the software processes all the SPC and MPC constraints in your model.

The software begins identifying local grid point singularities by examining the $3 \cdot 3$ matrices that represent the three translations and three rotations of each grid point. For each $3 \cdot 3$ matrix, the software performs an eigensolution to obtain the principal values (eigenvalues) and the principal directions (eigenvectors). The principal values are normalized to obtain a measure R_i as shown in Eq. 9-1:

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

$$R_i = 0.0 \quad \text{if } |K_{MAX}| = 0$$

$i = 1, 2, 3$

Equation 9-1.

K_i = the diagonal stiffness term corresponding to the i-th degree of freedom
 K_{MAX} = the maximum principal stiffness

Any principal value for which $|R_i| < \text{EPZERO}$ represents a singularity. The software examines the principal direction corresponding to the singularity and identifies the component with the largest ratio as the failed direction for the grid point. The software doesn't constrain any degrees of freedom that you specify as belonging to the a-set.

See also

- “[Understanding Sets and Matrix Operations](#)” in the *NX Nastran User's Guide* (for information on a-sets)
- “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#)” in the *NX Nastran User's Guide*

Understanding the Grid Point Singularity Table

When the GPSP examines the stiffness matrix for singularities at the grid point level, it lists any singularities in the Grid Point Singularity Table (GPST). The software prints the GPST in the .f06 file. The GPST lists all singular degrees-of-freedom, in the global coordinate system, and the ratio of stiffness between the softest and stiffest degree-of-freedom for the grid point. It also lists any degrees-of-freedom that the software automatically constrained.

You should always review the Grid Point Singularity Table carefully to understand which DOFs were singular and why. Automatic constraint generation can mask actual modeling errors, such as your forgetting to specify bending stiffness (via MID2 on the PSHELL entry).

The output includes a grid point singularity table with the following format:

- The TYPE specifies whether the failed degree of freedom is a component of a grid point (G) or a scalar point (S).
- The component of motion that failed. Constraints are applied in the global coordinate direction closest to the weakest direction as defined above.
- If possible, the failed degrees of freedom are placed in the sb-set if AUTOSPC is set to YES (the default for Solution 101). If AUTOSPC is set to NO, the singularity is identified but not constrained and is not placed in the sb-set.
- An asterisk (*) is printed at the end of the line and indicates that action was taken by NX Nastran.

To illustrate how failing to inspect the GPSP output can lead to a potential problem, consider the single CBAR element example shown in [Figure 9-2](#).

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

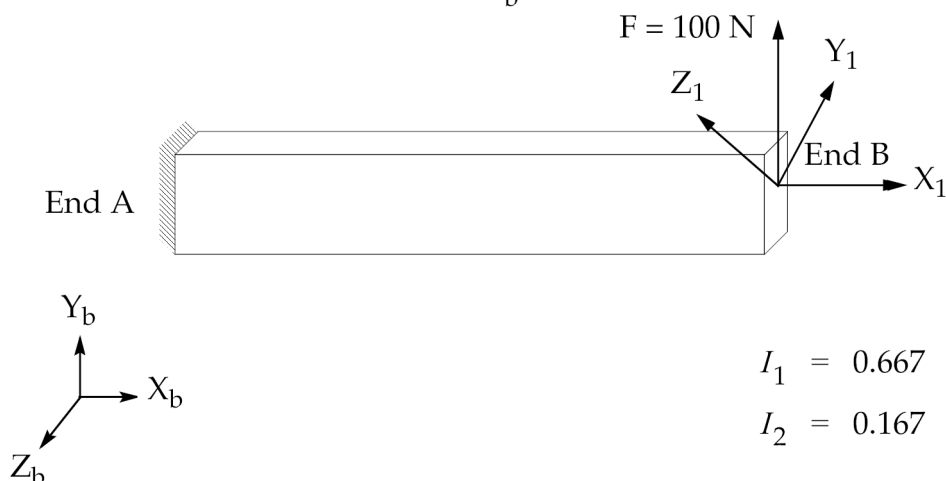


Figure 9-2. Single Element with Vertical Load

The CBAR element is fully constrained at end A and is loaded with a 100 N force in the Y_b -direction at end B. The input file for this model is shown in [Listing 9-4](#). Both grid points are defined in the basic coordinate system. The displacement coordinate system for grid point 1 is the basic system, but the displacement coordinate system for grid point 2 is in the local coordinate system 1. It is typical to define a local coordinate system when you need a displacement component in a specific direction.

A sample Grid Point Singularity Table is shown in [Figure 9-3](#).

GRID POINT SINGULARITY TABLE								
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET		NEW USET		
				EXCLUSIVE	UNION	EXCLUSIVE	UNION	
2	G	1	0.00E+00	B	F	SB	SB	*
2	G	3	0.00E+00	B	F	SB	SB	*
2	G	4	0.00E+00	B	F	SB	SB	*
2	G	5	0.00E+00	B	F	SB	SB	*
2	G	6	0.00E+00	B	F	SB	SB	*

Figure 9-3. Grid Point Singularity Table

In this table, grid point 2, the free end of the rod, failed (i.e. had no stiffness connection) in five of its six possible DOFs. DOF 2 (translation in the Y direction) of grid point 2 wasn't singular because it is attached to the extensional stiffness of the rod. CROD elements support extension and torsion, but in this example no value of J, the torsional constant, was provided in field 5 of the PROD element property entry. As a result, the CROD element had no torsional stiffness. The resulting unconnected DOF was detected and constrained by AUTOSPC.

```

$  FILENAME - BAR3A.DAT
ID    LINEAR,BAR3A
SOL    101
TIME    5
CEND
TITLE = BAR WITH I1 AND I2
SPC = 1
LOAD = 1
FORCE = ALL
DISP = ALL
BEGIN BULK
PARAM    POST    0
PARAM    AUTOSPC YES
CORD2R   1        0      0.0      0.0      0.0      0.0      .7071068.7071068+
        1.        0.0      0.0
$
GRID     1        0.0      0.0      0.0
GRID     2      10.      0.0      0.0      1
$
CBAR     1        1        1        2        1.        1.        0.0
$
SPC       1        1      123456  0.0
$
PBAR     1        1        1.        .667      .167
FORCE    1        2      100.      0.        1.        0.
MAT1     1        1.+7      .3
ENDDATA

```

Figure 9-4. Single Element with a Vertical Load

For this example, the displacement should be exclusively in the Y_b -direction. Since the displacement is computed and output in the CD coordinate system, you expect that the displacement components T2 (the Y_1 -direction) and T3 (the Z_1 -direction) for grid point 2 are the same for this model. The displacement and force results shown in [Figure 9-5](#) confirm this assumption.

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET	NEW USET				
2	G	4	0.00E+00	B	F	EXCLUSIVE SB	UNION S	*	
DISPLACEMENT VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	.0	.0		
2	G	.0	3.533767E-03	3.533767E-03	.0	-5.300650E-04	5.300650E-04		
FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL FORCE		TORQUE
	PLANE 1	PLANE 2	PLANE 1	PLANE 2	PLANE 1	PLANE 2	FORCE		
1	1.000000E+03	1.761739E-14	4.263256E-13	-4.944563E-14	1.000000E+02	6.706302E-15	.0		.0

Figure 9-5. Force and Displacement at Grid Point 2 with a Nonzero I2

However, suppose that when you construct the model, you know that the deflection is in plane 1 of the element only, so you leave the I2 field blank. The resulting grid point singularity table, element forces, and displacements are shown in **Figure 9-6**

GRID POINT SINGULARITY TABLE									
POINT ID	TYPE	FAILED DIRECTION		STIFFNESS RATIO	OLD USET		NEW USET		*
		2	4		B	F	SB	S	
2	G		2	0.00E+00	B	F	SB	S	*
2	G		4	0.00E+00	B	F	SB	S	*
2	G		5	0.00E+00	B	F	SB	S	*
DISPLACEMENT VECTOR									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
1	G	.0	.0	.0	.0	.0	.0		
2	G	.0	.0	7.067534E-03	.0	.0	1.060130E-03		
FORCES IN BAR ELEMENTS (CBAR)									
ELEMENT ID.	BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL FORCE		TORQUE
	PLANE 1	PLANE 2	PLANE 1	PLANE 2	PLANE 1	PLANE 2	FORCE		
1	1.000000E+03	.0	4.263256E-13	.0	1.000000E+02	.0	.0		.0

Figure 9-6. Force and Displacement at Grid Point 2 with a Zero I2

As can be seen, the GPSP module identified a singularity that is caused by the missing bending stiffness of the element. However, the constraint is placed on the Y_1 and θ_{y_1} the degrees of freedom, not the Y_b and θ_{y_b} degrees of freedom. The force in the CBAR element is correct, but the displacements are wrong. You made a common assumption that for a symmetric structure with in-plane loads, the out-of-plane properties are not needed. However, the choice of the output coordinate system and parameter AUTOSPC resulted in a modeling error. If you chose the output coordinate system for grid point 2 as the basic system, the displacements would be correct. Also, for this model, if parameter AUTOSPC is set to NO, the job will fail.

If this had been a production size model, the error would not have been so obvious and may well have gone undetected. Always look at the Grid Point Singularity Table results to make sure that the constrained degrees of freedom are acceptable.

9.4 Enforced Displacements at Grid Points (SPCD, SPC)

There are two methods available to you for specifying an enforced displacement at a component. The first method is to enter the value of the enforced displacement directly on an SPC entry.

The alternate method to enforce a displacement at a component is to use the SPCD Bulk Data entry. The SPCD entry is actually a force, not a constraint, but it is used in conjunction with an SPC entry to enforce the displacement. When you use an SPCD entry, internal forces are computed that are applied to the structure to produce the desired enforced displacements.

The format of the SPCD entry is as follows:

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

Field	Contents
SID	Identification number of a static load set.
Gi	Grid or scalar point identification number.
Ci	Component numbers.
Di	Value of enforced displacement for at Gi and Ci.

Grid points with an enforced displacement using the SPCD entry must also appear on an SPC or an SPC1 Bulk Data entry.

The SPCD method of enforcing a nonzero constraint is more efficient than using an SPC entry alone when you're using multiple subcases that specify different constraint conditions. Note also that when you use an SPCD entry, the displacement values entered on the SPC entry are ignored. The software only uses the SPCD values.

As an example of enforced displacement, consider the flat plate shown in [Figure 9-7](#). The plate is modeled with 2500 CQUAD4 elements.

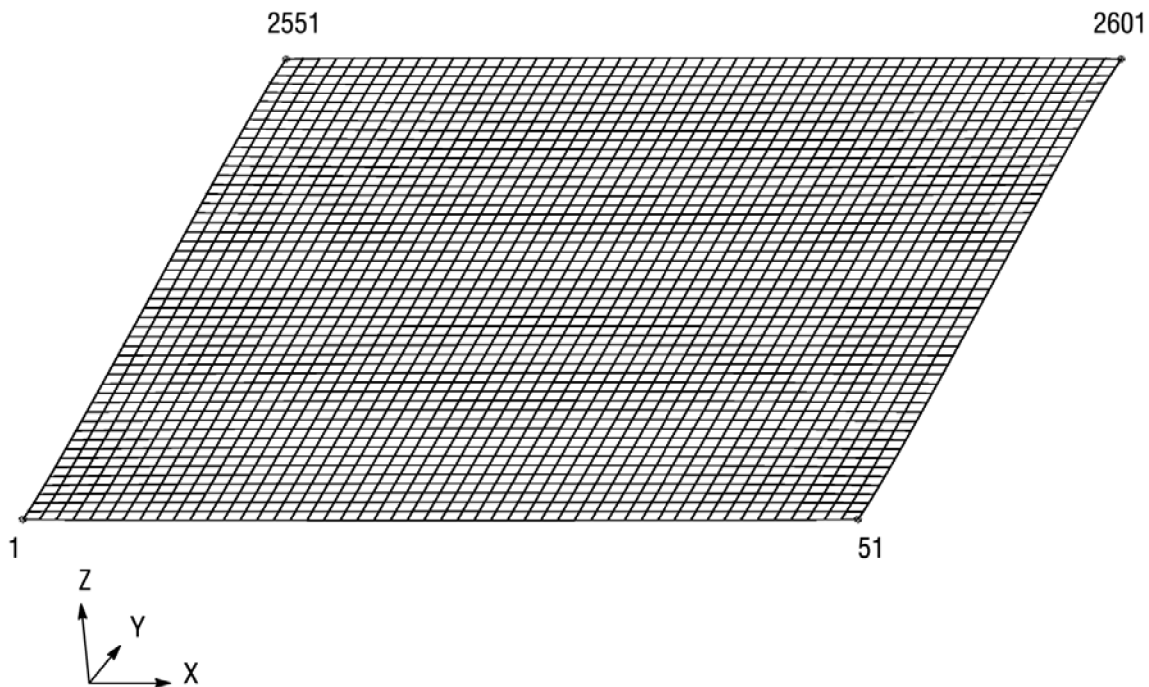


Figure 9-7. Flat Plate with Various Corner Constraints

The purpose of this example is to compare the efficiency of using the SPC/SPCD combination to that of using the SPC entry only. For the plate model, four runs are made, each having two subcases with different constraint conditions. For the input files “const1.dat” and “const2.dat”, the locations of the

constraints are the same for both subcases, but the magnitudes of the constraints change. The file “const1.dat” uses a different SPC for both subcases. The file “const2.dat” specifies the same SPC for both subcases, but it has different SPCD entries.

The input files “const3.dat” and “const4.dat” are similar to the first two; however, the locations of the constraints in the two subcases are different. [Table 9-1](#) summarizes the constraints and CPU time for all four analyses.

Table 9-1. Comparison of the SPC Versus the SPC/SPCD Combination								
	Location of Enforced Displacement							
	const1.dat		const2.dat		const3.dat		const4.dat	
	SPC Only		SPC/SPCD		SPC Only		SPC/SPCD	
Corner Displacement (in)	Subcase 1	Subcase 2	Subcase 1	Subcase 2	Subcase 1	Subcase 2	Subcase 1	Subcase 2
Grid 1	0	0	0	0	-	-	-	-
Grid 51	0	5	0	5	-	0	-	0
Grid 2551	5	5	5	5	0	5	0	5
Grid 2601	5	0	5	0	5	-	5	-
CPU Time*	7.3 (sec)		5.1 (sec)		7.3 (sec)		7.3 (sec)	
– free to move								
* CPU time on Windows 2000								

The SPC/SPCD combination is more efficient for both modeling situations. It is most advantageous to use this combination when the magnitudes but not the locations of the constraints change. The larger improvement seen by the magnitude only change is reasonable because only one SPC needs to be processed; the multiple boundary conditions are represented as multiple loads. In general, whenever you have multiple subcases with different constraint conditions, the SPC/SPCD combination is the preferred method for efficiency.

As mentioned previously, the SPCD is a force entry, not a constraint. Therefore, the SPCD is called out in the Case Control Section with the LOAD command. The input file for “const2.dat” is shown in [Listing 9-1](#).

```

$ FILENAME - CONST2.DAT
ID      LINEAR,CONST2
SOL     101
TIME    200
CEND
TITLE = ENFORCED DISPLACEMENTS USING SPC1 ONLY
DISP(PLOT) = ALL
SPC = 1
SUBCASE 1
  LOAD = 11
SUBCASE 2
  LOAD = 12
BEGIN BULK
PARAM   POST      0
$
SPC1    1          123456  1      51      2551      2601
$
SPCD    11         1        3      0.0
SPCD    11         51       3      0.0
SPCD    11        2551      3      5.0

```

```

SPCD    11      2601    3      5.0
$
SPCD    12      1      3      0.0
SPCD    12     51      3      5.0
SPCD    12    2551     3      5.0
SPCD    12    2601     3      0.0
$
INCLUDE 'const.bulk'
$
ENDDATA

```

Listing 9-1. The SPC/SPCD Combination

As a final comment, if the SPCD entry is the only loading in the model, the software issues a warning message stating that no loading is specified for your model. If you intend for the SPCD entry to be the only loading condition, you can ignore the message.

9.5 Multipoint Constraints

In NX Nastran, you can create linear constraint relationships between several degrees of freedom known as multiple-point constraints or MPCs. Each multipoint constraint is described by a single equation that specifies a linear relationship for two or more degrees-of-freedom.

In NX Nastran, the term multiple-point constraint (MPC) describes an imposed linear relationship between two or more degrees of freedom that are expressed in the form

$$\sum_j R_j u_j = 0$$

Equation 9-2.

u_j = any degree of freedom defined by a grid point or an S point

R_j = user-defined scale factor

See also

- Section 5.4 of *The NASTRAN Theoretical Manual*.

Uses of Multi-point Constraints

Multi-point constraints have many important practical applications and can be used to:

- Define the relative motion between two grid points as a degree of freedom.
- Define the average of several motions as a degree of freedom.
- Supply hinges or sliding joints between parts of a structure.
- Join dissimilar elements, for example, to join elements with rotational degrees of freedom to elements that have only translational degrees of freedom (e.g., to join shell elements to solid elements).
- Obtain the resultant of forces on a structure or on part of a structure.

- Distribute a force to several points in a structure. This is particularly useful if the force is an unknown force, for example, the force required to compress a fluid.
- Join elements with noncoincident grid points, for example, to change mesh size within a structure.
- Replace extremely stiff structural members with rigid connections. This modeling technique should only be used when necessary to improve the numerical conditioning of the stiffness matrix. MPCs can be used for this application, but they are not recommended. The preferred method is to use an R-type element because this element is simpler, and therefore less prone to error.
- Define a component of motion at a grid point that isn't aligned with the axes of the global coordinate system, for example, to let you supply a constraint in such a direction with an additional SPC entry.
- Describe rigid elements and mechanisms such as levers, pulleys, and gear trains. In this application, the degrees-of-freedom associated with the rigid element that are in excess of those needed to describe rigid body motion are eliminated with multipoint constraint equations. Treating very stiff members as rigid elements eliminates the ill-conditioning associated with their treatment as ordinary elastic elements.
- Enforce zero motion in directions other than those corresponding with components of the global coordinate system. In this case, the multipoint constraint will involve only the degrees-of-freedom at a single grid point. The constraint equation relates the displacement in the direction of zero motion to the displacement components in the global system at the grid point.
- Generate nonstandard structural elements and other special effects with scalar elements.
- Describe parts of a structure by local vibration modes (see Section 14.1 of *The NASTRAN Theoretical Manual*) where the matrix of local eigenvectors represents a set of constraints relating physical coordinates to modal coordinates.

Defining Multi-point Constraints in NX Nastran

In NX Nastran, you can define multi-point constraints on MPC, MPCADD, and MPCAX bulk data entries.

- MPC is the basic entry for defining multipoint constraints. The first component specified on the entry is the dependent degree-of-freedom, i.e., that degree-of-freedom that is removed from the equations of motion. Dependent degrees-of-freedom may appear as independent terms in other equations of the set; however, they may appear as dependent terms in only a single equation.
- The MPCADD entry defines a union of multipoint constraints.
- The MPCAX defines multipoint constraints in conical shell problems.

You can output multipoint forces of constraint with the MPCFORCE Case Control command in Solution Sequences 101 through 200.

In general, with these bulk data entries, you must provide the coefficients in the multi-point constraint equations. However, you can also use several rigid elements to generate the MPC equations automatically for some applications:

RROD	A pin-ended rod which is rigid in extension.
RBAR	A rigid bending element with six degrees-of-freedom at each end.
RTRPLT	A rigid triangular plate with six degrees-of-freedom at each vertex.
RBE1	A rigid body connected to an arbitrary number of grid points. The independent and dependent degrees-of-freedom can be arbitrarily selected by the user.
RBE2	A rigid body connected to an arbitrary number of grid points. The independent degrees-of-freedom are the six components of motion at a single grid point. The dependent degrees-of-freedom are specified at an arbitrary number of grid points.
RBE3	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.
RSPLINE	Defines multipoint constraints for the interpolation of displacements at grid points.
RSSCON	Defines multipoint constraints to model clamped connections of shell-to-solid elements.

The rigid elements will always meet equilibrium and continuity requirements, whereas this your responsibility if you use the bulk data entries to define the MPC equations.

See also

- “MPC” in the *NX Nastran Quick Reference Guide*
- “MPCADD” in the *NX Nastran Quick Reference Guide*
- “MPCAX” in the *NX Nastran Quick Reference Guide*

Using MPC

The format of the MPC entry is as follows:

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

Field	Contents
SID	Set identification number.
Gj	Identification number of grid or scalar point.
Cj	Component number.
Aj	Coefficient.

As an illustration of the use of the MPC entry, consider the following example.

Consider the model shown in **Figure 9-8**. The goal is to determine the relative motion between component 1 of grid point 1 and component 1 of grid point 2 using an MPC entry and a scalar point.

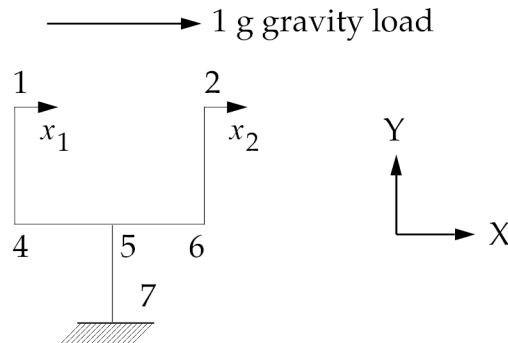


Figure 9-8. Using an MPC to Compute Relative Displacement

Let the components of motion at the two grid points whose difference is desired be designated as and as given by **Eq. 9-3**. The difference is represented by scalar point 3 defined on an SPOINT entry.

$$x_3 = x_2 - x_1$$

Equation 9-3.

The equation of constraint can be in any of the forms shown in **Eq. 9-4**.

$$x_3 - x_2 + x_1 = 0 \quad (a)$$

$$x_2 - x_1 - x_3 = 0 \quad (b)$$

$$x_1 - x_2 + x_3 = 0 \quad (c)$$

Equation 9-4.

Listing 9-2 shows the implementation of **Eq. 9-4(a)** to obtain the relative displacement. The first degree of freedom on the MPC entry is the dependent degree of freedom—which in this case is scalar point 3. It can just as well be made a component of motion at a grid point defined on a GRID entry. Grid points 1 and 2 are the independent degrees of freedom. Note that the MPC set ID (SID) is selected in the Case Control Section.

The first listed degree of freedom on the MPC entry (i.e., x_3 in this example) is designated as dependent and is placed in the m-set. The software eliminates the degrees of freedom in the m-set early in the solution process. If you want the relative motion to appear as an independent degree of freedom, express the multipoint constraint in either the second or third forms of **Eq. 9-4**.

```

$ FILENAME - RELDIFF.DAT
ID      LINEAR,RELDIFF
SOL     101
TIME    5
CEND
TITLE = RELATIVE DISPLACEMENT
SET 1 = 1,2,3
DISP = 1
MPC = 1
LOAD = 1
SPC = 1
BEGIN BULK
$
$ ONLY THE GRIDS, SPOINT AND MPC ENTRIES ARE SHOWN
$
SPOINT  3
MPC      1          3          0          1.          2          1          -1.
          1          1          1          1.
$
GRID     1          -10.      20.      0.0
GRID     2           10.      20.      0.0
GRID     4          -10.      10.      0.0
GRID     5           0.0      10.      0.0
GRID     6           10.      10.      0.0
GRID     7           0.0      0.0      0.0
ENDDATA

```

Listing 9-2. Relative Displacement Using an MPC Entry

Rules for Using Multi-point Constraints

There are three important rules for using MPCs (or R-type elements):

1. A member of the m-set can't be a member of any exclusive subset of the g-set, for example, it cannot be a degree of freedom that is removed by single-point constraints.
2. A given degree of freedom cannot be designated as a member of the m-set (dependent) more than once. Thus, it cannot appear as the first listed component on more than one MPC entry.
3. You must avoid redundant constraints. This redundancy occurs, for example, if two of the forms in Eq. 9-4 are supplied simultaneously on MPC entries. The technical difficulty that occurs is that the matrix of constraint coefficients for the m-set is singular so that the dependent degrees of freedom cannot be evaluated in terms of the independent ones. This problem also occurs when R-type elements are interconnected in a closed loop.

Multipoint constraints simultaneously supply relationships between components of motion and relationships between the forces of constraint. This can be demonstrated by using the principle that the forces acting on a rigid body can do no work. Consider, for example, the equation of constraint among three degrees of freedom shown in Eq. 9-5.

$$R_1 u_1 + R_2 u_2 + R_3 u_3 = 0$$

Equation 9-5.

The work done by the corresponding forces of constraint is given by Eq. 9-6.

$$W = 1/2(F_1u_1 + F_1u_2 + F_3u_3) = 0$$

Equation 9-6.

In order for both Eq. 9-5 and Eq. 9-6 to be valid, it is necessary that the following equation also be valid.

$$\frac{F_1}{R_1} = \frac{F_2}{R_2} = \frac{F_3}{R_3}$$

Equation 9-7.

Thus, the forces of constraint are proportional to the coefficients in the equations of constraint. As an example, consider the use of a constraint to evaluate the average of four motions, i.e.,

$$u_5 = 1/4(u_1 + u_2 + u_3 + u_4)$$

Equation 9-8.

From Eq. 9-8, the forces of constraint are given by:

$$F_1 = F_2 = F_3 = F_4 = 1/4F_5$$

Equation 9-9.

It is clear, from Eq. 9-9, that a force applied to u_5 is distributed equally to u_1 , u_2 , u_3 , and u_4 .

In general, any constraint can be viewed either as a means to constrain a component of displacement or as a means to distribute the net force acting on the constrained component of displacement (i.e., the resultant of applied force and internal forces).

See also

- “Understanding Sets and Matrix Operations” in the *NX Nastran User's Guide*

9.6 Rigid Body Supports

In the following discussion, a free body is defined as a structure that is capable of motion without internal stress; i.e., it has one or more rigid body degrees-of-freedom. The stiffness matrix for a free body is singular with the defect equal to the number of stress-free, or rigid body modes. A solid three-dimensional body has up to six rigid body modes. Linkages and mechanisms can have a greater number. In order to permit the analysis of mechanisms, no restriction is placed in the program on the number of stress-free modes.

You can define free-body supports with a SUPORT or SUPORT1 entry. With conical shell elements, you use the SUPAX entry. You must define free-body supports in the global coordinate system. You must select the SUPORT1 bulk entry with the SUPORT1 Case Control command in your input file.

In static analysis by the displacement method, the rigid body modes must be restrained in order to remove the singularity of the stiffness matrix. The required constraints may be supplied with single point constraints, multipoint constraints, or free body supports.

If you use free body supports, NX Nastran calculates the rigid body characteristics and checks the sufficiency of the supports. The software checks the supports by calculating the rigid body error ratio and the strain energy as defined in the Rigid Body Matrix Generator operation.

This error ratio and the strain energy are automatically printed following the execution of the Rigid Body Matrix Generator. The error ratio and the strain energy should be zero, but may be nonzero for any of the following reasons:

- Round-off error accumulation.
- Insufficient free body supports have been provided.
- Redundant free body supports have been provided.

The redundancy of the supports may be caused by improper use of the free body supports themselves or by the presence of single point or multipoint constraints that constrain the rigid body motions.

Static analysis with inertia relief is necessarily performed on a model that has at least one rigid body motion. Such rigid body motion must be constrained by the use of free body supports. These supported degrees-of-freedom define a reference system, and the elastic displacements are calculated relative to the motion of the support points. The element stresses and forces will be independent of any valid set of supports.

Rigid body vibration modes are calculated by a separate procedure provided that you supply a set of free body supports. This is done to improve efficiency and, in some cases, reliability. The determinant method, for example, has difficulty extracting zero frequency roots of high multiplicity, whereas the alternate procedure of extracting rigid body modes is both efficient and reliable. If you don't specify free body supports (or you specify an insufficient number of them), the (remaining) rigid body modes are calculated by the method selected for the finite frequency modes, provided zero frequency is included in the range of interest. If you don't provide free body supports, and if zero frequency isn't included in the range of interest, the rigid body modes won't be calculated.

You must specify free body supports if the mode acceleration method of solution improvement is used for dynamic problems having rigid body degrees-of-freedom. This solution improvement technique involves a static solution, and although the dynamic solution can be made on a free body, the static solution cannot be performed without removing the singularities in the stiffness matrix associated with the rigid body motions.

See also

- “[Static Solutions in SubDMAP SEKRRS](#)” in the *NX Nastran User's Guide*
- “Rigid-body Modes” in the *NX Nastran Basic Dynamic Analysis User's Guide*
- “SUPPORT Entry” in the *NX Nastran Basic Dynamic Analysis User's Guide*

9.7 Linear Contact Using Constraints

Note

Note: This section describes an older approach to defining linear contact conditions with the use of constraints. When this procedure was created, the options for linear contact were limited and required some creative modeling techniques. There are now improved procedures to define linear contact conditions, the use of this procedure is no longer recommended. The documentation for this procedure is included here for those interested in the technique, and to support legacy input files with this type of definition. For a full discussion on defining the recommended linear contact conditions, see the chapter “Contact for Solutions 101, 103, 111, 112” in this book.

Introduction

Traditionally, GAP elements were used to model contact-type problems and had required the use of nonlinear solution sequences—specifically, SOL 106 or 129. In contrast to the gap element available for nonlinear analysis (SOL 106 and SOL 129), the “linear” gap capabilities in SOL 101 did not require estimation of gap stiffness. It used an iterative constraint type approach. The constraints were applied to grid points or SPOINTS. The constraints ensured that:

1. The displacement (UR) cannot be negative. This is to ensure that there is no penetration. Therefore, the chosen degree-of-freedom must be perpendicular to the contact surface and positive in the opening direction.
2. The force of constraint (QR) cannot be negative. This is to ensure that there is no tension.

The constraints are satisfied by an iterative technique that is built into SOL 101. The iterative process starts with a random vector. This random vector assumes certain grids to be in contact and other grids to be in an open state. A solution is obtained when all the gap constraints are satisfied, i.e., there's no penetration and no tension forces. If a limit cycle (return to a previous state) is detected during the iterations, a new random start vector is tried.

This approach provides an alternate method to the use of GAP elements in SOL 106. Some experiments have shown that the cost of analysis will be about the same. The advantage is that you do not have to learn how to calculate the GAP stiffness nor how to control SOL 106. Multiple load conditions are allowed, and each will be solved separately.

If the constraint is between a finite element model and a fixed boundary, then arrange to have one of the degrees-of-freedom at the boundary grid points represent motion perpendicular to the boundary. A positive displacement represents motion away from the boundary. If, on the other hand, the constraint represents relative motion between two bodies, MPC equations are needed to define a relative motion degree-of-freedom, which is then constrained to have a non-negative displacement.

Input

A SUPORT entry along with PARAM,CDITER are required. These two items plus some optional parameters are described below.

SUPPORT entry	Selects constrained degrees-of-freedom. These points must be in the a-set of the residual structure. This means they must not be dependent in an MPC equation, constrained by SPC, partitioned by OMIT, or in an upstream superelement.
PARAM,CDITER,x	CDITER Constraints will be applied if CDITER is greater than zero. The value x is the maximum number of iterations allowed. (Default = 0).
PARAM,CDPRT	Controls the printing of constraint violations during the iterations. The sparse matrix printer prints UR (negative displacements) and QR (negative forces of constraint) for constrained degrees-of-freedom. (Default = 'YES')
DMIG,CDSHUT	Optional DMIG input of the vector defining the state of the constrained degrees-of-freedom. A one indicates closed and a zero indicates open. If DMIG,CDSHUT is not provided, all SUPPORT degrees of freedom will assumed to be closed initially.
PARAM,CDPCH	Controls the PUNCH output of DMIG,CDSHUT records for the final state of the constrained degrees-of-freedom. (Default = 'NO'). This output will be created in a separate .pch file.

Output

The output is similar to standard SOL 101 output. Forces for closed degrees-of-freedom will appear in the SPCFORCE output. In addition, optional diagnostic information for the iterations can be requested (PARAM,CDPRT). A final state vector may also be sent to a .pch file (PARAM,CDPCH).

The .f06 file should be examined to ensure that the iterations have converged, since the results of the last iteration will be output. The last iteration should have zero changes. A typical converged output is shown below. More detailed explanation are included in the actual examples.

```

1  DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1          SEPTEMBER  6, 2001  MSC.NASTRAN  4/ 9/01  PAGE   10
0
0
0  POINT          QRI          POINT          VALUE          POINT          VALUE          POINT          VALUE          POINT          VALUE          POINT          VALUE
0
COLUMN          1
5 T2 -2.37141E-02
*** END ITERATION          3, CHANGES          1
*** END ITERATION          4, CHANGES          0
*** USER INFORMATION MESSAGE 9097 (CSTRDISP)
*** CONSTRAINED DISPLACEMENT ITERATIONS CONVERGED          LOAD CASE =          1
1  DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1          SEPTEMBER  6, 2001  MSC.NASTRAN  4/ 9/01  PAGE   11

```

Limitations

The following is a list of limitations for the linear gap elements.

- The only nonlinearity allowed is the constrained displacements.
- There is no sliding friction.
- Free bodies—using the SUPPORT entry to define rigid body modes— cannot be used in the same model with linear gap elements. The parameters INREL and CDITER are mutually exclusive. A fatal message is issued if both parameters are present.

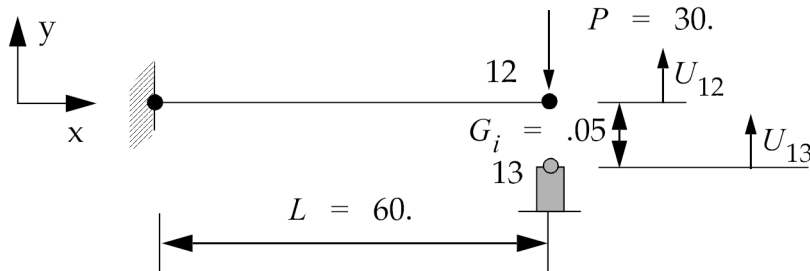
- No constraint changes are allowed between subcases.
- There is no guarantee that the solution will converge or that all systems will follow the same path.

Examples

Two examples are shown below to illustrate the use of the linear GAP element.

EXAMPLE 1(cd_0.dat)

This model consists of a cantilever beam with a stopper and a vertical load of 30 lbs applied at the free-end. There is a clearance of 0.05 inch between the free-end and the stopper. The purpose of this problem is to find out if 30 lbs is sufficient to cause the free-end to hit the stopper.



The first step in solving this problem is to define a variable S_{51} to monitor the opening between the free-end (grid point 12) and the stopper (grid point 13). The following equation

$$S_{51} = U_{12} + G_i - U_{13}$$

Equation 9-10.

- U_{12} = the displacement at grid point 12
- U_{13} = the displacement at grid point 13
- G_i = the initial gap opening
- S_{51} = the relative distance between grids points 12 and 13 including the effect of the initial opening

accomplishes this goal. Grid point 13 can be removed from the equation since the stopper does not move. Eq. 9-10 can be simplified and recast as

$$-U_{12} + S_{51} - G_{101} = 0$$

Equation 9-11.

so that an MPC equation can be written. Two scalar points are introduced in this case: Scalar points 101 for G_{101} (G_i) and 51 for S_{51} . Note that neither S_{51} nor G_{101} can be defined as dependent since they already belong to the r-set and s-set, respectively. A copy of the input file is shown in Listing 9-3.

```

$
$   cd_0.dat
$
sol 101
cend
Title = Gap elements, Cantilever beam
    mpc = 77
    spc = 88
    load = 300
    disp = all
    spcf = all
    mpcf = all
$
begin bulk
$
grid 11          0.    0.    0.
grid 12          60.    0.    0.
cbar 15          1    11    12    0.    1.    0.
pbar 1           2    1.    1.    1.    1.
mat1 2           3.e7    0.3
force 300        12    -30.    0.    1.    0.
spoint          51    101
support         51    0
spc 88          11    123456
spc 88          101    0    .05
mpc 77          12    2    -1.0  51    0    1.

```

Listing 9-3. Input file for Cantilever Beam with Stopper.

Eq. 9-11 is represented by MPC,77. Scalar point 51 must also be placed on the SUPORT entry. In addition, an enforced displacement of 0.05 must also be imposed on scalar point 101. PARAM,CDITER,10 requests that a maximum of 10 iterations be performed for this analysis. If more than 10 iterations are required for this problem, the results at the end of the 10th iteration will be printed. PARAM,CDPRT,YES is optional in this case since it is the default.

An abridged output is shown in Eq. 9-7. The large non-zero values for epsilon and strain energy for the SUPORT degree of freedom can be safely ignored as we are not performing inertia relief analysis. The solution converges in 1 iteration as indicated by the following output.

```

    ^^^ END ITERATION          1, CHANGES          0

```

The gap is in contact as indicated by the following scalar point 51 output.

0	POINT	SHUT	POINT	VALUE	POINT	VALUE	POINT	VALUE	POINT	VALUE
0	COLUMN	VALUE	1							
	51	S	1.000000E+00							

The displacement T2 at grid point 12 is –0.05, which implies that the cantilever beam hits the stopper. The load required to close the gap is:

$$P_g = \frac{3 \cdot EI \cdot U_{12}}{L^3} = \frac{3(3E7)(1)(0.05)}{60 \cdot 60 \cdot 60} = 20.833$$

Equation 9-12.

which is what the spcforce output at grid point 11 shows. The rest of the applied load 9.167 (30-20.833) is reacted by the stopper as indicated by spcforce at grid point 51 and, similarly, the mpcforce at grid point 12.


```

0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
    DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
    FOR DATA BLOCK KLR
    SUPPORT PT.NO.      EPSILON      STRAIN      ENERGY      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH
    ASTERISKS
        1      2.5000000E-01      2.0833333E+02      *****
*** USER INFORMATION MESSAGE 5293 (SSG3A)
    FOR DATA BLOCK KLL
    LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH
    ASTERISKS
        1      0.0000000E+00      1.5625000E+00
^^^ USER INFORMATION MESSAGE 9077 (CSTRDISP)
^^^ CONSTRAINED DISPLACEMENT ITERATIONS
^^^ CONSTRAINED DISPLACEMENT  DIAGNOSTICS, UR= NEGATIVE DISPLACEMENT, QR= NEGATIVE FORCE
^^^ END ITERATION      1, CHANGES      0
^^^ USER INFORMATION MESSAGE 9097 (CSTRDISP)
^^^ CONSTRAINED DISPLACEMENT ITERATIONS CONVERGED      LOAD CASE =      1
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
7

0
0      SHUT
    POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT
    VALUE

COLUMN      1
    51 S 1.000000E+00
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
8
      .
0      .

      D I S P L A C E M E N T      V E C T O R

    POINT ID.      TYPE      T1      T2      T3      R1      R2      R3
        11      G      0.0      0.0      0.0      0.0      0.0      0.0
        12      G      0.0      -5.000000E-02      0.0      0.0      0.0      -1.250000E-03
        51      S      0.0
       101      S      5.000000E-02
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
11
0

      F O R C E S      O F      S I N G L E - P O I N T      C O N S T R A I N T

    POINT ID.      TYPE      T1      T2      T3      R1      R2      R3
        11      G      0.0      2.083333E+01      0.0      0.0      0.0      1.250000E+03
        51      S      9.166666E+00
       101      S      -9.166666E+00
1  GAP ELEMENTS, CANTILEVER BEAM      JULY 31, 2001 MSC.NASTRAN 4/ 9/01 PAGE
13

0

      F O R C E S      O F      M U L T I P O I N T      C O N S T R A I N T

    POINT ID.      TYPE      T1      T2      T3      R1      R2      R3
        12      G      0.0      9.166666E+00      0.0      0.0      0.0      0.0
        51      S      -9.166666E+00
       101      S      9.166666E+00

```

Figure 9-9. Abridged Output of Cantilever Beam with Stopper.

EXAMPLE 2(cd_1.dat)

The second example is a thick pad supported on a rigid base with a vertical point load applied at the center. Due to symmetry, a half model is used by applying the appropriate boundary condition at the center. The problem is to determine where the lift off occurs along the base (grid points 1 through 7.) An abridged input file for this example is shown in [Listing 9-4](#). A planar problem ([Figure 9-10a](#)) is considered in this case by constraining 3456 on the GRDSET entry. SPC1,200 applies the symmetric boundary condition at the center line of the structure. The potential contact points are specified on the seven SUPORT entries—one for each grid point at the bottom. In this case, since the contact is between a structure and a rigid surface, and there is no initial gap, the use of the MPC is not required. A maximum of up to 20 iterations (PARAM,CDITER,20) are allowed for this case. Furthermore, an initial guess of where the contact may occur is provided (DMIG,SHUT.) In this case, grid points 1 and 7 are assumed to be shut and grid points 2 through 6 are assumed to be opened initially. PARAM,CDPCH,YES requests that the final open/shut configuration be created in a separate punch file. PARAM,CDPRT,YES requests that the printing of the constraint violations during the iterations, which is the default.

```
$ cd_1.dat
$
SOL 101
CEND
TITLE = DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1 CD_1
SPC = 200
LOAD = 300
DISPL = ALL
OLOAD = ALL
SPCFO = ALL
BEGIN BULK
$
$ SYMMETRIC BOUNDARY AT X=0, DOWNWARD LOAD AT TOP CENTER
$
SPC1 200 1 1 11 21 31 41
GRDSET 3456
FORCE 300 41 1. -1.
$
$ CONSTRAIN DISPLACEMENTS ON BOTTOM EDGE, OPTIONAL CDSHUT INPUT
$
$2345678 2345678
SUPORT 1 2
SUPORT 2 2
SUPORT 3 2
SUPORT 4 2
SUPORT 5 2
SUPORT 6 2
SUPORT 7 2
$
PARAM CDITER 20
PARAM CDPRT YES
PARAM CDPCH YES
$
DMIG CDSHUT 0 9 1 0 1
DMIG CDSHUT 1 0 1 2 1.
7 2 1.
$
.
ENDDATA
```

Listing 9-4. Input File for Pad on Rigid Foundation.

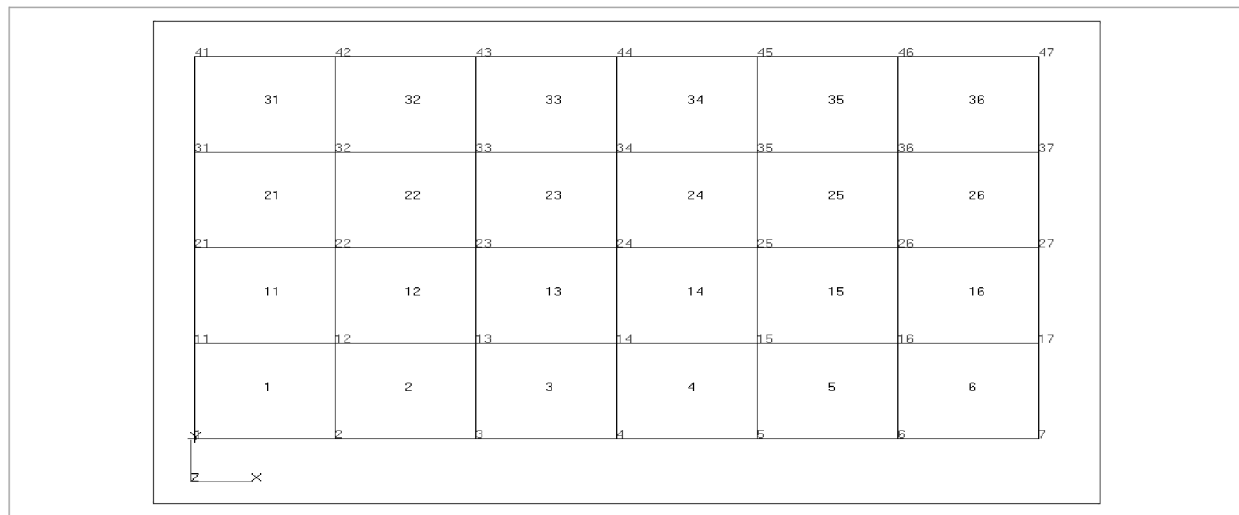


Figure 9-10. Undeformed Plot of the Thick Pad.

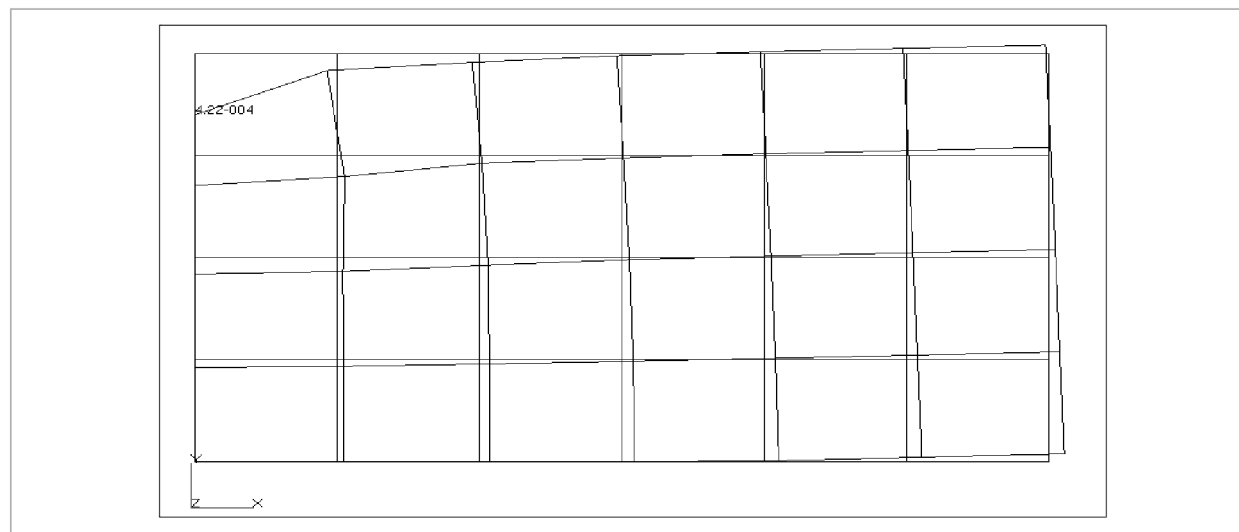


Figure 9-11. Deformed Plot of the Thick Pad.

From [Figure 9-11b](#), it is obvious that grid points 6, 7, and possibly 5 have lifted off the foundation due to the downward vertical load. Looking at the output ([Figure 9-12](#)), one can see that grid points 1, 2, 3, and 4 are in contact as indicated under the “SHUT” output. Furthermore, the displacement output illustrates that grid points 1, 2, 3, and 4 have zero displacement in the T2 direction—indicating that these four points are in contact. The T2 displacements for grid points 5, 6, and 7 are positive—indicating that these three points have lifted off the foundation. The applied vertical load is reacted at the four grid points (1-4) as illustrated by the spcforce output.

```

0
0
SHUT
POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT      VALUE      POINT
VALUE

COLUMN      1
      1 T2  1.00000E+00      2 T2  1.00000E+00      3 T2  1.00000E+00      4 T2  1.00000E+00
1 DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1  SEPTEMBER 10, 2001  MSC.NASTRAN  4/9/01
PAGE 12

0

      D I S P L A C E M E N T   V E C T O R

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
      1      G      0.0      0.0      0.0      0.0      0.0      0.0
      2      G      2.815052E-05  0.0      0.0      0.0      0.0      0.0
      3      G      4.985067E-05  0.0      0.0      0.0      0.0      0.0
      4      G      6.308859E-05  0.0      0.0      0.0      0.0      0.0
      5      G      7.122347E-05  8.332880E-06  0.0      0.0      0.0      0.0
      6      G      7.456562E-05  3.158361E-05  0.0      0.0      0.0      0.0
      7      G      7.511267E-05  5.419435E-05  0.0      0.0      0.0      0.0
      .
      .

0

      L O A D   V E C T O R

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
      41      G      0.0      -1.000000E+00  0.0      0.0      0.0      0.0
1 DEMONSTRATE CONSTRAINED DISPLACEMENT IN SOL 101, #1  CD_1  SEPTEMBER 10, 2001  MSC.NASTRAN  4/9/01  PAGE
15

0

      F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
      1      G      -8.072071E-02  2.248780E-01  0.0      0.0      0.0      0.0
      2      G      0.0      3.992608E-01  0.0      0.0      0.0      0.0
      3      G      0.0      2.689573E-01  0.0      0.0      0.0      0.0
      4      G      0.0      1.069040E-01  0.0      0.0      0.0      0.0
      11      G      -1.541045E-01  0.0      0.0      0.0      0.0      0.0
      21      G      -1.733866E-01  0.0      0.0      0.0      0.0      0.0
      31      G      -1.259178E-01  0.0      0.0      0.0      0.0      0.0
      41      G      5.341296E-01  0.0      0.0      0.0      0.0      0.0

```

Figure 9-12. Abridged Output of Thick Pad.

Chapter 10: Understanding Sets and Matrix Operations

- *Overview of Sets and Matrix Operations*
- *Understanding the Global Displacement Set and General Solution Process*
- *Static Condensation (Guyan Reduction)*
- *Sets for Dynamic Reduction*
- *Sets for Aerodynamics*
- *Output Selection Via Set Specification*

10.1 Overview of Sets and Matrix Operations

When you define a finite element model in NX Nastran, the software writes equations to simulate the necessary structural properties. These equations are rows and columns in the matrix equations that describe your model's structural behavior. Certain data entries (i.e., SPCs, MPCs, ASETs, etc.) cause matrix operations to be performed in the various stages of the solution process.

To organize the matrix operations, NX Nastran assigns each degree of freedom (DOF) in your model to a displacement set. NX Nastran uses the sets to successively eliminate variables during the solution process.

Most matrix operations used in a structural analysis involve partitioning, merging, and/or transforming matrix arrays from one subset to another. For example, all components of motion of a given type form a vector set that is distinguished from other vector sets. A given component of motion can belong to many combined sets but can belong to only one exclusive set.

In NX Nastran, there are two basic types of degree-of-freedom sets:

- Mutually exclusive sets

In NX Nastran, mutually exclusive sets are the data sets that are removed by a constraint or reduction operation during the analysis. If a DOF is a member of a mutually exclusive set, it can't be a member of any other mutually exclusive set. For example, a degree-of-freedom may not be in the m-set on two RBAR entries, or on a selected SPC entry and a SUPORT entry.

- Combined sets

Combined sets are formed through the union (combination) of two or more sets. Each degree-of-freedom is also a member of one or more combined sets called "supersets." Supersets are formed when the software combines the mutually exclusive sets.

The contents of both of these types of sets is listed in the *NX Nastran Quick Reference Guide*.

See also

- "[Data Processing and Matrix Operations by Functional Module](#)" in the *NX Nastran User's Guide*
- "[Degree-of-Freedom Sets](#)" in the *NX Nastran Quick Reference Guide*

10.2 Understanding the Global Displacement Set and General Solution Process

A finite element model's degrees of freedom include all the components of displacement at the grid points and scalar points. Each grid point has six degrees of freedom, while each scalar point has one. The union of all the local displacement components at all the grid points and scalar points is called the global displacement set or *g*-set. The *g*-set is the top level set in NX Nastran and represents an unconstrained set of structural equations.

The *g*-set displacement is represented as either u_g or $\{u_g\}$. Its significance is that the mathematical expression for the static equilibrium of the finite element model can be expressed in matrix form as:

$$[K_{gg}]\{u_g\} = \{P_g\}$$

Equation 10-1.

where $[K_{gg}]$ is the global stiffness matrix and $\{P_g\}$ is the vector of explicit loads (e.g., loads generated from FORCE entries) or implicit loads (e.g., loads generated from GRAV entries) applied to all of the grid points and scalar points.

The stiffness matrix $[K_{gg}]$ is formed by generating and assembling the stiffness matrices for all of the elements. The load vector is formed by assembling all of the implicitly and explicitly defined load components within a given loading condition. Loads defined within elements are automatically transferred to the appropriate grid points in a consistent manner.

The number of terms in the $\{u_g\}$ and $\{P_g\}$ vectors is equal to six times the number of grid points plus the number of scalar points. $[K_{gg}]$ is a real and symmetric matrix.

$[K_{gg}]$ is usually singular; therefore, the equations of equilibrium cannot be solved in the form shown by Eq. 10-1.

At the next step in the solution process, NX Nastran partitions the g -set into two subsets: the m -set and n -set.

The dependent DOFs of all multipoint constraint relations (MPCs, RBEs, etc.) define the m -set. During the solution process, the contributions of these m -set DOFs are condensed into the set of independent DOFs, the n -set. The n -set represents all the independent DOFs that remain after the dependent DOFs are removed from the active set of equations. The m -set represents the DOFs that are linearly dependent upon the n -set DOFs.

Using the n -set, SPCs are then applied to the independent equations to further partition the equations.

- Degrees of freedom identified by PARAM,AUTOSPC and/or SPCi entries are assigned to the sb -set.
- Permanent constraints, those DOFs specified in field 8 of the GRID entry, are assigned to the sg -set.
- The union of the sb - and sg -sets compose the s -set.

After reducing the n -set by applying the s -set constraints, the f -set remains. The f -set represents the “free” DOFs of the structure. If applied constraints are properly specified, then the f -set equations represent a statically stable solution (i.e., a nonsingular stiffness matrix).

If you want the software to perform static condensation (Guyan reduction), the f -set is further reduced by partitioning it into the o - and a -set. The o -set DOFs are those that are to be eliminated from the active solution by static condensation. The a -set is the “analysis” set and is often the partition at which the solution is performed. By default, all DOFs in the f -set are assigned to the a -set. In other words, the o -set is null, by default.

If you use the SUPORT entry, the software places the degrees-of-freedom defined on the SUPORT entry are placed in the r -set or “reference” set. This set aids in the calculation of inertia forces, balances the applied loads, and helps determine rigid body modes. When the r -set is partitioned from the t -set, the l -set remains. This final set is termed the “leftover” set and is the lowest level of partitioning performed in NX Nastran static analysis. The l -set partition is the matrix on which the final solution is performed.

See also

- “[Static Condensation \(Guyan Reduction\)](#)” in the *NX Nastran User's Guide*
- “[Inertia Relief in Linear Static Analysis](#)” in the *NX Nastran User's Guide*

Understanding Basic Partitioning Operations

NX Nastran partitions the combined sets and removes one set from the analysis. [Table 10-1](#), for example, lists the basic set partitioning operations. The *m*-, *s*-, *o*-, and *r*-sets are the mutually exclusive sets.

Table 10-1. Basic Set Partitioning Operations
<i>g</i> -set - <i>m</i> -set = <i>n</i> -set
<i>n</i> -set - <i>s</i> -set = <i>f</i> -set
<i>f</i> -set - <i>o</i> -set = <i>a</i> -set
<i>a</i> -set - <i>r</i> -set = <i>l</i> -set

Physically, the mutually exclusive set partitioning ensures that operations can't be performed on a DOF that's no longer active. For example, if you apply an SPC to a DOF that is a dependent degree of freedom on an RBAR, NX Nastran issues User Fatal Message (UFM) 2101A. Using an SPC entry moves a DOF to the *sb*-set, but this cannot occur if the same DOF is already a member of the *m*-set. Both the *m*-set and *sb*-set are mutually exclusive.

When a particular set has no DOFs associated with it, it is a null set. If a null set exists, NX Nastran applies a partition and moves the DOFs to the subsequent partition.

The final set of degrees of freedom remaining after the exclusive sets are eliminated is called the *l*-set (degrees of freedom left over). The static solution is performed on the *u_l* set. The reduced equations of equilibrium are expressed in the matrix form

$$[K_{ll}]\{u_l\} = \{P_l\}$$

Equation 10-2.

The reduced stiffness matrix $[K_{ll}]$ is symmetric and nonsingular if all mechanisms and rigid body motion are removed by the partitioning of the *m*-set, *s*-set, and *r*-set. [Eq. 10-2](#) is solved for $\{u_l\}$ by Gaussian elimination (decomposition and forward-backward substitution). All of the other subsets of vector—namely *u_r*, *u_o*, *u_s*, and *u_m*—can be obtained subsequently by successive substitution into their defining equations. By default, all *f*-set degrees of freedom are assigned to the *a*-set. If you don't have a rigid body support (*r*-set) in your model, then the *a*-set is identical to your *l*-set.

Determining Set Membership

You can output the set membership of all of the degrees of freedom with the PARAM,USETPRT and PARAM,USETSEL Bulk Data entries. Use this parameter with caution as it can produce a substantial amount of output for a large model.

See also

- “USETPRT” in the *NX Nastran Quick Reference Guide*
- “USETSEL” in the *NX Nastran Quick Reference Guide*

10.3 Static Condensation (Guyan Reduction)

The terms static condensation and Guyan reduction represent the same matrix operations and are used interchangeably. Guyan Reduction involves a reformulation of the stiffness and mass matrices based on partitions of the stiffness matrix. If you start with the stiffness equation in terms of the set of the unconstrained (free) structural coordinates, you have

$$[K_{ff}]\{u_f\} = \{P_f\}$$

Equation 10-3.

Partitioning the statically independent free degrees of freedom into two subsets of the *f*-set, you obtain

$$\{u_f\} = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix}$$

Equation 10-4.

u_a = the *a*-set variables (the “analysis” set)

u_o = the *o*-set variables, which are removed by static condensation (the “omitted” set)

Rewriting the static equation for u_f in terms of *o*-set and *a*-set partitions, you obtain

$$[K_{ff}][u_f] = \begin{bmatrix} \overline{K_{aa}} & \overline{K_{ao}} \\ \overline{K_{oa}} & \overline{K_{oo}} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} \overline{P_a} \\ \overline{P_o} \end{Bmatrix}$$

Equation 10-5.

Expanding the bottom equation, you arrive at

$$\{u_o\} = [G_{oa}]\{u_a\} + \{u_{oo}\}$$

Equation 10-6.

where:

$$\begin{aligned}[G_{oa}] &= -[K_{oo}]^{-1}[K_{oa}] \\ \{U_{oo}\} &= [K_{oo}]^{-1}[P_o]\end{aligned}$$

Expanding the top part of Eq. 10-5 and substituting u_o with Eq. 10-6, you arrive at the equation

$$\{u_a\} = [K_{aa}]^{-1}\{P_a\}$$

Equation 10-7.

where:

$$\begin{aligned}[K_{aa}] &= [\overline{K}_{aa} + K_{ao}G_{oa}] \\ \{P_a\} &= [\overline{P}_a + G_{oa}^T P_o]\end{aligned}$$

The solution to the f -set degrees of freedom is obtained in three phases. The first phase is to reduce the f -set to the a -set using Guyan reduction. The second phase is to solve for $\{u_a\}$ in Eq. 10-7, which is the analysis set. Once this is done, $\{u_o\}$ can then be solved by Eq. 10-6, which is the omitted set. The conventional method involves solving Eq. 10-3 in a single step.

In static analysis, the results using static condensation are numerically exact. The partitioned solution merely changes the order of the operations of the unpartitioned solution. However, Guyan reduction isn't generally recommended for static analysis. This recommendation stems from the cost of the Guyan reduction process since it creates reduced matrices $[K_{aa}]$ and $[M_{aa}]$, which are typically smaller but denser (i.e., matrix bandwidth is destroyed). Additional decomposition of the matrix $[K_{oo}]$ is also required. In other words, even though the solution size is smaller, the solution time may take longer and the disk space requirement may be larger due to a denser matrix. Furthermore, it requires additional interaction on your part. However, some understanding of the concept of Guyan reduction in static analysis is quite helpful if you ever decide to apply this feature in dynamic, cyclic symmetry, or superelement analyses.

Guyan reduction has special applications in dynamic analysis. In dynamic analysis, the reduction is approximate; the term $\{u_{oo}\}$ is ignored. The reduction is based solely on static transformation and is exact provided that no loads are applied to the o -set degrees of freedom.

Using Static Condensation

In NX Nastran, the ASETi and OMITi bulk data entries define the degrees-of-freedom in the analysis and omitted sets, respectively. You can use either the ASET/ASET1 or the OMIT/OMIT1 entries to specify only the a -set (with ASET/ASET1 entries) or o -set (with OMIT/OMIT1 entries) degrees of freedom. The software automatically places the remaining f -set DOFs in the complementary set, as described below. However, if you specify both the a -set and o -set DOFs, then the unspecified remaining f -set DOFs are automatically placed in the o -set. If you specify the same DOF on both the a -set and o -set, the job fails with UFM 2101A.

The a -set and the o -set are created in one of the following ways:

1. If OMITi entries are present, the o-set consists of degrees-of-freedom listed explicitly on OMITi entries. The remaining f-set degrees-of-freedom are placed in the b-set. They are then also members of the a-set.
2. If ASETi or QSETi entries are present, the a-set consists of all degrees-of-freedom listed on ASETi entries and any entries listing its subsets, such as QSETi, SUPORTi, CSETi, and BSETi entries. The remaining f-set degrees-of-freedom are placed in the o-set. Note, however, there must be at least one explicit ASETi or QSETi entry type present for the o-set to exist, even if it is a redundant specification.
3. If there are no ASETi or QSETi entries, but there are SUPORT, BSETi, or CSETi entries, the entire f-set is placed in the a-set, and the o-set isn't created.
4. There must be at least one explicitly defined ASETi, QSETi, or OMITi entry for the o-set to exist, even if the ASETi, QSETi, or OMITi entry is redundant.

Notes:

- If you're performing an analysis using conical shell elements, you use the OMITAX entry instead of the OMITi entry.
- In superelement analysis, exterior ("boundary") degrees-of-freedom are automatically determined and placed in the a-set of the superelement. Exterior degrees-of-freedom for superelements are therefore equivalent to degrees-of-freedom listed on ASETi entries.

The a-set defines a combined set. Degrees-of-freedom listed on Bulk Data entries that define its subsets, the q-, r-, c-, and b-set, may also be listed on ASETi Bulk Data entries, although this is a redundant specification and isn't necessary. However, automatic restarts are more reliable if ASETi entries are present for all degrees-of-freedom listed on SUPORT entries.

Since the reduction process is performed on an individual degree of freedom, it is possible to have some of the degrees of freedom at a grid point in the a-set and other degrees of freedom at a grid point in one of the other mutually exclusive sets. No additional user input is required.

Format for ASET

You can use ASET in conjunction with ASET1. The formats for the ASET/ASET1 are as follows:

1	2	3	4	5	6	7	8	9	10
ASET	G1	C1	G2	C2	G3	C3	G4	C4	

Field

Contents

- Gi The i-th grid or scalar point identification number
- Ci Component number(s) for the i-th grid or scalar point

Example

ASET	5	3	30	345	8	12345			
------	---	---	----	-----	---	-------	--	--	--

In this example, component 3 of grid point 5, components 3, 4, and 5 of grid point 30, and components 1,2,3,4, and 5 of grid point 8 are assigned to the *a*-set. All other degrees of freedom that aren't otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (*o*-set).

Format for ASET1

1	2	3	4	5	6	7	8	9	10
ASET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	G11	-etc.-				

Field	Contents
C	Component number(s).
Gi	The i-th grid or scalar point identification number.

Example

ASET1	345	3	4	1	16	8	9	5	
	13	20	25						

In this example, components 3,4, and 5 of grid points 1, 3, 4, 5, 8, 9, 13, 16, 20, and 25 are assigned to the *a*-set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the omitted set (*o*-set).

Alternate Format for ASET1

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	"THRU"	ID2					

Field	Contents
C	Component number(s).
ID1	The first grid or scalar point identification number in the range.
ID2	The last grid or scalar point identification number in the range.

Example

ASET1	123456	10	THRU	909					
-------	--------	----	------	-----	--	--	--	--	--

In this example, components 1,2,3,4,5, and 6 of grid point 10 through 909 are assigned to the *a*-set. As long as at least one grid point between 10 and 909 exists, all the grid points between grid points 10 and 909 don't need to exist. NX Nastran places all other degrees of freedom that are outside of grid points 10 through 909 and not otherwise specified (e.g., on SPCi or MPC entries) in the omitted set (*o*-set).

Format for OMIT

OMIT can be used in conjunction with OMIT1. The formats for the OMIT/OMIT1 are as follows:

1	2	3	4	5	6	7	8	9	10
OMIT	G1	C1	G2	C2	G3	C3	G4	C4	

Field	Contents
G_i	The i -th grid or scalar point identification number.
C_i	Component number(s) for the i -th grid or scalar point.

Example

OMIT	5	3	30	345	8	123456			
------	---	---	----	-----	---	--------	--	--	--

For the above example, component 3 of grid point 5, components 3, 4, and 5 of grid point 30, and components 1,2,3,4,5, and 6 of grid point 8 are assigned to the o -set. All other degrees of freedom that are not otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (a -set).

Format for OMIT1

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	G11	-etc.-				

Field	Contents
C	Component number(s).
G_i	The i -th grid or scalar point identification number.

Example

OMIT1	345	3	4	1	16	8	9	5	
	13	20	25						

For the above example, components 3, 4, and 5 of grid points 1, 3, 4, 5, 8, 9, 13, 16, 20, and 25 are assigned to the o -set. All other degrees of freedom that aren't otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (a -set).

Alternate Format for OMIT1

1	2	3	4	5	6	7	8	9	10
OMIT1	C	ID1	"THRU"	ID2					

Field	Contents
C	Component number(s).
ID1	The first grid or scalar point identification number in the range.
ID2	The last grid or scalar point identification number in the range.

Example

OMIT1	123456	10	THRU	909					
-------	--------	----	------	-----	--	--	--	--	--

For this example, components 1,2,3,4,5, and 6 of grid points 10 through 909 are assigned to the o-set. All grid points between grid points 10 and 909 need not exist as long as at least one of them exists. All other degrees of freedom that are outside of grid points 10 through 909 and not otherwise specified (e.g., on SPCi or MPC entries) are placed in the analysis set (a-set).

The choice of whether to use the ASET/ASET1 or OMIT/OMIT1 is really a matter of convenience.

Consider a simple example using Guyan reduction. The cantilever beam model in **Figure 10-1** is used for this purpose. The common Bulk Data entries for this model are contained in **Listing 10-1**.

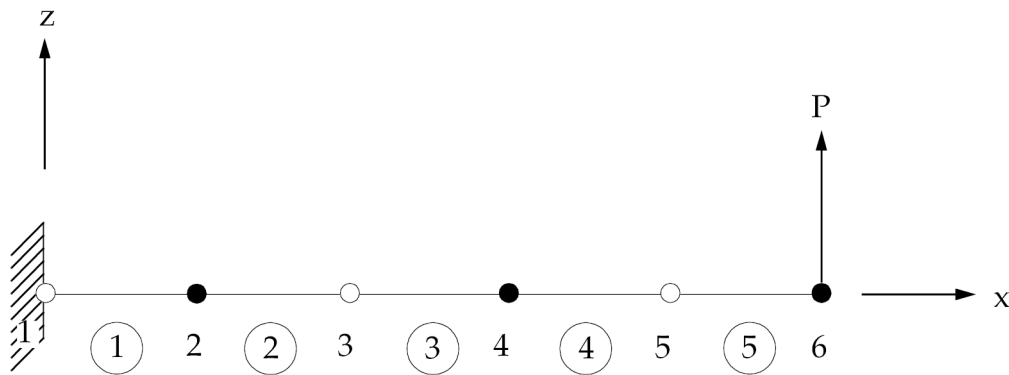


Figure 10-1. Cantilever Beam Using Guyan Reduction

```

$ filename - bulkm.dat
$
CBAR 1 1 1 2 10
CBAR 2 1 2 3 10
CBAR 3 1 3 4 10
CBAR 4 1 4 5 10
CBAR 5 1 5 6 10
FORCE 10 6 100. 0. 0. 1.
$
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
GRID 3 2. 0. 0.
GRID 4 3. 0. 0.
GRID 5 4. 0. 0.
GRID 6 5. 0. 0.
GRID 10 0. 0. 10.
MAT1 1 7.1+10 .33 2700.
PBAR 1 1 2.654-3 5.869-7 5.869-7 9.9-7
SPC1 10 123456 1
$

```

Listing 10-1. Input File for Common Bulk Data Entries

The first run is a conventional run without any reduction (**Listing 10-2**). Since Guyan reduction isn't used, then, by default, all degrees of freedom that aren't otherwise constrained are assigned to the a-set. In other words, degrees of freedom 1 through 6 for grid points 2 through 6 are assigned to the a-set. Components 1 through 6 of grid point 1 are assigned to the s-set due to the SPC Case Control command and SPC1 Bulk Data entry.

```

$ filename - full.dat
id bar aset
sol 101
time 10
cend
$
spc = 10
disp = all
$
subcase 1
load = 10
$
begin bulk
$
include 'bulkm.dat'
enddata

```

Listing 10-2. Input File for Using Conventional Analysis

The second run uses Guyan reduction by specifying degrees of freedom 1 through 6 at grid points 2, 4, and 6 to the *a*-set by using the ASET1 entries ([Listing 10-3](#)). Due to this action, the remaining unconstrained degrees of freedom (DOF 1 through 6 for grid points 3 and 5) are assigned to the *o*-set. Components 1 through 6 of grid point 1 are assigned to the *s*-set due to the SPC Case Control command and SPC1 Bulk Data entry.

```

$ filename - aset.dat
id bar aset
sol 101
time 10
cend
$
title =
subtitle =
spc = 10
disp = all
$
subcase 1
load = 10
$
begin bulk
$
aset1,123456,2,4,6
$
include 'bulkm.dat'
enddata

```

Listing 10-3. Input File Specifying A-Set DOFs

The third run is similar to the second run except the omitted degrees of freedom are specified rather than the analysis degrees of freedom ([Listing 10-4](#)). Degrees of freedom 1 through 6 for grid points 3 and 5 are assigned to the *o*-set by using the OMIT1 entries. Due to this action, degrees of freedom 1 through 6 for grid points 2, 4, and 6 are automatically assigned to the *a*-set. Components 1 through 6 of grid point 1 are assigned to the *s*-set due to the SPC Case Control command and SPC1 Bulk Data entry.

```

$ filename - omit.dat
id bar omit
sol 101
time 10
cend
$
title =
subtitle =
spc = 10
disp = all
$
subcase 1
load = 10
$
begin bulk
$
omit1,123456,3,5
$
include 'bulkm.dat'
enddata

```

Listing 10-4. Input File Specifying O-Set DOFs

Since Guyan reduction is numerically exact in static analysis, all three runs should yield the same results. **Figure 10-2** summarizes the results from the three runs—they are identical, as expected.

***** PARTIAL OUTPUT FROM MODEL WITHOUT USING GUYAN REDUCTION *****								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	.0
10	G	.0	.0	.0	.0	.0	.0	.0

***** PARTIAL OUTPUT FROM MODEL WITH GUYAN REDUCTION USING ASET ENTRIES *****								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	.0
10	G	.0	.0	.0	.0	.0	.0	.0

***** PARTIAL OUTPUT FROM MODEL WITH GUYAN REDUCTION USING OMIT ENTRIES *****								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	.0
10	G	.0	.0	.0	.0	.0	.0	.0

Figure 10-2. Results Comparison With and Without Guyan Reduction

Note that the only differences in the three input files are the ASET1/OMIT1 entries. The common Bulk Data entries are inserted in the run stream by using the INCLUDE statement.

10.4 Sets for Dynamic Reduction

One of the more important applications of partitioning is Guyan reduction. This technique is a means for reducing the number of degrees-of-freedom used in dynamic analysis with minimum loss of accuracy. The basis is that a lot fewer grid points are needed to describe the inertia of a structure than are needed to describe its elasticity with comparable accuracy. The error in the approximation is small provided that the set of displacements used for dynamic analysis is judiciously chosen. Its members should be uniformly dispersed throughout the structure, and all large mass items should be connected to grid points that are members of the analysis set.

Importantly, the matrix operations associated with this partitioning procedure tend to both create nonzero terms and fill what were previously very sparse matrices. The partitioning option is most effectively used if the members of the omitted set are either a very large fraction or a very small fraction of the total set. In most of the applications, the omitted set is a large fraction of the total and the matrices used for analysis, while small, are usually full. If the analysis set isn't a small fraction of the total, a solution using the large, but sparser, matrices may be more efficient.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

$$\begin{aligned}\xi_o &= \text{rigid body (zero frequency) modal degrees-of-freedom} \\ \xi_f &= \text{finite frequency modal degrees-of-freedom} \\ \xi_i &= \xi_o + \xi_f, \text{ the set of all modal degrees-of-freedom}\end{aligned}$$

One vector set is defined that combines physical and modal degrees-of-freedom:

$$u_h = \xi_i + u_e, \text{ the set of all physical and modal degrees-of-freedom}$$

In dynamic analysis, the a-set is further partitioned into the q-, r-, c-, and b-subsets.

- The q-set stores the coefficients for the generalized coordinates determined by dynamic reduction and/or component mode calculations.
- The r-set is again used to determine rigid body modes.
- The c-set contains coordinates considered free to move while computing the generalized coordinate functions.
- The b-set contains coordinates fixed during this process.

The d-set ("dynamic") for direct formulations is formed from the union of the structural degrees-of-freedom in the a-set and the extra points of the e-set. The d-set can be used to input unsymmetrical terms into the equations of motion using DMIG and TF Bulk Data entries. In the modal formulations, the h-set is composed of the union of the c_i (modal) set and the e-set. Again, you can use the DMIG and TF Bulk Data entries to introduce unsymmetrical matrix coefficients.

See also

- “Dynamic Reduction” in the *NX Nastran Basic Dynamic Analysis User's Guide*
- “The Set Notation System Used in Dynamic Analysis” in the *NX Nastran Basic Dynamic Analysis User's Guide*
- “**Direct Matrix Input**” in the *NX Nastran User's Guide*

10.5 Sets for Aerodynamics

Aerodynamic calculations are made in a Cartesian aerodynamic coordinate system. By convention, the flow is in the positive x direction. The basic coordinate system was not chosen, since it would place an undesirable restriction upon the description of the structural model. Any NX Nastran Cartesian system may be specified, and flow will be defined in the direction of its positive x-axis. All element and aerodynamic grid point data, computed initially in the basic coordinate system, will be converted to the aerodynamic coordinate system. The global (displacement) coordinate system of the aerodynamic grid points will have its T1 direction in the flow direction. T3 is normal to the element for boxes, and parallel to the aerodynamic T3 in the case of bodies. Coordinate system data are generated for the aerodynamic grid points.

The grid points are physically located at the centers of the boxes and body elements. Permanent constraints are generated for the unused degrees-of-freedom. A second set of grid points, used only for undeformed plotting, is located at the element corners. All six degrees-of-freedom associated with each grid point in this second set are permanently constrained. Grid point numbers are generated based upon the element identification number. For any panel, the external grid point numbers for the boxes start with the panel identification number and increase consecutively.

Aerodynamic degrees-of-freedom, along with the extra points, are added after the structural matrices and modes have been determined.

See also

- “**Aerodynamic Data Input and Generation**” in the *NX Nastran Aeroelastic Analysis User's Guide*

10.6 Output Selection Via Set Specification

The membership of grid points and their components and scalar points in the various sets is stored in the USET table. In NX Nastran, sets are defined by their name. For example, if you want the TABPRT module to print the M, R, N, SG, K, and SA sets, then specify:

```
PARAM, USETSTR1, M:R:N:SG
PARAM, USETSTR2, K:SA.
```

The data block naming convention in the solution sequences for matrices related to the sets is based on the set names. For example, the system stiffness matrix $[K_{gg}]$ has the name KGG. Some sets are used that do not appear in the USET table. The h-set, for example, is the set used for modal formulations. Its size is not known until after the USET table is formed, so it does not appear in the table. The j-set, a set synonymous with the g-set, is used in superelement analysis. There are several other sets used that also do not appear in the USET table.

Some names may be used for other purposes. You can input the name H, for example, to the MATGPR module when you want to label rows or columns of matrices sequentially, rather than by their external sequence numbers. This feature may be used to number vectors in load matrices or in other instances not related to modal analysis, even though “H” is defined for modal analysis.

Chapter 11: Direct Matrix Input

- *Overview of Direct Matrix Input*
- *Using DMIG*
- *DMIG Examples*

11.1 Overview of Direct Matrix Input

You can use the Bulk Data entry DMIG to input a stiffness (or mass) matrix which connects specified degrees-of-freedom. The matrix so defined will be added to the stiffness (or mass) matrix computed from finite element properties.

The DMIG entry includes provisions for unsymmetrical terms and complex values, both of which are useful in dynamic analysis. These provisions should not be used in static or normal modes. Note that an entry in the Case Control Section is required (K2GG = NAME for a stiffness matrix or M2GG = NAME for a mass matrix).

The primary application of the DMIG Bulk Data entry is to enter stiffness and mass data for parts of the structure which are obtained from another computer run. The format is cumbersome (two matrix terms per continuation entry) and the matrix should be input to high precision. For stiffness matrices only, the GENEL Bulk Data entry is an alternative for manually inputting data.

The finite element approach simulates the structural properties with mathematical equations written in matrix format. Once you provide the grid point locations, element connectivities, cross-sectional properties, material properties, applied loads, and boundary conditions, NX Nastran then automatically generates the appropriate structural matrices. The structural behavior is then obtained by solving these equations.

If these structural matrices are available externally, you can input these matrices directly without providing all the modeling information. Normally this is not a recommended procedure since it requires additional effort. However, there are occasions where the availability of this feature is very useful and in some cases is extremely crucial. Some possible applications are listed below:

- Suppose you are a subcontractor to a classified project. The substructure that you are analyzing is attached to the main structure built by the primary contractor. The flexibility of this main structure is crucial to the response of your component, but the geometry of the main structure is classified. The main contractor, however, can provide you with the stiffness matrix of the classified structure. By reading in this stiffness matrix and adding it to your NX Nastran model, you can account for the flexibility of the attached structure without compromising the security. The stiffness matrix is the inverse of the flexibility matrix.
- Suppose you are investigating a series of design options on a component attached to an aircraft bulkhead. Your component consists of 500 DOFs and the aircraft model consists of 100,000 DOFs. The flexibility of the backup structure is somewhat important. You can certainly analyze your component by including the full aircraft model (100,500 DOFs). On the other hand, if the flexibility at the attachment points on the aircraft can be measured experimentally, then you can add the experimental backup structure stiffness to your component without including the whole aircraft model. The experimental backup structure stiffness matrix is the inverse of the measured flexibility matrix. This way your model size remains at 500 DOFs, and you still have a good approximation of the backup structure stiffness.
- The same concept can be applied to a component attached to a test fixture. The stiffness of the fixture at the attachment locations can be read in as a stiffness matrix. Once again, the experimental test fixture stiffness matrix at the attachment points is the inverse of the measured flexibility at these points.

There are several ways that these matrices can be read in, such as DMIG, GENEL, and INPUTT4. Only DMIG is covered here. The DMIG and the INPUTT4 options offer alternate methods for inputting

large matrices. Note that INPUTT4 provides more precision than the DMIG input; the DMIG yields more precision than the GENEL on a short word machine.

See also

- “General Element Capability (GENEL)” in the *NX Nastran Element Library*
- “INPUTT4” in the *NX Nastran DMAP Programmer's Guide*

11.2 Using DMIG

You can use the DMIG feature to directly input a stiffness, mass, or loading matrix to the grid and/or scalar points in static analysis. These matrices are referenced in terms of their external grid point IDs and DOFs. The symbols for these *g*-type matrices in standard mathematical format are

$$[K_{gg}^2], [M_{gg}^2], \text{ and } [P_g^2]$$

All three matrices must be real. Furthermore,

$$[K_{gg}^2] \text{ and } [M_{gg}^2]$$

must also be symmetric. These matrices are implemented on the *g*-set level. In other words, these terms are added to the finite element model at the specified DOFs prior to the application of constraints. This section focuses on the DMIG features that are used in static analysis. The DMIG has additional features that can be used in dynamic analysis.

See also

- “DMIGs, Extra Points, and Transfer Functions” in the *NX Nastran Advanced Dynamic Analysis User's Guide*

DMIG Bulk Data User Interface

In the Bulk Data section, the DMIG matrix is defined by a single DMIG header entry followed by a series of DMIG data entries. Each of these DMIG data entries contains a column containing the nonzero elements for the particular matrix that you are entering.

Header Entry Format

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT			NCOL	

Field

Contents

NAME

Name of the matrix. It consists of one to eight alphanumeric characters, the first of which must be alphabetic. (Required)

IFO

Form of matrix input: Integer, required

1 = square

Field	Contents
	9 or 2 = rectangle
	6 = symmetric
TIN	Type of matrix that is input: Integer, required
	1 = real, single precision
	2 = real, double precision
TOUT	Type of matrix that is created: Integer
	0 = set by your machine precision (default)
	1 = real, single precision
	2 = real, double precision
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0).

Remarks

1. One header entry is required for each input matrix.
2. The matrix name used in field 2 must be unique.
3. Field 3 of the header entry must contain an integer 0.
4. The default value of 0 for TOUT is recommended.
5. If the input matrix is rectangular, using IFO = 9 is recommended instead of IFO = 2.

Column Data Entry Format

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	GJ	CJ		G1	C1	A1		
	G2	C2	A2		-etc.-				

Field	Contents
NAME	This is the same name that is used with its corresponding header entry as described above.
GJ	The external grid or scalar point identification number for the column. (Integer > 0).
CJ	The component number for the GJ grid point. (0 < Integer ≤ 6 if GJ is a grid point; blank or zero if GJ is a scalar point).
Gi	The external grid or scalar point identification number for the row. (Integer > 0).

Field	Contents
Ci	The component number for the Gi grid point. ($0 < \text{Integer} \leq 6$ if Gi is a grid point; blank or zero if Gi is a scalar point).
Ai	The value of the matrix term.

Remarks

1. Each non-null column (a column with at least one nonzero term) starts with a GJ, CJ pair. The entries (Gi, Ci pairs) for each row of that column follow. The number of Gi, Ci pairs equals the number of nonzero terms in that column. The terms may be input in arbitrary order. High precision is required when using DMIG; therefore, in general, the use of large field input is recommended. Each non-null column of your matrix makes up a DMIG column data entry. Therefore, the total number of DMIG entries is equal to the number of non-null columns plus one. The extra entry is the DMIG header entry.
2. For symmetric matrices (IFO = 6), you can input a given off-diagonal term either below or above the diagonal. While the upper and lower triangle terms may be mixed, a fatal error results if the corresponding symmetric element is input both below and above the diagonal. However, to keep it simple, for a given matrix, you should decide to input the terms either completely below the diagonal or above the diagonal, but not both. The symmetric option is generally used for the mass and stiffness matrices.
3. Rectangular matrices can be input using the option IFO = 9. This option is generally used to input load matrices.

See also

- “**DMIG**” in the *NX Nastran Quick Reference Guide*

DMIG Case Control User Interface

In order to include the direct input matrices you define with DMIG, the Case Control Section must contain the appropriate K2GG, M2GG, B2GG, K42GG, or P2G command.

See also

- “**K2GG**” in the *NX Nastran Quick Reference Guide*
- “**M2GG**” in the *NX Nastran Quick Reference Guide*
- “**B2GG**” in the *NX Nastran Quick Reference Guide*
- “**K42GG**” in the *NX Nastran Quick Reference Guide*
- “**P2G**” in the *NX Nastran Quick Reference Guide*

Examples

```
K2GG = mystif
```

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “mystif”, to the *g*-set stiffness matrix.

```
M2GG = yourmass
```

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “yourmass”, to the *g*-set mass matrix.

```
B2GG = mydamp
```

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “mydamp”, to the *g*-set viscous damping matrix.

```
K42GG = strdamp
```

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “strdamp”, to the *g*-set structural damping matrix.

```
P2G = hisload
```

The above Case Control command adds the terms of the matrix, which are defined by the DMIG entries with a name of “hisload”, to the *g*-set load matrix.

11.3 DMIG Examples

The following examples of using DMIG features use the cantilever beam model shown in [Figure 11-1](#).

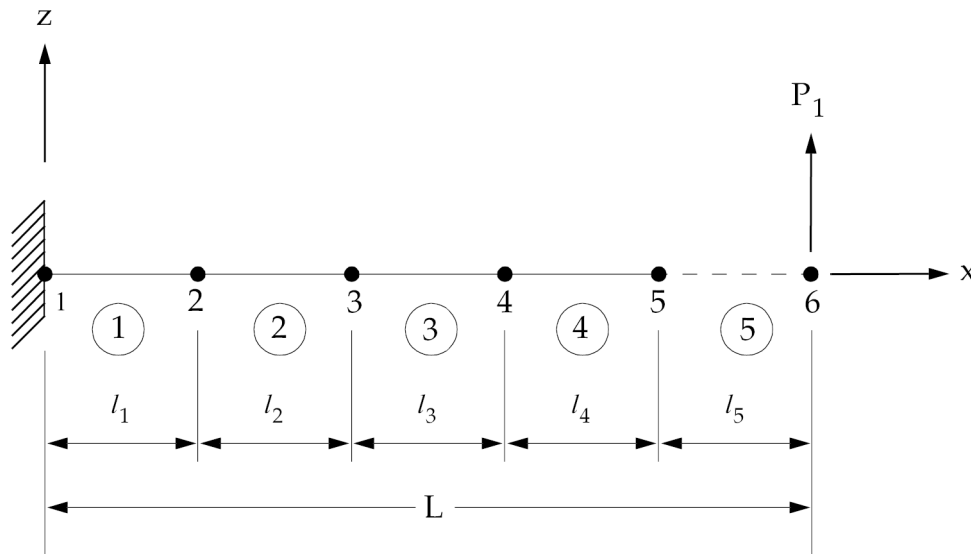


Figure 11-1. Planer Cantilever Beam

Stiffness Matrix Using DMIG

First, we must analyze this problem using the conventional method with two loading conditions—a vertical tip load and a gravity load. [Listing 11-1](#) contains the input file for this run. Note that for illustrative purposes, a planar problem (xz plane) is shown. However, this same concept is applicable to a general three-dimensional structure.

```

$  FILENAME - DMIGFULL.DAT
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = PLANAR CANTILEVER BEAM
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
SUBCASE 2
LABEL = GRAVITY LOAD
LOAD = 20
$
BEGIN BULK
$
CBAR 1 1 1 2 10
CBAR 2 1 2 3 10
CBAR 3 1 3 4 10
CBAR 4 1 4 5 10
CBAR 5 1 5 6 10
FORCE 10 6 100. 0. 0. 1.
GRAV 20 9.8 0. 0. 1.
GRID 1 0. 0. 0. 246
GRID 2 1. 0. 0. 246
GRID 3 2. 0. 0. 246
GRID 4 3. 0. 0. 246
GRID 5 4. 0. 0. 246
GRID 6 5. 0. 0. 246
GRID 10 0. 0. 10. 123456
MAT1 1 7.1+10 .33 2700.
PBAR 1 1 2.654-3 5.869-7
SPC1 10 123456 1
$
ENDDATA

```

Listing 11-1. Input File for Conventional Analysis

The next run removes CBAR element number 5 from the input file and uses the DMIG feature to directly read in the bar element stiffness matrix. The bar element stiffness matrix for a planar element and ignoring the axial stiffness is as follows:

$$[K]_e = \frac{2EI}{l^3} \begin{bmatrix} 6 & -3l & -6 & -3l \\ -3l & 2l^2 & 3l & l^2 \\ -6 & 3l & 6 & 3l \\ -3l & l^2 & 3l & 2l^2 \end{bmatrix}$$

Since the matrix is symmetric, either the lower or upper triangular portion of the matrix must therefore be provided. For $E = 7.1 \cdot 10^{10} \text{ N/m}^2$, $I = 5.869 \cdot 10^{-7} \text{ m}^4$, and $l_5 = 1 \text{ m}$, the stiffness matrix for CBAR element number 5 is as follows:

$$[K]_e = \begin{matrix} & \begin{matrix} 5, 3 & 5, 5 & 6, 3 & 6, 5 \end{matrix} \\ \begin{matrix} 5, 3 \\ 5, 5 \\ 6, 3 \\ 6, 5 \end{matrix} & \begin{bmatrix} 500038.3 & & & \\ -250019.4 & 166679.6 & & \text{SYM} \\ -500038.8 & 250019.4 & 500038.8 & \\ -250019.4 & 83339.8 & 250019.4 & 166679.6 \end{bmatrix} \end{matrix}$$

In this case, you chose UGSTIF as the name of the input stiffness for bar element number 5. Therefore, in order to bring in this stiffness matrix and add it to the global stiffness matrix, the Case Control command K2GG= UGSTIF is required. Note these stiffness terms are additions to the global stiffness matrix at that location and are not a replacement of the stiffness terms at that location. In the Bulk Data Section, five DMIG entries are required—one for the header entry and four for the data column entries—since there are four non-null columns in the above matrix.

For the header entry, the same name UGSTIF must be used to match the name called out in the Case Control Section. The third field is 0, which must be the value used for the header entry. The fourth field (IFO) is set to 6 to denote a symmetric matrix input. The fifth field (TIN) is set to 1 to denote that the matrix is provided as real, single precision.

Once again, the terms in the matrix are referenced in terms of their external grid IDs when using the DMIG feature. Physically, each term in a particular column of the stiffness matrix (k_{ij}) represents the induced reactive load in the i -th degree of freedom due to a unit displacement in the j -th direction with all other displacement degrees of freedom held to zero. Since the matrix is symmetric, only the lower triangular portion of the matrix is read.

The first DMIG data column entry reads the first column of the above matrix. Field 2 of this DMIG entry must have the same name UGSTIF as referenced by the Case Control K2GG = UGSTIF command. Fields 3 and 4 of this entry identify this column in terms of its external grid ID and corresponding degree of freedom, respectively. In this case, it is grid point number 5, degree of freedom 3 (z-translation at grid number 5).

Once this column is defined and by following the format description as described in the section for column data entry format, you can then input the four terms in this column row by row. These four terms are defined by sets of three fields. They are the external grid point ID number, corresponding degree of freedom, and the actual matrix term, respectively. The first row of column one is defined by external grid point ID 5, degree of freedom 3 (z-translation at grid point number 5) with a stiffness value of 500039. The second row of column one is defined by external grid point ID 5, degree of freedom 5 (y-rotation at grid point number 5) with a stiffness value of -250019. The third row of column one is defined by external grid point ID 6, degree of freedom 3 (z-translation at grid point number 6) with a stiffness value of -500039. The fourth row of column one is defined by external grid point ID 6, degree of freedom 5 (y-rotation at grid point number 6) with a stiffness value of -250019.

The next DMIG entry reads the second column of the above matrix starting with the diagonal term. Fields 3 and 4 of this entry identify this column in terms of its external grid point ID and corresponding degree of freedom, respectively. In this case, it is grid number 5, degree of freedom 5 (y-rotation at grid point number 5). The rest of the procedure is similar to that of column one with the exception that only three rows need to be read due to symmetry (rows two through four).

The next two DMIG entries read columns three and four of the stiffness matrix, respectively. Note that due to symmetry, one less row needs to be read for each additional column.

The input file using DMIG is contained in [Listing 11-2](#). The results as compared with the conventional analysis are tabulated in the first two sections of [Figure 11-2](#). The comparison is performed for Subcase 1 with a vertical load applied to the tip of the cantilever beam. In this case, the results are accurate to within three to four digits.

As mentioned previously, a high degree of precision should be retained when inputting a matrix with the DMIG method. In this case, since the small-field input format is used, the maximum number of characters that can be used are eight (including sign and decimal place). The same job was then rerun for the DMIG input with additional significant digits added using double-precision and large-field format ([Listing 11-3](#)). In this case, the results are identical by comparing the results of the first and last sections of [Figure 11-2](#). Note that even the double-precision input on short-word machines does not contain all of the significant digits of the machine.

```

$  FILENAME - DMIGSTFS.DAT
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = PLANER CANTILEVER BEAM
SUBTITLE = USE DMIG TO BRING IN STIFFNESS MATRIX OF ELEMENT # 5 USING SMALL FIELD
K2GG = UGSTIF
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
$  DMIG HEADER ENTRY
$
DMIG    UGSTIF  0      6      1
$
$  DMIG DATA COLUMN ENTRIES
$
DMIG    UGSTIF  5      3      5      3      500039.      +000001
++0000015      5      -250019.      6      3      -500039.      +000002
++0000026      5      -250019.      +000003
$
DMIG    UGSTIF  5      5      5      5      166680.      +000004
++0000046      3      250019.      6      5      83340.      +000005
$
DMIG    UGSTIF  6      3      6      3      500039.      +000006
++0000066      5      250019.      +000007
$
DMIG    UGSTIF  6      5      6      5      166680.
$
FORCE   10      6      100.      0.      0.      1.
GRAV    20      9.8      0.      0.      1.
GRID    1      0.      0.      0.      246
GRID    2      1.      0.      0.      246
GRID    3      2.      0.      0.      246
GRID    4      3.      0.      0.      246
GRID    5      4.      0.      0.      246
GRID    6      5.      0.      0.      246
GRID    10     0.      0.      10.      123456
MAT1    1      7.1+10      .33      2700.
PBAR    1      1      2.654-3  5.869-7
SPC1    10     123456  1
$
ENDDATA

```

Listing 11-2. Input File for Reading a Stiffness Matrix Using DMIG with Small-Field Format

```

$  FILENAME - DMIGSTFL.DAT
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
TITLE = PLANER CANTILEVER BEAM
SUBTITLE = DMIG TO BRING IN STIFFNESS MATRIX FOR ELEMENT # 5 USING LARGE FIELD
K2GG = UGSTIF
SPC = 10
DISP = ALL
$
SUBCASE 1
LABEL = TIP LOAD AT END
LOAD = 10
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
$
$  DMIG HEADER ENTRY
$
DMIG    UGSTIF  0      6      2
$
$  DMIG DATA COLUMN ENTRIES
$
DMIG*   UGSTIF      5      3      *A
*A      5      3      5.000388 D+5      *B
*B      5      5      -2.500194 D+5      *C
*C      6      3      -5.000388 D+5      *D
*D      6      5      -2.500194 D+5
$
DMIG*   UGSTIF      5      5      *A2
*A2     5      5      1.666796 D+5      *B2
*B2     6      3      2.500194 D+5      *C2
*C2     6      5      8.33398 D+4
$
DMIG*   UGSTIF      6      3      *A3
*A3     6      3      5.000388 D+5      *B3
*B3     6      5      2.500194 D+5
$
DMIG*   UGSTIF      6      5      *A4
*A4     6      5      1.666796 D+5
$
FORCE   10      6      100.    0.    0.    1.
GRAV    20      9.8    0.    0.    1.
GRID    1      0.    0.    0.    246
GRID    2      1.    0.    0.    246
GRID    3      2.    0.    0.    246
GRID    4      3.    0.    0.    246
GRID    5      4.    0.    0.    246
GRID    6      5.    0.    0.    246
GRID    10     0.    0.    10.    123456
MAT1    1      7.1+10 .33    2700.
PBAR    1      1      2.654-3 5.869-7
SPC1    10     123456 1
ENDDATA

```

Listing 11-3. Input File for Reading a Stiffness Matrix with Large-Field Format

***** PARTIAL OUTPUT FROM DMIGFULL.F06 - FULL MODEL *****								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

***** PARTIAL OUTPUT FROM DMIGSTIFS.F06 - DMIG TO INPUT STIFFNESS FOR ELEM # 5 USING SMALL FIELD								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	5.598505E-03	.0	-1.079704E-02	.0	
3	G	.0	.0	2.079415E-02	.0	-1.919427E-02	.0	
4	G	.0	.0	4.318711E-02	.0	-2.519168E-02	.0	
5	G	.0	.0	7.037757E-02	.0	-2.878928E-02	.0	
6	G	.0	.0	9.996621E-02	.0	-2.998813E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

***** PARTIAL OUTPUT FROM DMIGSTIFL.F06 - DMIG TO INPUT STIFFNESS FOR ELEM # 5 USING LARGE FIELD								
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0	
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0	
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0	
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0	
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0	
10	G	.0	.0	.0	.0	.0	.0	

Figure 11-2. Results Comparison Between the Conventional Method Versus DMIG

Mass Matrix Using DMIG

You can also include a mass matrix with DMIG in a similar manner as the stiffness matrix with the following exceptions:

- You use the Case Control command M2GG = "name", instead of K2GG= "name," where "name" is the name of the mass matrix referenced in field 2 of the DMIG Bulk Data entries.
- The matrix defined in the DMIG entries represents the mass matrix terms instead of the stiffness matrix terms.

Viscous Damping Matrix Using DMIG

You can also include a viscous damping matrix with DMIG in a similar manner as the stiffness matrix with the following exceptions:

- You use the Case Control command B2GG = "name", instead of K2GG= "name," where "name" is the name of the viscous damping matrix referenced in field 2 of the DMIG Bulk Data entries.

- The matrix defined in the DMIG entries represents the viscous damping matrix terms instead of the stiffness matrix terms.

Structural Damping Matrix Using DMIG

You can also include a structural damping matrix with DMIG in a similar manner as the stiffness matrix with the following exceptions:

- You use the Case Control command K42GG = "name", instead of K2GG = "name," where "name" is the name of the structural damping matrix referenced in field 2 of the DMIG Bulk Data entries.
- The matrix defined in the DMIG entries represents the structural damping matrix terms instead of the stiffness matrix terms.

Load Matrix Using DMIG

The load vectors can also be read using the DMIG feature if desired, although this is not a commonly used feature. It is similar to the format for reading in the stiffness or mass matrix, with the following exceptions:

- You use the Case Control command P2G = "name", instead of K2GG = "name" or M2GG = "name."
- The matrix defined in the DMIG entries are columns of the load vectors instead of the stiffness or mass matrix terms.
- The load matrix, in general, is rectangular and unsymmetrical (i.e., IFO = 9 on the header entry).
- The column number (GJ) you specify on the DMIG input must match the load sequence number. The GJ field is the load sequence number, and the CJ field isn't used at all for a rectangular matrix. For example, if there is a total of ten load conditions in your model and you want to add load vectors to the third and seventh subcases using DMIG, then the NCOL field should be assigned as 10 on the header entry. Two additional DMIG Data entries, representing the two load vectors added to the third and seventh subcases, are needed. The GJ field for the first of these two data entries must have a value of 3 to denote the third load sequence. The remaining data for this entry are the nonzero rows of this load vector. Their input format is identical to that of the stiffness or mass matrix. The GJ field for the second of these two data entries must have a value of 7 to denote the seventh load sequence number. Again, the remaining data for this entry are the nonzero rows of this load vector.

Again the problem in [Figure 11-1](#) is used to illustrate this feature. The input file is shown in [Listing 11-4](#). In this example, the DMIG feature is used to read in the load for the vertical tip load in Subcase 10. In the Case Control Section, the P2G = UGLOAD command is used to activate the DMIG entries with the name of UGLOAD. In the Bulk Data Section, the DMIG load header entry is similar to that of the stiffness and mass header entry with the exception of the IFO and NCOL fields. In general, the matrix is rectangular; therefore, IFO is defined as 9 instead of 6. NCOL is the total number of load cases for the problem.

Since there are two load cases, NCOL must be defined as 2. The number of DMIG load data entries is equal to the number of external load columns added. In this case, you only need one DMIG load data entry since only one load condition is added to the first subcase. The GJ field of 1 denotes that you are adding the load to the first subcase (first load sequence). Note that GJ references the

load sequence (1) rather than the SUBCASE ID (10). The CJ field should be blank when using the DMIG load feature. The rest of the fields are read in the same manner as the stiffness and mass matrices—only nonzero entries need to be defined. Note that if you have reversed your subcase setup—that is, if you apply your gravity load as the first subcase and bring in the tip load as the second subcase—then field three of the second DMIG entry must then be modified to 2 instead of 1.

```

$   FILENAME - dmigload.dat
ID BAR DMIG
SOL 101
TIME 10
DIAG 8
CEND
$
TITLE = USE DMIG TO BRING IN LOAD VECTOR FOR SUBCASE 1
P2G = UGLOAD
SPC = 10
DISP = ALL
$
SUBCASE 10
LABEL = TIP LOAD AT END
$
SUBCASE 20
LABEL = GRAVITY LOAD
LOAD = 20
$
BEGIN BULK
$
CBAR    1      1      1      2      10
CBAR    2      1      2      3      10
CBAR    3      1      3      4      10
CBAR    4      1      4      5      10
CBAR    5      1      5      6      10
$
DMIG    UGLOAD  0      9      1      6      3      100.      2
DMIG    UGLOAD  1
$
GRAV    20      9.81    0.      0.      1.
GRID    1      0.      0.      0.      246
GRID    2      1.      0.      0.      246
GRID    3      2.      0.      0.      246
GRID    4      3.      0.      0.      246
GRID    5      4.      0.      0.      246
GRID    6      5.      0.      0.      246
GRID    10     0.      0.      10.     123456
MAT1    1      7.1+10   .33     2700.
PBAR    1      1      2.654-3  5.869-7
SPC1    10     123456  1
$
ENDDATA

```

Listing 11-4. Input File for Reading in a Load Vector Using DMIG

***** PARTIAL OUTPUT FROM DMIGFULL.F06 - FULL MODEL *****							
TIP LOAD AT END				DISPLACEMENT VECTOR			
				SUBCASE 1			
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0
GRAVITY LOAD				DISPLACEMENT VECTOR			
				SUBCASE 2			
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	9.278418E-03	.0	-1.729160E-02	.0
3	G	.0	.0	3.233388E-02	.0	-2.783526E-02	.0
4	G	.0	.0	6.326194E-02	.0	-3.331796E-02	.0
5	G	.0	.0	9.784514E-02	.0	-3.542669E-02	.0
6	G	.0	.0	1.335530E-01	.0	-3.584844E-02	.0
10	G	.0	.0	.0	.0	.0	.0
***** PARTIAL OUTPUT FROM DMIGLOAD.F06 - DMIG TO INPUT LOAD CONDITION # 1 *****							
LOAD CONDITION # 2 IS APPLIED BY THE CONVENTIONAL MANNER							
TIP LOAD AT END				DISPLACEMENT VECTOR			
				SUBCASE 10			
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	5.599566E-03	.0	-1.079916E-02	.0
3	G	.0	.0	2.079839E-02	.0	-1.919851E-02	.0
4	G	.0	.0	4.319665E-02	.0	-2.519804E-02	.0
5	G	.0	.0	7.039454E-02	.0	-2.879777E-02	.0
6	G	.0	.0	9.999225E-02	.0	-2.999767E-02	.0
10	G	.0	.0	.0	.0	.0	.0
GRAVITY LOAD				DISPLACEMENT VECTOR			
				SUBCASE 20			
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	9.278418E-03	.0	-1.729160E-02	.0
3	G	.0	.0	3.233388E-02	.0	-2.783526E-02	.0
4	G	.0	.0	6.326194E-02	.0	-3.331796E-02	.0
5	G	.0	.0	9.784514E-02	.0	-3.542669E-02	.0
6	G	.0	.0	1.335530E-01	.0	-3.584844E-02	.0
10	G	.0	.0	.0	.0	.0	.0

Figure 11-3. Results Comparison Between Conventional Load Input Versus DMIG

Inputting large matrices by hand using the DMIG method can be time consuming, error prone, and quite tedious. However, in most instances, the DMIG input may be automatically generated by a DMAP module when using NX Nastran or other external programs. You can also use PARAM,EXTOUT,DMIGPCH to generate these DMIG entries.

See also

- “EXTOUT” in the *NX Nastran Quick Reference Guide*

Chapter 12: Element Data Recovery Resolved at Grid Points

- *Introduction to Element Data Recovery*
- *Stress Recovery at Grid Points*
- *Mesh Stress Discontinuities at Grid Points*

12.1 Introduction to Element Data Recovery

Element stresses may be averaged at grid points and then used to calculate estimates of stress discontinuities in the model mesh. Resolving element data at the grid points is supported only for selected linear elastic element types (see [Table 12-1](#) for supported element types). The calculation of average stresses at grid points and the calculation of estimates of stress discontinuities use a common mesh definition method.

You define the mesh in the OUTPUT(POST) section of the Case Control section using one of two methods:

- SURFACE input for 2-D shell element meshes
- VOLUME input for 3-D solid element meshes

The averaging of grid point stresses (and strains) and stress discontinuities are calculated independently for meshes composed of only 2-D shell elements and those composed of only 3-D solid elements. You use the STRESS Case Control command to request the element stress and strain data used in the calculation of grid point stresses and stress discontinuities. To substitute the calculation of strains for stresses, use the STRAIN Case Control command in place of the STRESS command in the Case Control Section.

You request grid point stresses for reports with the GPSTRESS Case Control command. The calculation of stress discontinuities is supported for both elements and grid points associated with an element mesh by the ELSDCON and GPSDCON commands, respectively.

The following sections explain in more detail the requesting and calculation of grid point stresses and stress discontinuities, respectively.

Table 12-1. Elements Types Supported in Element Data Resolved at Grid Point Calculations (Hyperelastic Elements Not Included)			
Element Type	2-D Shell Stress	3-D Solid Stress	Stress Discontinuity
CQUAD4	X		X
CQUADR	X		X
CQUAD8	X		X
CTRIA3	X		X
CTRIAR	X		X
CTRIA6	X		X
CHEXA		X	X
CPENTA		X	X
CPYRAM		X	X
CTETRA		X	X

See also

- “**STRESS**” in the *NX Nastran Quick Reference Guide*
- “**STRAIN**” in the *NX Nastran Quick Reference Guide*
- “**ELSDCON**” in the *NX Nastran Quick Reference Guide*

- “GPSDCON” in the *NX Nastran Quick Reference Guide*

12.2 Stress Recovery at Grid Points

You can have NX Nastran compute stresses at grid points:

- On surfaces of two-dimensional plate elements, namely, the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, and CTRIA6
- In volumes containing CHEXA, CPENTA, CPYRAM and CTETRA solid elements

By default, NX Nastran outputs stresses at the center of each of these elements and also at the vertices of CQUADR, CQUAD8, CTRIAR, and CTRIA6 shell elements and the CHEXA, CPENTA, CPYRAM and CTETRA solid elements. However, the center output isn't always adequate. For example, when you're working with the surfaces of CQUAD4 and/or CTRIA3 elements, you may also need realistic stress values at the grid points. Also, in the case of surfaces of CQUADR, CQUAD8, CTRIAR, and CTRIA6 elements or of volumes containing CHEXA, CPENTA, CPYRAM and CTETRA elements, the vertex stresses output for the elements that connect to a common grid point are often not identical.

This section describes a rational method that yields accurate and unique stresses at each vertex grid point in user-defined surfaces or volumes. Grid point stress recovery is only available in linear static analysis, real eigenvalue analysis, and transient analysis. Grid point stress recovery isn't available for lamina stresses.

You can also use NX Nastran to estimate the probable error in these grid point stresses (see “[Error Estimates for Grid Point Stress Data](#)”). Estimates are generated for each stress component. The root mean square error of these estimates is then computed to provide a single measure of the error in the computation of the stresses at a grid point. This data provides a gross indication of the adequacy of a finite element model to represent the physical phenomena under investigation. The error estimator is available in SOLution Sequence 101.

Description of Method

A scheme to interpolate and/or extrapolate over a surface from a known set of stresses (the element stresses) requires that these known stress components be transformed into a consistent coordinate system prior to interpolation and extrapolation. This coordinate system is defined by the analyst and it should be generally compatible with the surface of interest in order to obtain accurate grid point stresses. Generally, such a coordinate system will be the natural coordinate system that the analyst used in the generation of the model of the surface or volume in question. Calculated grid point stress components are output in this coordinate system, called the output coordinate system of the surface or volume.

Each stress component is treated independently of other stress components in the calculation of average grid point stress components. The stress invariants at grid points (i.e., principal stresses, the von Mises stress, and the mean pressure) are evaluated from the average values of the stress components at grid points. The grid point stress components are output in the output coordinate system of the surface or volume and the stress invariants are oriented relative to this output coordinate system.

Two methods are used to calculate grid point stress components for plate and shell elements: topological and geometric. A single, simplified, topological method is used to calculate grid point stress components in volumes containing solid elements. However, prior to a discussion of these methods, it is necessary to define the four categories into which grid points are divided.

1. Interior Grid Points

Grid points that are connected only to interior line segments; i.e., line segments that are coincident with the edges of two or more elements.

2. Corner Grid Points

Grid points that are only connected to a single element.

3. Edge Grid Points

Grid points that are connected to both interior and exterior line segments. An exterior line segment is coincident with the edge of only one element.

4. Exception Grid Points

Grid points at which discontinuities in stress can occur. Exception points will be described more completely under “**Treatment of Exception Points – 'BRANCHing'.**”

Stresses at interior, corner, or edge grid points are considered to be continuous between directly connected elements. Stresses at exception points may be discontinuous between connected elements and, as will be seen, a different grid point stress may be output for each of the connected elements. You must exercise good engineering judgment when you assess the validity of grid point stresses at exception points.

Topological Method – Plate Elements and Solid Elements

The topological method for the evaluation of grid point stresses only recognizes the existence of interior, edge, and corner points; i.e., exception points are not considered as a distinct category of grid point (exception points arise as a result of geometrical rather than topological characteristics). The relations used to compute average grid point stresses in surfaces of plate elements by the topological method are described in **Table 12-2**. The relations used to compute the average grid point stresses in volumes containing solid elements are described in **Table 12-4**. Only the topological method is available for solid elements.

Table 12-2. Evaluation of Average Grid Point Stresses for Plate and Shell Elements by the Topological Method

Type of Grid Point	Average Grid Point Stress Component	Remarks
Interior	$\sigma_I = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ <p>where:</p> <p>σ_I = average stress at interior grid point</p> <p>N_e = number of directly connected elements</p> $\sigma_{ei} = \begin{cases} \text{element center stresses if only CQUAD4} \\ \text{and/or CTRIA3 elements are connected} \\ \text{element vertex stresses if CQUAD8 and/or} \\ \text{CTRIA6 are connected} \end{cases}$	If CQUAD4 and/or CTRIA3 elements connect to the same grid point as CQUAD8 and/or CTRIA6 elements, then only the CQUAD8 and or CTRIA6 elements are used in the calculation of average grid point stresses.
Edge	$\sigma_E = \frac{2}{N_e} \sum_{i=1}^{N_e} \sigma_{ei} - \frac{1}{N_I} \sum_{i=1}^{N_I} \sigma_{Ii}$ <p>where:</p> <p>σ_E = average stress at edge grid point</p> <p>N_I = number of interior points connected by line segments</p> <p>σ_{Ii} = average stress at interior grid points</p>	<p>If an edge point is not connected to an interior point by line segments, then $N_I = 0$, the second term in the relation σ_E is set to zero, and the factor 2 in the first term is replaced by 1.</p> <p>Same as remark for interior grid points.</p>
Corner	<p>Corner points connected to CQUAD4 element with the grid point diagonally opposite the corner point, an interior point, or an edge point.</p> $\sigma_c = 2\sigma_1 - \sigma_D$ <p>where:</p> <p>σ_c = average stress at corner grid point</p> <p>σ_1 = CQUAD4 center stress</p> <p>σ_D = average stress at grid point diagonally opposite corner point</p> <p>(If point diagonally opposite corner point is not an interior or edge point, then $\sigma_c = \sigma_1$.)</p>	Same as remark for interior points

Table 12-2. Evaluation of Average Grid Point Stresses for Plate and Shell Elements by the Topological Method

Type of Grid Point	Average Grid Point Stress Component	Remarks
Corner	<p>Corner points connected to CTRIA3 elements and the other two grid points, namely, E1 and E2, of the connected CTRIA3 are edge points:</p> $\sigma_c = 3\sigma_1 - (\sigma_{E1} + \sigma_{E2})$ <p>where:</p> <p>σ_c = average stress at corner grid point</p> <p>σ_1 = CTRIA3 center stress σ_{E1}, σ_{E2} = stresses at edge points E1 and E2</p> <p>(if either E1 or E2 is not an edge point, then σ_c)</p> <p>Corner points connected to one CQUAD4 or one CTRIA6 element:</p> $\sigma_c = \sigma_r$ <p>where:</p> <p>σ_r = stress at element vertex connected to corner point</p> <p>Corner points connected to two or more CTRIA6 elements:</p> $\sigma_c = \frac{1}{N_c} \sum_{i=1}^{N_c} \sigma_{r_i}$ <p>where:</p> <p>N_c = number of CTRIA6 elements connected to corner point.</p>	Same as remark for interior points

Table 12-3. Evaluation of Average Grid Point Stress for Solid Elements by the Topological Method

Type of Grid Point	Average Grid Point Stress Component	Remarks
Interior and Edge	$\sigma_I = \frac{1}{N_e} \sum_{i=1}^{N_e} \sigma_{ei}$ <p>where:</p> <p>σ_I = average stress at interior or edge grid points</p> <p>N_e = number of directly connected elements</p> <p>σ_{ei} = element vertex stresses</p>	Mixtures of CHEXA, CPENTA, CPYRAM and CTETRA elements may connect to an interior or edge grid point.
Corner	$\sigma_c = \sigma_r$ <p>where:</p> <p>σ_r = stress at element vertex connected to corner point</p>	The same relation is used whether a CHEXA, CPENTA, CPYRAM or CTETRA element is connected to the grid point.

Geometric Method – Surface Elements

In the geometric method, grid point stresses for points which connect to CQUAD8 or CTRIA6 elements use the average of the element corner stresses. If any CQUAD4 or CTRIA3 elements connect to these points, their influence is not considered.

The geometric method for the determination of average grid point stresses which connect only to CQUAD4 or CTRIA3 elements utilizes the numerical technique of least squares. The “best fit” value of the stress Σ at a point (x, y) is functionally denoted as:

$$\Sigma(x,y) = A_0 + A_1x + A_2y$$

Equation 12-1.

where the coefficients A_0 , A_1 , and A_2 must be determined. The coordinates x and y are the locations of points at which known stresses exist. In accordance with the least squares technique, it is required to minimize the function:

$$f = \frac{1}{2} \left\{ \sigma - \Sigma \right\}^T \left\{ \sigma - \Sigma \right\}$$

Equation 12-2.

with respect to $\{A\}$. Here σ represents the known element stresses.

To perform the minimization, Eq. 12-1 is written in matrix form as:

$$\left\{ \Sigma \right\} = [T] \{A\}$$

Equation 12-3.

where:

$$[T] = \begin{bmatrix} 1 & X_1 & X_2 \\ \vdots & \vdots & \vdots \\ i & X_i & X_i \end{bmatrix}$$

Equation 12-4.

and Eq. 12-2 is differentiated with respect to the coefficients A_i .

$$\frac{\partial f}{\partial A_i} = [T]^T [T] \{A\} - [T]^T \{\sigma\} = 0$$

Equation 12-5.

The coefficients $\{A\}$ are then determined from Eq. 12-5 to be:

$$\{A\} = ([T]^T [T])^{-1} [T]^T \{\sigma\}$$

Equation 12-6.

The average grid point stress at a given grid point can then be calculated by substituting Eq. 12-6 into the matrix representation of Eq. 12-1 as follows:

$$\Sigma = [1 \ X_o \ Y_o] ([T]^T [T])^{-1} [T]^T \{\sigma\} = [Q] \{\sigma\}$$

Equation 12-7.

where:

- Q = matrix of interpolation factors
- $X_o \ Y_o$ = locations of grid points at which stress is required

Note that if $([T]^T[T])^{-1}$ is singular, the matrix of interpolation factors is taken to be $\frac{1}{N_I}$ so that Eq. 12-7 becomes:

$$\Sigma = \frac{1}{N_I} \sum_{i=1}^{N_I} \sigma_{ei}$$

Equation 12-8.

where N_I = number of known stress points associated with the grid point. In most cases, Eq. 12-8 is equivalent to the topological interpolation method.

The strategy used in the evaluation of Eq. 12-7 for interior, edge, and corner grid points is presented in Table 12-4.

As previously mentioned, the coordinates x and y of Eq. 12-1 are the locations of points at which known stresses exist. These locations are actually taken as the projected coordinates of the known stress points on the average surface evaluated at the grid point at which the stress is to be computed. This average surface is determined to be the surface whose normal is the average of the unit normals to the elements connected to the grid point of interest. The local x -axis used in the evaluation of the geometric coefficients of the matrix $[T]$ is taken as the projection of the x -axis of the output coordinate system on the average surface evaluated at the grid point of interest.

Table 12-4. Evaluation of Average Grid Point Stresses for Plate and Shell Elements by the Geometric Method

Type of Grid Point	Number of Elements Involved in Stress Calculation
Interior	All directly connected elements
Edge	Two directly connected elements: Use the two known element stresses plus the stresses at all interior points connected by line segments of the directly connected elements.
	Three or more directly connected elements: Use the known element stresses of the directly connected elements.
Corner	Same as topological method (see Table 12-1).

Treatment of Exception Points – “BRANCHing”

Exception points have been defined as grid points at which stress discontinuities may occur. Such discontinuities may occur because a substantial difference in slope exists between adjacent elements directly connected to the same line segment or if more than two elements are directly connected to the same line segment. Examples of these two cases are illustrated below.

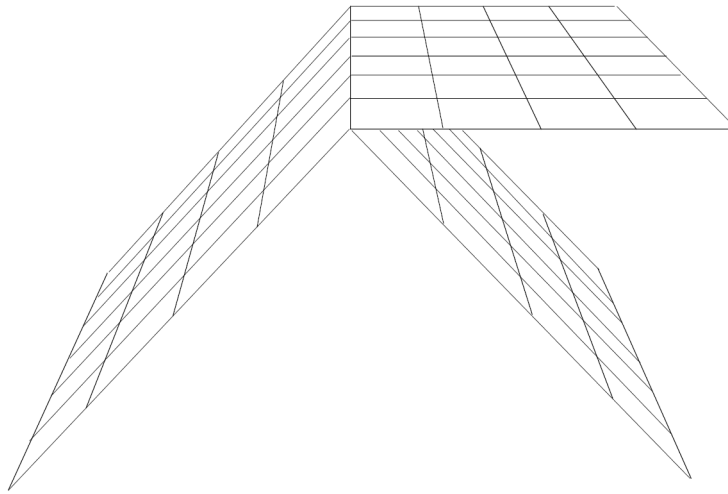


Figure 12-1. Three or More Elements Connected to the Same Line Segment

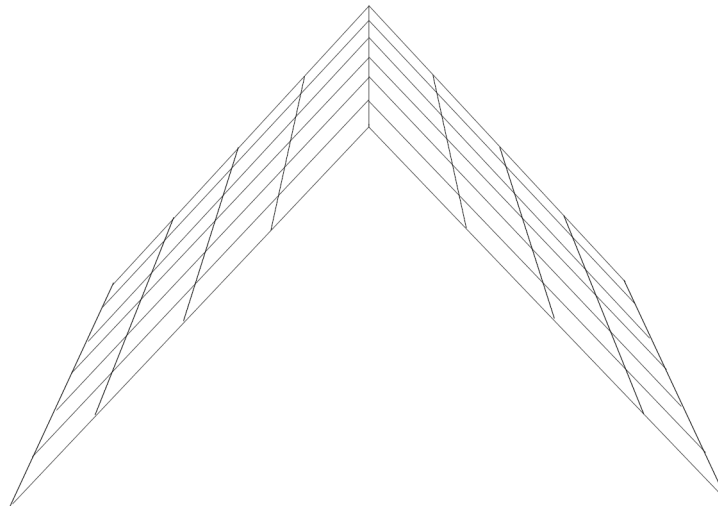


Figure 12-2. Slope Difference between Adjacent Elements Connected to the Same Line

Note

That these two situations are geometric rather than topological in nature and, therefore, are only considered under the geometric interpolation option.

Since exception points are defined as grid points at which stress discontinuities may exist, it is conceivable that a computed average grid point stress at an exception point may be different for each of the directly connected elements. However, it is more common to find that a subset of the elements that are directly connected to an exception point forms a subsurface on which stresses may be considered to be continuous. A unique average grid point stress may then be computed for the elements of the subsurface.

The calculation of average grid point stresses for a subsurface that contains two or more elements is performed with the geometric interpolation method used for edge points. If a subsurface consists of a single element, the calculation of an average grid point stress is performed in the same manner as

utilized in the calculation of average grid point stresses at corner points. It must be noted, however, that σ_D , σ_{E1} , and σ_{E2} may be interior, exterior, or corner points (see Table 12-4).

User Information

The use of the grid point stress recovery option is demonstrated in this section through the consideration of several example problems. These problems are discussed separately after the presentation of the requirements that must be met in the Executive Control Section, the Case Control Section, and the Bulk Data Section.

Executive Control Section

A typical Executive Control Section would be of the following form:

```
SOL 101
CEND
```

Case Control Section

The option to obtain grid point stress output is exercised through commands in the Case Control Section. The general request for the recovery of grid point stresses is accomplished using commands with the form:

```
GPSTRESS = ALL
```

or

```
SET j = k, l, m, . . . GPSTRESS = j
```

The k, l, m, . . . reference the identification numbers of the surfaces or volumes of elements that are defined in the OUTPUT(POST) section of the Case Control Section. Grid point stress data will be output only for those surfaces or volumes referenced through the GPSTRESS Case Control command.

Note

If grid point stress data is desired, an ELSTRESS (or STRESS) request for all elements in the surfaces or volumes of interest must be included in the Case Control Section. Such requests will cause the output of element stress data as well as grid point stress data. If the output of element stress data is not desired, the requests ELSTRESS(PLOT) = or STRESS(PLOT) = may be used.

In static analysis, one may place the GPSTRESS command above the subcase level or in individual subcases. If the GPSTRESS command is above the subcase level, it will be utilized by all subcases and subcoms unless overridden by a GPSTRESS command in a SUBCASE or SUBCOM.

The OUTPUT(POST) section of the Case Control Section contains specific requests for grid point stress data for the surfaces or volumes of elements. Any number of element surfaces or volumes may be defined, but only those surfaces or volumes that are referenced through the GPSTRESS command will have grid point stress data generated and output.

Element surfaces and volumes are defined through a SURFACE and VOLUME commands in the OUTPUT(POST) section of the Case Control Section.

Grid Point Stress Output Description

Listing 12-1 through **Listing 12-3** are examples of grid point stress output produced for two-dimensional plate elements requested on the SURFACE Case Control command. **Listing 12-1** is the most common form of output and is always the result when the TOPOLOGICAL method is used, and when the GEOMETRIC method is used and no surface “BRANCHing” is detected. Note that when the “ELEMENT ID” value is zero, it always indicates that all elements in the SURFACE connected to the grid point contribute to the average grid point stress and lie in the “best” subsurface. **Listing 12-2** and **Listing 12-3** show examples of the output when the GEOMETRIC method is used and when “BRANCHing” occurs. The negative value for the “ELEMENT ID” indicates the “best” average grid point stress and the element identification number of an element associated with “best” subsurface. Similarly a positive element identification numbers indicate another average grid point stress for a “less best” grid point stress and the element identification number associated with it's subsurface. When TOLERANCE is set positive as in **Listing 12-2** only the “best” grid point stress is calculated when “BRANCHing” is detected and when TOLERANCE is set negative all subsurface average grid point stress are calculated. See “SURFACE” in the *NX Nastran Quick Reference Guide*.

```

SQUARE PLATE WITH CIRCULAR HOLE
GRID POINT STRESS TEST CASE
UNIFORM LOAD ALONG X=5.0.
SUBCASE = 1
1
STRESSES AT GRID POINTS - - SURFACE 4
SURFACE X-AXIS X NORMAL(Z-AXIS) Z REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 1
STRESSES IN SURFACE SYSTEM PRINCIPAL STRESSES MAX
GRID ELEMENT
ID ID FIBER NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR SHEAR VON MISES
10304 0 MID 4.965E+03 4.041E+03 -6.431E+03 -42.9450 1.095E+04 -1.944E+03 6.447E+03 1.204E+04
10305 0 MID 6.132E+03 3.839E+03 -5.518E+03 -39.1310 1.062E+04 -6.503E+02 5.636E+03 1.096E+04
10306 0 MID 7.033E+03 3.408E+03 -4.246E+03 -33.4429 9.837E+03 6.037E+02 4.617E+03 9.549E+03
10400 0 MID 6.101E+02 1.631E+04 -1.038E+03 -86.2353 1.638E+04 5.418E+02 7.919E+03 1.612E+04
10401 0 MID 1.463E+03 1.417E+04 -3.402E+03 -75.9185 1.503E+04 6.098E+02 7.209E+03 1.473E+04
10402 0 MID 2.228E+03 1.211E+04 -5.354E+03 -66.3558 1.446E+04 -1.156E+02 7.286E+03 1.452E+04
10403 0 MID 2.809E+03 1.019E+04 -6.381E+03 -60.0211 1.387E+04 -9.721E+02 7.371E+03 1.433E+04
10404 0 MID 3.159E+03 8.514E+03 -6.308E+03 -56.5007 1.269E+04 -1.016E+03 6.852E+03 1.323E+04
10405 0 MID 3.307E+03 7.318E+03 -5.482E+03 -55.0445 1.115E+04 -5.249E+02 5.837E+03 1.142E+04

```

Listing 12-1. Grid Point Stress Output – Typical

```

INTERSECTING PLATES
GRID POINT STRESS TEST CASE
SUBCASE = 1
1
STRESSES AT GRID POINTS - - SURFACE 4
SURFACE X-AXIS X NORMAL(Z-AXIS) Z REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0
STRESSES IN SURFACE SYSTEM PRINCIPAL STRESSES MAX
GRID ELEMENT
ID ID FIBER NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR SHEAR VON MISES
10304 0 Z1 9.107E+02 1.209E+03 2.799E+03 46.5224 3.863E+03 -1.743E+03 2.803E+03 4.969E+03
Z2 1.052E+03 1.571E+03 2.913E+03 47.5464 4.236E+03 -1.613E+03 2.925E+03 5.233E+03
MID 9.813E+02 1.390E+03 2.856E+03 47.0453 4.049E+03 -1.678E+03 2.863E+03 5.099E+03
10305 0 Z1 1.906E+03 5.699E+02 1.789E+03 34.7594 3.147E+03 -6.714E+02 1.909E+03 3.531E+03
Z2 2.032E+03 5.648E+02 1.934E+03 34.6114 3.367E+03 -7.698E+02 2.069E+03 3.811E+03
MID 1.969E+03 5.674E+02 1.861E+03 34.6824 3.257E+03 -7.206E+02 1.989E+03 3.671E+03
10306 0 Z1 2.771E+03 -1.983E+02 2.678E+02 5.1132 2.795E+03 -2.223E+02 1.508E+03 2.912E+03
Z2 3.024E+03 -1.118E+02 4.889E+02 8.6589 3.098E+03 -1.862E+02 1.642E+03 3.196E+03
MID 2.897E+03 -1.550E+02 3.783E+02 6.9615 2.944E+03 -2.012E+02 1.572E+03 3.049E+03
10400 -119 Z1 -5.775E+04 1.181E+03 -2.060E+04 -72.5215 7.667E+03 -6.423E+04 3.595E+04 6.839E+04
Z2 -5.821E+04 1.207E+04 -1.647E+04 -77.4454 1.573E+04 -6.188E+04 3.880E+04 7.106E+04
MID -5.798E+04 6.623E+03 -1.853E+04 -75.0777 1.156E+04 -6.292E+04 3.724E+04 6.942E+04
10401 -119 Z1 -1.856E+04 7.608E+03 -4.939E+03 -79.4558 7.979E+03 -1.948E+04 1.373E+04 2.446E+04
Z2 -1.714E+04 1.656E+04 -3.881E+03 -83.5131 1.700E+04 -1.758E+04 1.729E+04 2.995E+04
MID -1.785E+04 1.181E+04 -4.410E+03 -81.7177 1.245E+04 -1.849E+04 1.547E+04 2.696E+04

```

Listing 12-2. Grid Point Stress Output with “BRANCHing” and TOLERANCE > 0

INTERSECTING PLATES									
GRID POINT STRESS TEST CASE									
SUBCASE = 1									
STRESSES AT GRID POINTS - - SURFACE 5									
GRID	ELEMENT	SURFACE X-AXIS X NORMAL (Z-AXIS) Z			REFERENCE COORDINATE SYSTEM FOR SURFACE			DEFINITION	CID
ID	ID	FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	SHEAR
STRESSES IN SURFACE SYSTEM									
PRINCIPAL STRESSES									
MAX									
VON MISES									
10304	0	Z1	9.107E+02	1.209E+03	2.799E+03	46.5224	3.863E+03	-1.743E+03	2.803E+03
		Z2	1.052E+03	1.571E+03	2.913E+03	47.5464	4.236E+03	-1.613E+03	2.923E+03
		MID	9.813E+02	1.390E+03	2.856E+03	47.0453	4.049E+03	-1.678E+03	2.863E+03
10305	0	Z1	1.906E+03	5.699E+02	1.789E+03	34.7594	3.147E+03	-6.714E+02	1.909E+03
		Z2	2.032E+03	5.648E+02	1.934E+03	34.6114	3.367E+03	-7.698E+02	2.068E+03
		MID	1.969E+03	5.674E+02	1.861E+03	34.6824	3.257E+03	-7.206E+02	1.989E+03
10306	0	Z1	2.771E+03	-1.983E+02	2.678E+02	5.1132	2.795E+03	-2.223E+02	1.508E+03
		Z2	3.024E+03	-1.118E+02	4.889E+02	8.6589	3.098E+03	-1.862E+02	1.642E+03
		MID	2.897E+03	-1.550E+02	3.783E+02	6.9615	2.944E+03	-2.012E+02	1.572E+03
10400	-119	Z1	-5.775E+04	1.181E+03	-2.060E+04	-72.5215	7.667E+03	-6.423E+04	3.595E+04
		Z2	-5.821E+04	1.207E+04	-1.647E+04	-77.4454	1.573E+04	-6.188E+04	3.880E+04
		MID	-5.798E+04	6.623E+03	-1.853E+04	-75.0777	1.156E+04	-6.292E+04	3.724E+04
	219	Z1	-5.772E+04	1.152E+03	2.064E+04	72.4819	7.667E+03	-6.423E+04	3.595E+04
		Z2	-5.819E+04	1.204E+04	1.652E+04	77.4058	1.573E+04	-6.188E+04	3.880E+04
		MID	-5.795E+04	6.597E+03	1.858E+04	75.0381	1.156E+04	-6.292E+04	3.724E+04
10401	-119	Z1	-1.856E+04	7.060E+03	-4.939E+03	-79.4558	7.979E+03	-1.948E+04	1.373E+04
		Z2	-1.714E+04	1.656E+04	-3.881E+03	-83.5131	1.700E+04	-1.758E+04	1.729E+04
		MID	-1.785E+04	1.181E+04	-4.410E+03	-81.7177	1.245E+04	-1.849E+04	1.547E+04
	120	Z1	-1.856E+04	7.060E+03	-4.939E+03	-79.4558	7.979E+03	-1.948E+04	1.373E+04
		Z2	-1.714E+04	1.656E+04	-3.881E+03	-83.5131	1.700E+04	-1.758E+04	1.729E+04
		MID	-1.785E+04	1.181E+04	-4.410E+03	-81.7177	1.245E+04	-1.849E+04	1.547E+04
	219	Z1	-1.855E+04	7.053E+03	4.957E+03	79.4163	7.979E+03	-1.948E+04	1.373E+04
		Z2	-1.713E+04	1.655E+04	3.905E+03	83.4735	1.700E+04	-1.758E+04	1.729E+04
		MID	-1.784E+04	1.180E+04	4.431E+03	81.6781	1.245E+04	-1.849E+04	1.547E+04
	220	Z1	-1.855E+04	7.053E+03	4.957E+03	79.4163	7.979E+03	-1.948E+04	1.373E+04
		Z2	-1.713E+04	1.655E+04	3.905E+03	83.4735	1.700E+04	-1.758E+04	1.729E+04
		MID	-1.784E+04	1.180E+04	4.431E+03	81.6781	1.245E+04	-1.849E+04	1.547E+04

Listing 12-3. Grid Point Stress Output with “BRANCHing” and TOLERANCE < 0

General Remarks

Each element stress component (independently of all other stress components) is transformed into the output coordinate system before interpolation. It is necessary that this output coordinate system vary slowly over the surface for the interpolated stresses to be accurate. This approach is certainly valid for stress interpolation over elements that are in a common plane and it should be adequate for engineering analysis when the difference in slope between adjacent elements is small. However, when the elements do vary substantially from a smooth surface, the analyst must utilize engineering judgment relative to the accuracy of the resultant stress data.

Average grid point stresses are only computed at element vertices. For elements with defined midpoints, you can estimate the stress at these midpoints to be the average of the computed stresses at the grid point at the ends of the edge in question.

12.3 Mesh Stress Discontinuities at Grid Points

Error Estimates for Grid Point Stress Data

This section describes the error estimation tools that can help you identify areas of your model that may require refinement.

The essence of the previous averaging procedures used to obtain grid point stress data is to:

1. Convert these local element stress components into a common user-defined coordinate system.
2. Average the several values of each stress component to obtain a unique value of the stress component that is to be associated with the grid point in question.
3. Compute the stress invariants at the grid points from the stress components at the grid points.

In the general case, the stress components are σ_x , σ_y , σ_z , T_{xy} , T_{xz} , and T_{yz} .

For discussion purposes, the averaging process used to compute the stress components at the grid points can be represented in the form of Eq. 12-4.

$$\sigma_g = \sum_{i=1}^{N_e} (W_i \sigma_{ei})$$

Equation 12-9.

where:

- σ_g = weighted mean value of the stress component computed at the grid point.
- σ_{ei} = value of the stress component in the i th element ($i = 1, 2, \dots, N_e$) in the neighborhood of the grid point. δ_{gi} is in the same coordinate system as δ_g .
- W_i = weighting factor assigned to the i th element. The sum of the N_e values of W_i must equal 1. (This requirement assures that all computed statistics will be unbiased.) The attribute of being unbiased implies that the variance is equal to the mean square error. Equal weighting (i.e., $W_i = 1/N_e$) is assumed.

An estimate of the error in a particular component of stress at a grid point can then be computed by assuming that the values of the corresponding stress components computed for the elements in the neighborhood of the grid point are data points with uncorrelated random errors. It then follows that an estimate of the probable error in the stress component δ_g at the grid point is:

$$\begin{aligned} \delta_g &= \sqrt{\sum_{i=1}^{N_e} (W_i \delta_{ei})^2} \\ &= \frac{1}{\sqrt{N_e}} \sqrt{\sum_{i=1}^{N_e} (\delta_{ei})^2} \end{aligned}$$

Equation 12-10.

where $\delta_{ei} = \sigma_{ei} - \sigma_g$. Thus, the probable error δ_g is the root mean square error in δ_{ei} divided by $\sqrt{N_e}$.

It should be noted that the standard deviation or root mean square error is a reasonable measure of precision in many practical cases, but it is easy to provide examples in which the standard deviation is a poor measure of the concentration of the distribution about the mean. Eq. 12-9 is assumed to provide an approximate error estimator for the grid point stress data.

Element and Grid Point Stress Discontinuity Output Requests

It should be clear that element and grid point stress discontinuity output can only be obtained if the analyst has requested grid point stress output via the GPSTRESS command in the Case Control Section of the NX Nastran input file. The GPSTRESS command, of course, requires that the analyst define all required SURFACES and VOLUMES in the OUTPUT(POST) portion of the Case Control Section.

To output Grid Point Stress DisCONTinuities:

```
GPSDCON = ALL
```

or

```
SET j = k, l, m, ....GPSDCON = j
```

To output ELement Stress DisCONTinuities:

```
ELSDCON = ALL
```

or

```
SET j = k, l, m, ....ELSDCON = j
```

The following remarks should be noted relative to the use of the GPSDCON and/or ELSDCON commands.

1. The GPSDCON and ELSDCON commands are honored only in SOLution Sequences 101, 114, 144, and 200.
2. The GPSDCON and ELSDCON commands may be placed above the SUBCASE level or in individual SUBCASEs and/or SUBCOMs. The use of these commands above the SUBCASE level causes stress discontinuity data to be output for ALL SUBCASEs in the NX Nastran input file. If the analyst wishes to restrict stress discontinuity output to specific SUBCASEs, GPSDCON and/or ELSDCON commands should appear only under these specific SUBCASEs.
3. The analyst is cautioned that these commands produce a substantial amount of output.
4. The GPSDCON and/or ELSDCON commands may be effectively used in restarts. For example, one could include requests for GPSTRESS data during the initial run of a project and then obtain data on stress discontinuities on a subsequent restart by including GPSDCON and/or ELSDCON commands in the Case Control Section.
5. Stress discontinuity output will only be provided when grid point stresses have been previously computed by the methods described in this article. Thus, stress continuity will not be computed when, for example, both plate and solid elements are connected to a grid point that is involved in stress discontinuity calculations.

Discussion of Error Measures

Large values of error estimators for CQUAD4 and CTRIA3 elements aren't uncommon occurrences in statistical error measures. For example, the mean and variance have direct analogs in engineering mechanics. The mean is analogous to the centroid of a body and is thus a measure of where the mass is centered. The variance is the second moment about the mean, and it tends to be small if the majority of the mass is concentrated about the centroid. As the mass is dispersed further from the

centroid, the moment of inertia tends to increase. If the mass is concentrated at the centroid (as it might be in some idealized lumped mass models), the moment of inertia becomes zero.

Thus, the error estimates for CQUAD4 and CTRIA3 elements will tend toward larger values because the data used to compute the estimators are dispersed relatively far from the mean. For those elements that provide stress data at vertices, the estimators will tend toward smaller values because the stress data tend to be concentrated about the mean in well-designed finite element models.

Generally, the goal is to design a finite element mesh for static analysis so that all important stress gradients will be adequately represented. If the mesh is not sufficiently detailed, the stress data at element vertices or element centroids will result in the relatively large values of the error estimators.

Very inaccurate values of these error estimators may be obtained at the edges of defined SURFACES and on the faces of defined VOLUMES. Such edges and surfaces will often correspond with the boundaries of superelements.

In summary, error estimators can, in some cases, be highly inaccurate. Nevertheless, error estimate data is quite useful when properly interpreted by the analyst.

Grid Point Stresses and Mesh Stress Discontinuities

Grid point stresses may be requested for grid points connected by CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR, CHEXA, CPENTA, CPYRAM and CTETRA elements (with linear material properties only). In linear static analysis only, a mesh stress discontinuity error analysis may then be performed based on the computed element and grid point stresses.

Grid point stresses are not computed for laminar stresses.

The grid point stresses are calculated in the user-specified coordinate system. The following real membrane stresses are output on request:

- Normal stresses in the x and y directions.
- Shear stresses on the x face in the y direction.
- Angle between the x axis and the major principal axis.
- Major and minor principal stresses.
- von Mises equivalent or maximum shear stress (see STRESS Case Control command).

The following real stresses are output on request for grid points connected to solid elements:

- The three normal stresses.
- The three shear stresses.
- Magnitude and direction of the three principal stresses.
- The mean pressure.
- The von Mises equivalent or octahedral stress (see STRESS Case Control command).

Only real stresses are available at the grid points. The user has the following options:

- The capability to vary output requests by SUBCASE.

- Choice of elements in stress field.
- Choice of topological or geometric interpolation (surface fields only).
- Choice of output coordinate system.
- Choice of output stress fiber locations (surface fields only).

The grid point stresses are requested in the Case Control Section for printing and postprocessing using the Case Control commands GPSTRESS (Grid Point Stress) and STRFIELD (Stress Field). The user is required to request element stress output (STRESS = N) for all elements referenced by selected SURFACE and VOLUME commands. The grid point stress requests a set of surface or volume fields which are defined in the OUTPUT(POST) part of the Case Control Section on the SURFACE or VOLUME commands, respectively. The postprocessing output requests are separated from case control and plot requests by the OUTPUT(POST) command.

The general rules for Case Control syntax are the same as for plotting. The grid point stress output is defined on the SURFACE and VOLUME commands. The SURFACE command defines the following: output coordinate system, list of elements in the stress surface field, output stress fiber location for stress evaluation and the choice of topological or geometric interpolation method used to determine the grid point stresses from the elements. The VOLUME command defines the following: output coordinate system, list of elements in the stress volume, and the choice of DIRECT and/or PRINCIPAL stresses.

In the linear static structured solution sequences only (SOLutions 101, 114, 144, and 200) mesh stress discontinuities may be requested to estimate errors due to the coarseness of the finite element mesh. The estimation of the error is computed for the element and grid point stresses and requested by the ELSDCON and GPSPDCON Case Control commands, respectively. These commands select the desired surfaces and volumes which are defined in the OUTPUT(POST) section of the case control. This output option also requires a corresponding GPSTRESS or STRFIELD request for the desired surfaces and volumes and a STRESS case control request for the elements in those surfaces and volumes.

The following is computed and output for stress discontinuities on shell elements:

- Normal stress discontinuities in the x and y directions.
- Shear stress discontinuities on the xy direction.
- Major and minor principal stress discontinuities.
- von Mises and maximum shear stress discontinuities.
- Error estimate.

The following is computed and output for stress discontinuities on solid elements:

- Normal stress discontinuities in the x, y, and z directions.
- Shear stress discontinuities on the xy, yz, and xz directions.
- Major and minor principal stress discontinuities.
- von Mises stress and mean pressure discontinuities.

- Error estimate.

Chapter 13: Solution Sequences

- *Understanding Solution Sequences*
- *Data Processing and Matrix Operations by Functional Module*
- *Summary of Solution Sequence Operations*

13.1 Understanding Solution Sequences

NX Nastran is comprised of a large number of building blocks called modules. A module is a collection of FORTRAN subroutines designed to perform a specific task, such as processing model geometry, assembling matrices, applying constraints, solving matrices, calculating output quantities, conversing with the database, printing the solution, and so on. The modules are controlled by an internal language called the Direct Matrix Abstraction Program (DMAP).

Each type of analysis available in NX Nastran is called a solution sequence. Each solution sequence is a prepackaged collection of hundreds or thousands of DMAP commands that you select with a SOL Executive Control statement. Once you select a solution sequence, the software automatically sends that sequence's particular set of DMAP commands to the modules that are needed to perform the requested solution.

The major type is called a Solution Sequence. They are called with numbers ranging from 100 to 701. The solution sequences are listed in [Table 13-1](#).

Note: You can also select a solution sequence by using the SOL keyword followed by the name of the solution sequence. For example, SOL SESTATIC is equivalent to SOL 101. Using the solution numbers is generally recommended rather than their names because the solution numbers are more easily recognized.

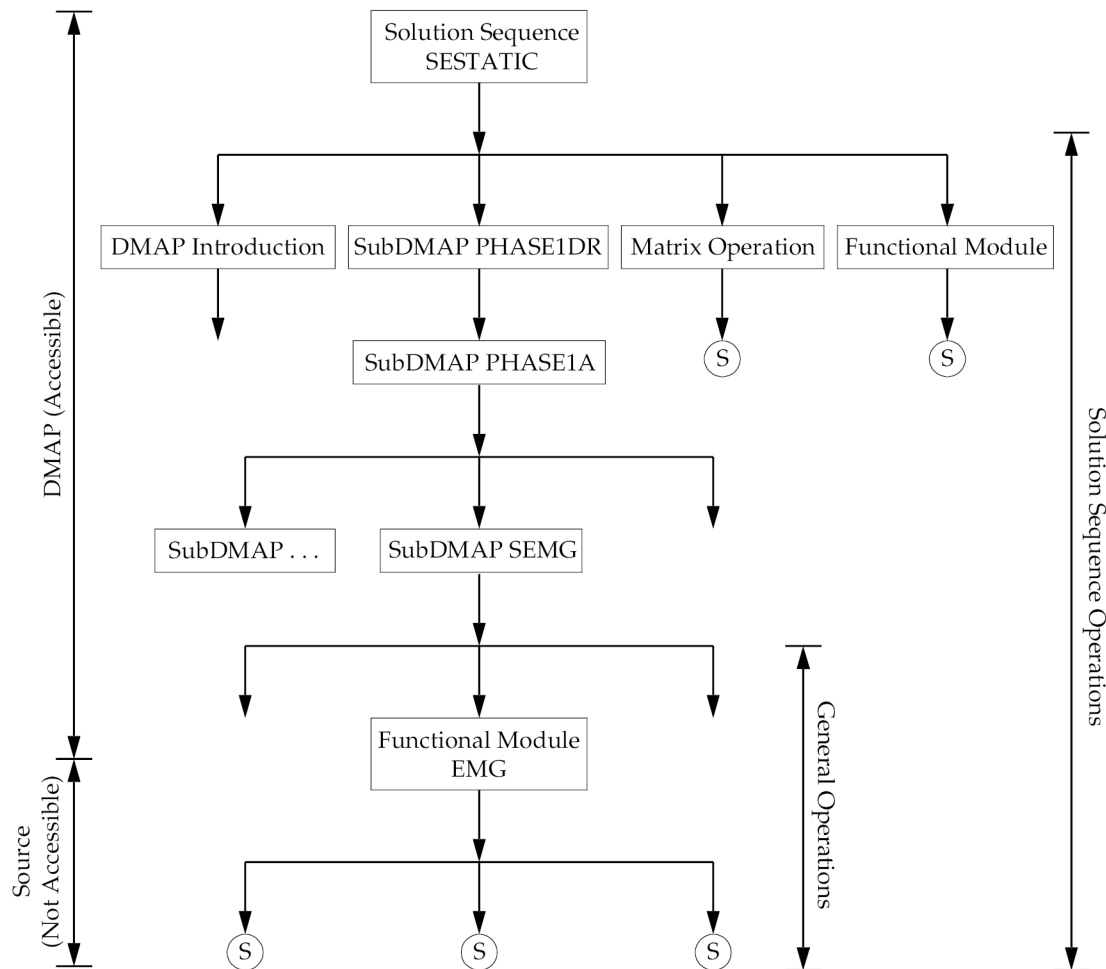


Figure 13-1. Hierarchy in the Structural Solution Sequences

Table 13-1. Structured Solution Sequences		
SOL Number	SOL Name	Description
101	SESTATIC	Statics with options: Linear Heat Transfer, Alternate Reduction, Inertia Relief
103	SEMODES	Normal Modes
105	SEBUCKL	Buckling
106	NLSTATIC	Nonlinear Statics See the <i>NX Nastran Basic Nonlinear Analysis User's Guide</i>
107	SEDCEIG	Direct Complex Eigenvalues
108	SEDFREQ	Direct Frequency Response
109	SEDTRAN	Direct Transient Response
110	SEMCEIG	Modal Complex Eigenvalues

Table 13-1. Structured Solution Sequences

SOL Number	SOL Name	Description
111	SEMFREQ	Modal Frequency Response
112	SEMPTRAN	Modal Transient Response
114	CYCSTATX	Cyclic Statics with option: Alternate Reduction
115	CYCMODE	Cyclic Normal Modes
116	CYCBUCKL	Cyclic Buckling
118	CYCFREQ	Cyclic Direct Frequency Response
129	NLTRAN	Nonlinear Transient Response See the <i>NX Nastran Basic Nonlinear Analysis User's Guide</i>
144	AESTAT	Static Aeroelastic Response See the <i>NX Nastran Aeroelastic Analysis User's Guide</i>
145	SEFLUTTR	Aerodynamic Flutter See the <i>NX Nastran Aeroelastic Analysis User's Guide</i>
146	SEAERO	Aeroelastic Response See the <i>NX Nastran Aeroelastic Analysis User's Guide</i>
153	NLSCSH	Static Structural and/or Steady State Heat Transfer Analysis
159	NLTCSH	Transient Structural and/or Transient Heat Transfer Analysis
187	DDAM	DDAM analysis.
190	DBTRANS	Database Transfer
200	DESOPT	Design Optimization with additional option for only Sensitivity Analysis See the <i>NX Nastran Design Sensitivity and Optimization User's Guide</i>
401	NLSTEP	Multistep, structural solution which supports a combination of static (linear or nonlinear) subcases and modal (real eigenvalue) subcases.
601,N		Advanced nonlinear implicit
701		Advanced nonlinear explicit

Obtaining DMAP Listings for Solution Sequences

You can obtain the DMAP listings for any of the solutions sequences by including a COMPILER LIST statement in the Executive Control Section.

- A description of the data processing and matrix operations for the solution sequences is given in [“Summary of Solution Sequence Operations.”](#)
- The complete details for each of the functional modules are given in Chapter 4 of the *NX Nastran Programmer's Manual*.

- Those DMAP modules which are of general use (matrix operations and utility modules) are described along with examples of their use in “[Data Processing and Matrix Operations by Functional Module](#).”

Using Alters to Modify Solution Sequences

If you want to modify a DMAP sequence, you can use the alter feature described in *NX Nastran DMAP Programmer's Guide*. For example, you can use an alter to:

- Schedule an EXIT prior to completion in order to check intermediate output
- Schedule the printing of a table or matrix for diagnostic purposes
- Delete or add a functional module to the DMAP sequence

To make alterations to a solution sequence, you should be familiar with the rules for DMAP programming

A library of DMAP alters is delivered with NX Nastran. You can use these alters to request options not available in the solution sequences. You can also use them to perform exceptional operations that do not fit in with the normal flow of the solution sequences.

Understanding Scheduled and Unscheduled Exits

Scheduled exits can be requested at any point in a structured solution sequence format by means of the SEMG, SEKR, SELG, SELR, SEMR, and SEALL Case Control commands. If none of these commands are specified, then SEALL = ALL is the default.

Unscheduled exits are usually caused by errors on input entries or errors in the structural model resulting from missing or inconsistent input data. When such errors are detected, an unscheduled exit is performed accompanied with the output of the applicable user error messages. Following the correction of the input data errors, a modified restart can be performed.

Unscheduled exits may also occur because of machine failure or insufficient time allowance. In these cases, an unmodified restart is usually made. In some cases, where a portion of the problem has been completed, including the output for the completed portion, a modified restart must be made following an unscheduled exit due to insufficient time allowance. The situations are discussed under case control requirements in the sections dealing with the individual solution sequences.

The initial execution of any problem must be made with complete NX Nastran input data, including all of the bulk data. However, all or part of the bulk data may be assembled from alternate input sources.

Restart Procedures

NX Nastran also has an automatic restart capability. The automatic restart is not only driven by modifications to the Case Control and Bulk Data Sections but also by modifications in upstream superelements.

In NX Nastran, restarts are accomplished with the NASTRAN Data Definition Language feature, a file that lists the names and attributes of data types saved for restart. The evidence that a Solution Sequence has restart capability can be found by the existence of TYPE statements in its DMAP compile listings. It uses the SubDMAP feature, which allow breaking a large collection of DMAP

statements into more manageable data structures similar to subroutines in other programming languages.

For a restart analysis, the Bulk Data Section consists only of delete “/” entries and any new entries you want to add. The previous Bulk Data is read from the database. All other parts of the NX Nastran Input Data (including the File Management statements, the Executive Control statements, the Case Control commands, the BEGIN BULK command and the ENDDATA entry) must be resubmitted even though no changes are made in the case control and no new bulk data is added. When changing solution sequences, the solution number (SOL) must be changed to the number or name of the new solution sequence.

Any changes in the Case Control Section associated with Bulk Data entry selection or subcase definition, or changes in the Bulk Data Section.

Restarts always start at the beginning of a specified DMAP or SOL sequence. In a restart analysis, NX Nastran queries the database in two phases.

Phase 1: The software marks all appropriate existing database data blocks and parameters as existing for the current restart run.

Phase 2: The software checks the current input against the version from which the restart initiates and deletes any data blocks with modified input from the Phase 1 determined database data blocks and parameters. This phase is performed by the RESTART module(s) contained within the solution's DMAP.

Restart only executes DMAP Modules for which some or all of the output does not exist on the current version database. The S-type parameter is considered as an output data block for restart purposes, hence if it is not listed in the NDDL, then the module that contains it will be re-executed. All DMAP modules are executed until the first RESTART module after which output checking is performed. Forced execution of modules after this point may be manually controlled by SYSTEM (109) flag.

If no such changes are made, the executive system performs an unmodified restart. If changes have been made only in the output requests, the restart is considered unmodified. However, some modules may have to be re-executed in order to prepare the output. Output requests that were satisfied, plots made, etc., should be removed if they are not to be output again.

For modified restarts, a number of previously executed DMAP instructions may have to be re-executed, depending on the nature of the modifications made by the user. The DMAP instructions that need to be executed in a modified restart are automatically determined within the program by comparing all changes made in Case Control commands and Bulk Data entries.

See also

- “**Restarts**” in the *NX Nastran User's Guide*

13.2 Data Processing and Matrix Operations by Functional Module

The following sections will refer specifically to the structural analysis matrices such as stiffness, damping, and mass. However, this information also applies to heat transfer analysis for conductance and capacitance. There is no analogy for mass in heat transfer analysis.

See also

- *NX Nastran DMAP Programmer's Guide* (for additional information regarding the use of DMAP modules for matrix operations)
- Chapter 4 of the *NX Nastran Programmer's Manual* (for detailed descriptions of the individual functional modules)

Geometry Processing in SubDMAP PHASEO

NX Nastran begins the initial processing of your total model in PHASEO. In PHASEO, the software takes data from the input files and stores it in data structures that feed more expensive processes in later SubDMAPS. Flags are set to control branching for special features. All superelement inputs are processed. Tables are indexed for faster processing in post-processors. Parameters are read from Case Control and Bulk Data files by module PVT. The shell element normals at each grid point are computed in module TASN2, and output in table GPSNT.

Tables for internal program use are generated by the geometry processors from the grid point entries, coordinate system definition entries, sequence entries, connection entries, static load entries, temperature definition entries, and constraint entries. The computer time for all geometry processing is usually only a small part of the time required for problem solution.

The basic geometry processing is performed by GP1 (Geometry Processor – Phase 1). The software assembles a list of all grid points and scalar points in internal order for use in relating internal and external identification numbers. Matrices for transformation from basic to all defined global coordinate systems are computed, and all grid point locations are transformed to the basic coordinate system.

The element connection tables (ECT) are generated by GP2 (Geometry Processor – Phase 2) from the connection entries. The external grid point numbers are converted to the internal indices defined in GP1.

The temperature data and static loads are processed by GP3 (Geometry Processor – Phase 3). The temperature data is collected by sets and the external grid point numbers. Element numbers are converted to the internal indices defined in GP1 and GP2. In the case of static analysis, the static loads are also collected by sets and the external identification numbers are converted to internal indices.

The constraint data is processed by GP4 (Geometry Processor – Phase 4). The displacement set definition table (USET) is generated from the constraint definition entries. USET contains one coded word for each degree-of-freedom in the model to identify the set or sets of coordinates to which the degree-of-freedom belongs (see “[Understanding Sets and Matrix Operations](#)” for the definition of degree-of-freedom sets). The multipoint constraint matrix $[R_{mg}]$ is formed from the MPC and rigid element Bulk Data entries. In static analysis, the enforced displacement matrix $[Y_s]$ is formed for use in the generation of equivalent static loads.

The BNDSPC module moves SPCs on boundary grid points to downstream superelements.

Model Checkout

A checkout procedure is provided for the development of new models. It includes the bookkeeping and control blocks, and part of the generation block of [Figure 13-3](#). It is requested by the PARAM,CHECKOUT,YES option.

See also

- “**Superelement Analysis**” in the *NX Nastran User's Guide*

SEP1 Module

Generates the SEMAP table, which lists superelement membership of grid points and elements. Provides estimates for computation costs and storage requirements. This module will identify any structural partitioning errors.

There are two paths through the superelement capability. If the bulk data entries have a BEGIN SUPER and/or BEGIN SUPER = [SEID] entries the data flow goes through the partitioned superelement capability modules, SEP1X and SEP2X. If only a BEGIN BULK entry is present the grid point list capability modules SEP1 and SEP2 are used. The SEP1X path also allows grid point list superelements, with the exception of fluid structure analysis, SOL 200 (Design Optimization), and the RELEASE entry. Models using these features must use the grid point list path.

See also

- “**Superelement Analysis**” in the *NX Nastran User's Guide*

SEP2, SEP2CT Modules

Partitions out the data for any superelement selected by SE-type Case Control commands. These modules identify load and rigid element partitioning errors, and errors in the Case Control Section.

Constraint Modules

The GP1 and GP4 modules process all selected superelements. They check for the presence of requested coordinate systems; for illegal duplicate specifications of mutually exclusive sets, such as SPCi and ASETi entries referencing the same point; and for correct specifications on SPCi, xSETi, and MPCi entries, and rigid elements.

You may print the user set table with PARAM,USETPRT, when PARAM,CHECKOUT,YES is present. If rigid elements or selected MPC entries exist, the constraint matrix $[R_{mg}]$ (see “**Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB**”) is given three tests:

1. A matrix of rigid body vectors $[u_{gh}^o]$ is calculated by the VECPLOT module. This matrix includes zeroes for the rows corresponding to scalar points. The product

$$[E_{mh}] \leftarrow [R_{mg}][u_{gh}^o]$$

is calculated. Terms of $[E_{mh}]$ larger than PARAM,TINY are printed. If the constraint matrix contains internal constraints, nonzero terms will exist in $[E_{mh}]$. If an MPC equation includes scalar points, nonzero terms may exist in the corresponding row of $[E_{mh}]$ although the equation may be in equilibrium.

2. The product

$$[R_{mm}^g] \leftarrow [R_{mg}][R_{mg}]^T$$

is calculated. The product matrix is decomposed by the DCMF module. Diagonal terms of the factor matrix that are larger than PARAM,MAXRATIO are printed. These terms indicate that the printed row is linearly dependent on prior rows. Unless the constraint matrix is changed, it is likely that the model will encounter fatal errors due to singularity in the MCE1 module, or will have a poorly-conditioned constraint elimination process.

3. The product

$$[R_{mm}^m] \leftarrow [R_{mm}][R_{mm}]^T$$

is calculated, where $[R_{mm}]$ is the partition of $[R_{mg}]$. This product is also processed by the DCMF module. If a row appears here that also appears in the messages from the prior step, it confirms that the constraint matrix is inherently ill-conditioned. If a row appears here that did not appear in the prior step, it indicates that a change in the m-set exists that can also make the $[R_{mm}]$ matrix well conditioned.

The DBC module stores data for use in postprocessors. It runs at many points in the Solution Sequences. In this SubDMAP it stores basic structural geometric data. The AEROE SubDMAP generates the geometry of the aerodynamic model. The RESTART module controls the modules that will be run on restart, based on changes in the input files. The run exits after these checks are performed.

Matrix Assembly Operations in SubDMAP SEMG

The stiffness, mass, damping, differential stiffness, and heat conduction matrices are formed during the matrix assembly operations in subDMAP SEMG.

Stiffness is generated from the following sources:

- Structural elements defined on connection entries (e.g., CBAR and CROD).
- General elements defined on GENEL entries.
- Scalar springs defined on CELASi entries.

Mass is generated from the following sources:

- A 6×6 matrix of mass coefficients at a grid point defined on a CONM1 entry.
- A concentrated mass element defined on a CONM2 entry in terms of its mass and moments of inertia about its center of gravity.
- Structural mass for all elements, except plate elements without membrane stiffness, using the mass density on the material definition entry.
- Nonstructural mass for all elements specifying a value on the property entry.
- Scalar masses defined on CMASSi entries.

Damping is generated from the following sources:

- Viscous rod elements defined on CVISC entries.

- Scalar viscous dampers defined on CDAMPi entries.
- Element structural damping by multiplying the stiffness matrix of an individual structural element by a damping factor obtained from the material properties (MATi) entry for the element.

These matrices are formed from the connection and geometric tables prepared by the geometry processors and the material and element property tables prepared in the Phase 0. The matrix assembly operations will usually represent a significant portion of the total solution time. The computer time for matrix assembly operations is linear with the number and type of elements. Unit times for stiffness matrix assembly are given in the *NX Nastran Installation and Operations Guide*. Assembly times for coupled mass matrices, differential stiffness matrices, and heat conduction matrices are about the same as those for stiffness matrices. Assembly times for lumped mass matrices and damping matrices are usually relatively small.

The matrix assembly process begins with the generation of various tables in TA1 (Table Assembler) from the geometric data, connection data and property data. These tables are used in data recovery operations as well as in the matrix assembly operations.

The matrix assembly operations are performed by EMG (Element Matrix Generator) and EMA (Element Matrix Assembler) modules. All of the element matrices (KELM, KDICT, MELM, MDICT, BELM, and BDICT) are generated by EMG. The stiffness matrix exclusive of general elements, $[K_{gg}^x]$, as well as the structural damping matrix, $[K_{gg}^4]$, the mass matrix, $[M_{gg}]$, and the viscous damping matrix, $[B_{gg}]$, are assembled in separate executions of EMA.

The parameter COUPMASS is used to select either coupled mass or lumped mass generation in EMG. If the parameter WTMASS is present, the mass matrices generated in EMG are multiplied term by term by the value in field 3 of the PARAM entry. This operation allows you to specify mass data in units of weight.

The structural damping matrix $[K_{gg}^4]$ is assembled from elements with structural damping. Structural damping is specified in the GE field of the MATi entry.

$$[K_{gg}^4] = \sum_{e=1}^{NE} GE_e [K_e]$$

Equation 13-1.

where:

- NE = number of elements
- GE_e = element structural damping coefficient from MATi entries
- $[K_e]$ = element stiffness matrix

The direct matrix inputs requested by the K2GG, K42GG, M2GG, and B2GG Case Control commands define the stiffness matrix $[K_{gg}^2]$, structural damping matrix $[K_{gg}^{4,2}]$, mass matrix $[M_{gg}^2]$, and damping matrix $[B_{gg}^2]$, and are assembled by the MTRXIN module. The outputs from EMA and MTRXIN are combined by the ADD module.

The stiffness contributions from the general elements (GENEL Bulk Data entry) are assembled in SMA3 (Structural Matrix Assembler – Phase 3). The final stiffness matrix $[K_{gg}]$ is formed by combining $[K^x_{gg}]$ prepared by the EMA module with the contributions from general elements generated by SMA3.

Weight and balance information is calculated from the mass matrix $[M_{gg}]$ by GPWG (Grid Point Weight Generator). The execution of GPWG is controlled by the parameter GRDPNT. The details of the output are described in “GRDPNT” in the *NX Nastran Quick Reference Guide*.

Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB

The multipoint constraint operations apply constraints to the stiffness matrices using the linear relationships among the displacements you specify on MPC, MPCAX, and rigid element Bulk Data entries.

The multipoint constraint equations are initially expressed in the form,

$$[R_g]\{u_g\} = 0 ,$$

Equation 13-2.

where you supply the coefficients. You also specify the degree-of-freedom that is made dependent by each equation of constraint, so that the $\{u_g\}$ matrix may immediately be partitioned into two subsets,

$$\{u_g\} = \begin{Bmatrix} u_n \\ u_m \end{Bmatrix} ,$$

Equation 13-3.

where the set, u_m where you supply the coefficients, is the set of dependent degrees-of-freedom. The matrix of constraint coefficients is similarly partitioned

$$[R_g] = [R_n \mid R_m] ,$$

Equation 13-4.

so that **Eq. 13-2** becomes

$$[R_n]\{u_n\} + [R_m]\{u_m\} = 0 .$$

Equation 13-5.

$[R_m]$ is a nonsingular matrix. We can, therefore, form the multipoint constraint matrix,

$$[G_m] = -[R_m]^{-1}[R_n] ,$$

Equation 13-6.

so that Eq. 13-4 may be stated as

$$\{u_m\} = [G_m]\{u_n\} .$$

Equation 13-7.

Prior to the imposition of constraints, the structural problem may be written as

$$[K_{gg}]\{u_g\} = \{P_g\} ,$$

Equation 13-8.

or, partitioning in terms of the coordinate sets, u_n and u_m

$$\begin{bmatrix} \bar{K}_{nn} & K_{nm} \\ K_{nm}^T & K_{mm} \end{bmatrix} \begin{Bmatrix} u_n \\ u_m \end{Bmatrix} = \begin{Bmatrix} \bar{P}_n \\ P_m \end{Bmatrix}$$

Equation 13-9.

Bars over symbols are used to designate arrays that are replaced in the reduction process.

The addition of constraints to the structure requires that the forces of constraint be added to the equilibrium equations. Thus, writing the equilibrium and constraint equations together in partitioned form,

$$\begin{bmatrix} \bar{K}_{nn} & K_{nm} & G_m^T \\ K_{nm}^T & K_{mm} & -I \\ G_m & -I & 0 \end{bmatrix} \begin{Bmatrix} u_n \\ u_m \\ q_m \end{Bmatrix} = \begin{Bmatrix} \bar{P}_n \\ P_m \\ 0 \end{Bmatrix} ,$$

Equation 13-10.

where $\{q_m\}$ is the vector of constraint forces on $\{u_m\}$. Straightforward elimination of u_m and q_m gives

$$[\bar{K}_{nn} + K_{nm}G_m + G_m^TK_{mm}^T + G_m^TK_{mm}G_m]\{u_n\} = \{\bar{P}_n\} + [G_m^T]\{P_m\} ,$$

Equation 13-11.

or

$$[K_{nn}]\{u_n\} = \{P_n\} ,$$

Equation 13-12.

where

$$K_{nn} = \bar{K}_{nn} + K_{nm}G_m + G_m^TK_{nm} + G_m^TK_{mm}G_m ,$$

Equation 13-13.

and

$$P_n = \bar{P}_n + G_m^TP_m .$$

Equation 13-14.

The initial partition of K_{gg} and the operations indicated by [Eq. 13-6](#), [Eq. 13-13](#) and [Eq. 13-14](#) are performed by appropriate modules of the program. The multipoint constraint matrix, G_m , is used in structural matrix reduction ([Eq. 13-13](#)), load vector reduction, ([Eq. 13-14](#)) and data recovery ([Eq. 13-7](#)).

The multipoint constraint equations, R_{mg} , formed in the GP4 module, are partitioned by the MCE1 module (Multipoint Constraint Eliminator – Phase 1) as follows:

$$[R_{mg}] = \begin{bmatrix} R_{mm} & R_{mn} \end{bmatrix}$$

Equation 13-15.

MCE1 also solves the equation

$$[R_{mm}][G_{mn}] = -[R_{mn}]$$

Equation 13-16.

for the transformation matrix $[G_{mn}]$. An alternate path is available for module MCE1 in SubDMAP XMCE1, requested by PARAM,MCE1,NEW. It contains a partitioned solution for $R_{mm}^{-1} \cdot R_{mn}$, where the part of R_{mm} that is diagonal is partitioned from the coupled part. It was developed at a time when the partitioned solution was faster than the single-pass solution used in MCE1. The single pass solution was improved so that it is now faster than the partitioned solution. The partitioned solution is maintained to provide an alternate solution when the single-pass solution is implausible, or does not complete in a normal manner.

The MCE2 module (Multipoint Constraint Eliminator – Phase 2) partitions the stiffness matrix

$$[K_{gg}] = \begin{bmatrix} \bar{K}_{nn} & K_{nm} \\ K_{mn} & K_{mm} \end{bmatrix}$$

Equation 13-17.

and performs matrix reduction

$$[K_{nn}] = [G_{mn}]^T [K_{mm} G_{mn} + K_{mn}] + [K_{mn}^T G_{mn} + \bar{K}_{nn}]$$

Equation 13-18.

The other structural matrices, $[K_{gg}]$, $[B_{gg}]$ and $[M_{gg}]$, are transformed by formulas that are identical in form to [Eq. 13-13](#), in subDMAPs SEMR2 and SEMRB.

The MCE2 module is also used in subDMAP SEMR2 to obtain $[M_{nn}]$.

The UPARTN module partitions $[K_{gg}]$ to $[K_{mg}]$ form which is equivalent to

$$[K_{mg}] = [K_{mn} \mid K_{mm}]$$

Equation 13-19.

and is used to obtain multipoint forces of constraint. See [“Data Recovery Operations in SubDMAP SEDISP.”](#)

Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB

The single point constraint operations apply constraints to the structural matrices using the single point constraints you specify on SPCi Bulk Data entries, the PS field on GRID entries, or constrained by the AUTOSPC feature described in [“Constraint and Mechanism Problem Identification in SubDMAP SEKR.”](#)

Single point constraints are applied to the set, u_s , in the form

$$\{u_s\} = \{Y_s\},$$

Equation 13-20.

where $\{Y_s\}$ is a vector of enforced deformations, any or all of whose elements may be zero. The set, u_n , is partitioned into u_s and u_f (the free or unconstrained set)

$$\{u_n\} = \begin{Bmatrix} u_f \\ u_s \end{Bmatrix}$$

Equation 13-21.

The stiffness matrix, K_{nn} , is similarly partitioned

$$[K_{nn}] = \left[\begin{array}{c|c} K_{ff} & K_{fs} \\ \hline K_{fs}^T & K_{ss} \end{array} \right]$$

Equation 13-22.

The complete structural equations including the single point forces of constraint, q_s , may be written in partitioned matrix form as

$$\left[\begin{array}{c|c|c} K_{ff} & K_{fs} & 0 \\ \hline K_{fs}^T & K_{ss} & -I \\ \hline 0 & I & 0 \end{array} \right] \left\{ \begin{array}{c} u_f \\ u_s \\ q_s \end{array} \right\} = \left\{ \begin{array}{c} \bar{P}_f \\ P_s \\ Y_s \end{array} \right\}$$

Equation 13-23.

Straightforward elimination gives

$$[K_{ff}]\{u_f\} = \{\bar{P}_f\} - [K_{fs}]\{Y_s\} = \{P_f\}$$

Equation 13-24.

The forces of constraint are recovered by means of the middle row of [Eq. 13-23](#), i.e.,

$$\{q_s\} = -\{P_s\} + [K_{fs}^T]\{u_f\} + [K_{ss}]\{u_s\}$$

Equation 13-25.

The distinct partitions K_{fs} and K_{ss} , are needed in subsequent calculations of SPC forces in static analysis. For the other structural matrices (K_{nn}^4 , B_{nn} , and M_{nn}) the (sf) and (ss) partitions are saved, for dynamic SPC forces calculation.

The partially constrained stiffness matrix $[K_{nn}]$ is partitioned by UPARTN module as follows:

$$[K_{nn}] = \left[\begin{array}{c|c} K_{ff} & K_{fs} \\ \hline \bar{K}_{sf} & \bar{K}_{ss} \end{array} \right]$$

Equation 13-26.

Similar partitioning operations are performed on $[M_{nn}]$ in subDMAP SEMR2 to obtain $[M_{ff}]$, $[M_{sf}]$ and M_{ss} . In subDMAP SEMRB, the structural and viscous damping matrices $[K_{sf}^4]$, $[K_{ss}^4]$, $[B_{sf}]$ and $[B_{ss}]$

are obtained from the MATREDU module. All (sf) and (ss) matrices are required for the recovery of single point constraint reaction forces described in “Data Recovery Operations in SubDMAP SEDISP.”

Constraint and Mechanism Problem Identification in SubDMAP SEKR

A singularity in a linear equation solution leads to a condition where a unique solution is not possible. For example, if $[K]\{u\} = \{F\}$, and $[K]$ is zero, then if $\{F\}$ is not zero, no value of $\{u\}$ will satisfy the equation. However, if $\{F\}$ is zero, any value of $\{u\}$ will satisfy the equation. The equation can always be satisfied by prescribing $\{u\}$, rather than $\{F\}$. This is equivalent to constraining u with a single-point constraint.

There are two types of singularity:

1. The singularity can be identified by considering the stiffness terms of only one grid point. This is called a grid point singularity.
2. The singularity requires consideration of stiffness terms of more than one grid point. This is called a mechanism type of singularity.

Grid Point Singularity

Grid point singularities are automatically identified by the GPSP module. They may be automatically constrained, under user control. First, multipoint constraints are eliminated, leading to the matrix $[K_{nn}]$ (see “Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB”). Singularities are identified by grid point and component number of the failed direction. The ratio of stiffness at the degree-of-freedom to the largest stiffness at that grid point is listed. If automatic constraint is selected (see “AUTOSPC” in the *NX Nastran Quick Reference Guide*), the set membership of the singular degrees-of-freedom is listed before the automatic constraint and after the automatic constraint. Both the mutually exclusive set and the highest combined set identical to the mutually exclusive set are listed for each identified singularity.

The output will include a Grid Point Singularity Table with the following format:

GRID POINT SINGULARITY TABLE						
POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET	NEW USET	
3801	G	1	0.00E+00	L	S	*
3801	G	2	0.00E+00	L	S	*
3801	G	3	0.00E+00	L	S	*
3801	G	4	0.00E+00	L	S	*
3801	G	5	0.00E+00	L	S	*
3801	G	6	0.00E+00	L	S	*

In this table,

1. G (under the heading TYPE) stands for grid point (S for scalar point is also output).
2. FAILED DIRECTION is the degree-of-freedom component in the displacement coordinate system and closest to the weakest direction.
3. The STIFFNESS RATIO is the ratio of the stiffness in the weakest direction to the stiffness in the strongest direction at the grid point (considering all possible directions rather than just the coordinate directions). Translational and rotational stiffnesses are considered separately. The default value for the failure criterion is 10^{-8} , which may be changed by the PARAM,EPZERO entry.

- Note that, for the example shown, the failed degrees-of-freedom were originally in the u_i set and were placed in the u_s set. This may not always be the case, for reasons noted earlier, or because you didn't request the AUTOSPC feature.
- You should carefully inspect the Grid Point Singularity Table because a singularity is very often the result of a modeling error, which should be corrected.

User Options

The identification of singularities is an automatic feature; however, you may exercise the following options through PARAM Bulk Data entries described in “Parameters” in the *NX Nastran Quick Reference Guide*.

Parameter Name	Description
AUTOSPC	Controls automatic constraint of a singularity.
EPPT	Specifies the maximum value of a singularity to print.
EPZERO	Specifies the maximum value of a singularity.
PRGPST	Controls printout of singularity table.
SPCGEN	Writes SPC entries to the PUNCH file based on the singularity table.

The singular degrees-of-freedom are shifted to other sets by the following logic if PARAM,AUTOSPC,YES is specified:

o-set Condition	Set Before Constraint	Set After Constraint
If o-set exists	o	s
	c	b
If no o-set	c	s
	b	s

Other mutually exclusive sets not listed above will not be constrained. In addition, singular degrees-of-freedom on CYAX, CYJOIN, and SPCOFFi entries are identified but not constrained. The most usual cause of grid point singularity is degrees-of-freedom not constrained by elements because they are not of interest to the modeler. An example is out-of-plane rotations on a component made of plates lying in a plane. The automatic constraint provides the correct action for this type of singularity. Another cause is elements left out by oversight. Here, one typically observes that many or all degrees-of-freedom at a grid point are singular. The automatic constraint may mask a modeling error in this instance. For this reason, all singularity messages should be carefully inspected during the modeling checkout phase to ensure that modeling errors are not being masked.

When using the K2PP, M2PP, B2PP, or TFL Case Control command, PARAM,AUTOSPC,NO is recommended.

Mechanism-Type Singularities

A more complicated type of instability arises when two or more grid points participate in a singularity in a linear equation solution. This is known as a mechanism type of instability. Again, for the simple example shown in [Figure 13-2](#), the equations of equilibrium can not be satisfied at all, or may have nonunique solutions for special loading conditions. Note, however, that the structure of this example will pass the grid point singularity checks.

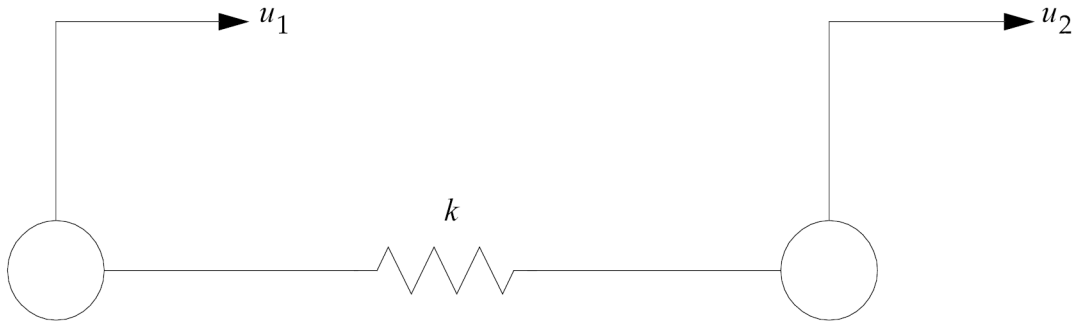


Figure 13-2. A Mechanism Type of Instability

The static equilibrium equation for the example in **Figure A Mechanism Type of Instability** is

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Equation 13-27.

If $P_1 = -P_2$, many solutions are possible, where $(u_1 - u_2) = P_1/k$, but u_1 may have any value. If P_1 and P_2 are arbitrary, the equation cannot be satisfied.

This condition is detected during the matrix decomposition phase of the linear equation solution process by the DCMP module. The stiffness matrix is decomposed into a lower triangular matrix, $[L]$, and a diagonal matrix, $[D]$. As discussed in “User Interface” in the *NX Nastran Numerical Methods User's Guide* a diagonal term of $[D]$ (“ d_i ”) will approach zero for every mode of instability of the structure. (In this example there is one instability.) These instabilities are identified by dividing all d_i 's into their corresponding stiffness term, then listing all ratios larger than a large number. See “MAXRATIO” in the *NX Nastran Quick Reference Guide* for a description of several other parameters used to control this operation.

$$\text{Let } [K] = [L][\backslash D \backslash][L]^T$$

$$\text{Then } [\backslash D \backslash] = \begin{bmatrix} k & 0 \\ 0 & (k - k) \end{bmatrix}, [L] = \begin{bmatrix} 1 & 0 \\ -1 & 1 \end{bmatrix}$$

$$\text{If } (k - k) = 0.0, \text{ replace } d_i \text{ with } 10^{-10}$$

$$Ratio_i = k_{ii} / d_i$$

Identify Grid Component for All Ratios > PARAM,MAXRATIO.

A corrective action for this example would be to constrain one of the grid points or to add springs connecting this spring to other parts of the structure (not shown). Since, in general, there may

be many grid points active in a singularity, there is little reason for picking one over another for connection. The usual cause of this type of instability is elements left out by oversight. While this may be your intention, it cannot be detected by the program. For these reasons, there is no automatic method to constrain mechanisms. The discussion of “MAXRATIO” in the *NX Nastran Quick Reference Guide* describes the various actions and user options that are available in the several contexts where singularities may be detected. Diagnosis techniques and corrective actions are described in the *NX Nastran Numerical Methods User's Guide*.

Static Condensation in SubDMAPs SEKR and SEMR2

Structural partitioning is used to refer structural matrices to the boundary points in substructuring operations and to perform a static condensation in dynamic analyses.

At user option the set of free coordinates, u_f , may be partitioned into two sets, u_o and u_a , such that the u_o set is eliminated first. Thus

$$\{u_f\} = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix}$$

Equation 13-28.

The equilibrium equations after the elimination of constraints (Eq. 13-24) may be written in partitioned form as

$$\begin{bmatrix} K_{aa} & K_{ao} \\ K_{ao}^T & K_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} \bar{P}_a \\ P_o \end{Bmatrix}$$

Equation 13-29.

Rearrange the bottom half of Eq. 13-29:

$$[K_{oo}]\{u_o\} = \{P_o\} - [K_{ao}]^T\{u_a\},$$

Equation 13-30.

and solve for $\{u_o\}$:

$$\{u_o\} = [K_{oo}]^{-1}\{P_o\} - [K_{oo}]^{-1}[K_{ao}]^T\{u_a\}$$

Equation 13-31.

Substitute for u_o into the top half of Eq. 13-29:

$$[\bar{K}_{aa} - K_{ao}K_{oo}^{-1}K_{ao}^T]\{u_a\} = \{\bar{P}_a\} - [K_{ao}][K_{oo}]^{-1}\{P_o\}$$

Equation 13-32.

It is convenient to define the matrix

$$[G_o] = -[K_{oo}]^{-1}[K_{ao}]^T,$$

Equation 13-33.

so the Eq. 13-32 becomes

$$[\bar{K}_{aa} + K_{ao}G_o]\{u_a\} = \{\bar{P}_a\} + [G_o]^T\{P_o\},$$

Equation 13-34.

where advantage is taken of the symmetry of $[K_{oo}]$.

Following the practice of condensation established in preceding subsections,

$$[K_{aa}]\{u_a\} = \{P_a\},$$

Equation 13-35.

where

$$[K_{aa}] = [\bar{K}_{aa}] + [K_{ao}][G_o],$$

Equation 13-36.

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

Equation 13-37.

The $[G_o]$ matrix defined in Eq. 13-33 is obtained practically from the solution of

$$[K_{oo}][G_o] = -[K_{ao}]^T,$$

Equation 13-38.

where $[K_{ao}]^T$ is treated as a set of load vectors. Each such vector produces a column of $[G_o]$.

Once $\{u_a\}$ is obtained the set of omitted coordinates, $\{u_o\}$, is obtained as follows. Define the set

$\left\{u_o^o\right\}$
as the solution of

$$[K_{oo}]\left\{u_o^o\right\} = \{P_o\}$$

Equation 13-39.

Then, using Eq. 13-33 and Eq. 13-39 in Eq. 13-30,

$$\{u_o\} = \left\{u_o^o\right\} + [G_o]\{u_a\}$$

Equation 13-40.

In subDMAP SEKR, the constrained stiffness matrix, $[K_{ff}]$, is partitioned by UPARTN as follows:

$$[K_{ff}] = \left[\begin{array}{c|c} \bar{K}_{aa} & K_{ao} \\ \hline K_{oa} & K_{oo} \end{array} \right]$$

Equation 13-41.

DCMP and FBS are used to solve the equation

$$[K_{oo}][G_{oa}^t] = -[K_{oa}]$$

Equation 13-42.

for the static transformation matrix, $[G_{oa}^t]$, and MPYAD performs the matrix reduction

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

Equation 13-43.

If PARAM,ALTRED,YES is specified then the $[K_{ff}]$ matrix is partitioned and reassembled to form $[K_{ff}^x]$ which has the a-set degrees-of-freedom sequenced last. $[K_{ff}^x]$ is partially decomposed

$$[K_{ff}^x] \rightarrow [L_{fo}], [K_{aa}]$$

Equation 13-44.

where $[L_{fo}]$ is a trapezoidal matrix and $[K_{aa}]$ is the boundary stiffness matrix.

The $[L_{fo}]$ matrix is partitioned

$$[L_{fo}] = \begin{bmatrix} L_{oo} \\ L_{ao} \end{bmatrix}$$

Equation 13-45.

While $[L_{fo}]^T$ is of the same dimension as the $[G_{oa}^t]$ matrix, it is on the average half as dense, thereby reducing the storage requirements.

In subDMAP SEMR2, UPARTN partitions the constrained mass matrix $[M_{ff}]$ as follows:

$$[M_{ff}] = \begin{bmatrix} \bar{M}_{aa} & M_{ao} \\ M_{oa} & M_{oo} \end{bmatrix}$$

Equation 13-46.

and successive executions of MPYAD and SMPYAD modules perform the matrix reduction

$$[M_{aa}] = [G_{oa}^t]^T [M_{oo} \quad G_{oa}^t + M_{oa}] + [M_{oa}^T \quad G_{oa}^t + \bar{M}_{aa}]$$

Equation 13-47.

In subDMAP SEMRB, the structural and viscous damping matrices $[B_{ff}]$ and $[K_{ff}^4]$ are condensed by the MATREDU module.

The partitioned solution for statics is an exact solution. It merely changes the order of operations of the unpartitioned solution. The static reduction for dynamics is exact for o-set degrees-of-freedom that are massless, but represents an approximate relumping of o-set masses to the a-set. You must select enough a-set degrees-of-freedom to adequately define the eigenvectors of the system. The dynamic reduction technique discussed in “[Dynamic Reduction and Component Mode Synthesis in SubDMAP SEMR3](#)” provides a way of improving the approximation by computing the normal modes for the o-set degrees-of-freedom.

Static Solutions in SubDMAP SEKRRS

There are two major branches through this subDMAP, standard static analysis and static analysis with inertia relief effects.

Inertia Relief Not Selected

When inertia relief is not selected and there are no SUPORTi entries are present the a-set equations are solved directly

$$K_{aa} \cdot u_a = P_a$$

Equation 13-48.

K_{aa} is decomposed into its factor matrices L_{aa} and D_a . These factors are passed on to SubDMAP STATRS for solution of u_a .

When inertia relief is not selected but there are SUPORTi entries are present the a-set equations are partitioned into the r-set, defined on the SUPORTi entries, and the l-set, what is “left over” after the r-set is removed.

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{bmatrix} u_l \\ u_r \end{bmatrix} \geq \begin{bmatrix} P_l \\ P_r \end{bmatrix}$$

Equation 13-49.

In this context, the r-set is treated as an alternate form of single-point constraints. The partitions of K involving r are discarded. u_r is set to zero. u_l is found from the first row of this equation. Loads in P_r go directly to ground, and are output with the SPC forces, along with the product $-K_{rl} \cdot u_l$.

Inertia Relief Selected

The basic assumption of inertia relief is that the structure is in a state of steady acceleration, and that all transients due to dynamic effects are negligible. The solution is obtained in a moving reference frame from which all displacements are measured. For inertia relief analysis inertial loads are computed automatically that exactly balance the applied loads. A support system is applied to make the system numerically stable. The inertial loads assume rigid body accelerations when computing the loads.

The derivations given here assume that the system mass matrix M_{gg} has been reduced to M_{aa} and is available, as described elsewhere. In the actual DMAP, equivalent methods that never generate M_{aa} are used instead, in the interests of efficiency. M_{aa} can be a larger size data block, and costly to compute. It can be shown that the results provided by the two derivations are identical. For example, the rigid body mass matrix M_{rr} , defined below as

$$M_{rr} = D_{ar} \cdot M_{aa} \cdot D_{ar}$$

Equation 13-50.

can just as easily be formed from the g-set equations

$$M_{rr} = D_{gr} \cdot M_{gg} \cdot D_{gr}$$

Equation 13-51.

Similar comments can be made about computing the inertial loads that balance the applied loads. The derivatives are presented here in the form of a-set equations that are more readily understood,

if less efficient in practice, than the g-set equations. You can find equations for the more efficient method TANS 4002 and 4854 or in DMAP compilations. Current research indicates that this capability may shift to a-set operations in future versions to obtain more automation.

There are at present two forms of inertia relief: automatic selection of a support system (“auto-support”); and manual selection of the support system, using SUPORTi entries. Both react to PARAM,GRDPNT, the parameter used to locate a reference point for grid point weight summaries. The more automatic method is described first.

Auto-Support (PARAM,INREL,-2)

Vectors of rigid body motion D_{gr} for six unit motions of a reference point are computed from grid point geometry in module VECPLT. The D_{ar} component is partitioned from this matrix. D_{ar} is used to compute the rigid body mass M_{rr} about the reference point.

$$MD_{ar} = M_{aa} \cdot D_{ar}$$

Equation 13-52.

$$M_{rr} = D_{ar} \cdot MD_{ar}$$

Equation 13-53.

M_{rr} is printed out with the matrix name of QRR.

The acceleration of the reference point ar due to the applied loads is computed,

$$M_{rr} \cdot ar \geq D_{ar} \cdot P_a$$

Equation 13-54.

The accelerations ar are printed with the matrix name of URA.

The resulting inertial loads P_{ai} and total loads P_a are

$$P_{ai} = M_{aa} \cdot ar$$

Equation 13-55.

$$P_{a1} = P_a - P_{ai}$$

Equation 13-56.

A constraint equation is written that states that the average weighted motion of the system is zero,

$$MD_{ar} \cdot u_{ar} = 0$$

Equation 13-57.

This equation is appended to the a-set to define the 1-set equations in their augmented form,

$$K_{ll} \cdot u_l = P_l$$

Equation 13-58.

$$\begin{bmatrix} K_{aa} & MD_{ar} \\ MD_{ar} & 0_{rr} \end{bmatrix} \begin{bmatrix} u_a \\ -q_r \end{bmatrix} = \begin{bmatrix} P_{a1} \\ 0_r \end{bmatrix}$$

Equation 13-59.

q_r are the unknown forces required on the system to produce equilibrium. They should be computational zeros, and are output with the SPC forces. The equation involving K_{ll} is solved directly, with u_a and q_r partitioned from u_l .

Manual Support (PARAM, INREL, -1 and SUPORTi)

You prescribe a set of DOFs that constrain the structure in a statically-determinate manner on SUPORTi entries. These entries define the r-set. There must be six or less DOFs listed, and they must constrain all rigid body modes that are not constrained by SPCs or other modeling devices. The rigid body modes are computed from the statics equation, where there are no loads applied, and u_{rr} is the identity matrix. Given

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{le} & K_{rr} \end{bmatrix} \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix} = \begin{bmatrix} 0_{lr} \\ 0_{rr} \end{bmatrix}$$

Equation 13-60.

D_{lr} is found by solving the first row of this equation, then merged with I_{rr} to form D_{ar} .

$$K_{ll} \cdot D_{lr} \geq -K_{lr} \cdot I_{rr}$$

Equation 13-61.

The rigid body mass, reference point accelerations, and inertial loads are formed with the same equations used for auto-support. The loads are partitioned into the l- and r-sets, and the partitioned form of K_{ll} is used for solution for displacements,

$$K_{ll} \cdot u_l = P_{ll}$$

Equation 13-62.

u_a is merged from u_l and a null partitioned for u_r .

A Comparison of Auto- and Manual-Support Solution

If one analyzes the same model with both methods, you should find that stresses, constraint forces, and other internal quantities are identical except for truncation errors, but the displacements may differ appreciably. This is because they use a different reference frame. For the many supports, the reference frame is set up so that the motion at the r-set DOFs are identically zero. One can think of a reference frame drawn through these points, and all displacements measured relative to it. This reference frame is moving through space at a constant acceleration rate, so the displacements are relative, rather than inertial displacements.

For the auto-support option, one can visualize the reference frame as a set of axes connected to every point with mass with a very complicated set of levers that allows the structure to “breathe”, as loads are applied to it, but keep the weighted average motion to a zero value. If the model happens to have a grid point at the reference point, you will observe very small motion there, computational zeros but not the binary zeros of the manual support method. This is because this motion is computed for the auto-support method, but assumed to be zero for the manual method.

The choice of an r-set as defined by SUPORTi DOFs requires meeting both a geometric requirement and a stiffness requirement. The geometric requirement is that the set should form a statically determinate tie to ground. This can be difficult to do with complicated geometries. The stiffness requirement is that the DOFs should be stiff enough to provide a stable solution. A method that has been used by some modelers is to pick points stiff enough that they could be used to lift the structure without damaging it. Sometimes such points do not exist. The difficulties in picking a stable, reliable support system was one of the main motivations for development of the more modern auto-support systems.

PARAM, GRDPNT, GID is optional for both methods when superelements are not present, but required by the manual method when superelements are present. The auto-method does not support superelement analysis. The default for GRDPNT is GID = -1, which is a flag to use the basic origin as the reference point when computing rigid body modes. Use of a GID near the c.g. of the structure produces more esthetically pleasing results. The rigid body accelerations produced by the solution are then measured near the center of the structure, rather than at a point that may be outside of the structure.

Static and Dynamic Load Generation**Applied Static Loads in SubDMAPs SELG, SELR, and SELRRS**

The static load vectors $\{P_g\}$ are generated by SSG1 (Static Solution Generator – Phase 1) in subDMAP SELG from your input of applied loads, temperature fields, and enforced deformations.

The direct matrix input requested by the P2G Case Control command define the load matrix $[P_g^2]$ which is assembled by the MTRXIN module and is added to the output from SSG1.

If PARAM,INREL,-1 is specified, then

1. The rigid body transformation matrix $[D_{gr}]$ is generated with the VECPLOT module. Its columns contain the motion of all degrees-of-freedom due to unit motion of the corresponding reference point degree-of-freedom.
2. Loads due to unit rigid body accelerations are computed,

$$\{P_j^i\} = -[M_{jj}][D_{gr}]$$

3. The loads above are appended to the applied loads $\{P_j\}$,

$$\{P_j\} \leftarrow \{P_j \mid P_j^i\}$$

The SELA module assembles $\{P_j\}$ into $\{P_g\}$.

The constraints are applied to the static load vectors by SSG2 (Static Solution Generator – Phase 2) in subDMAP SELR as follows:

$$\{P_g\} = \begin{Bmatrix} \bar{P}_n \\ P_m \end{Bmatrix}$$

Equation 13-63.

$$\{P_n\} = \{\bar{P}_n\} + [G_{mn}]^T \{P_m\}$$

Equation 13-64.

$$\{P_n\} = \begin{Bmatrix} \bar{P}_f \\ P_s \end{Bmatrix}$$

Equation 13-65.

$$\{P_f\} = \{\bar{P}_f\} - [K_{fs}]\{Y_s\}$$

Equation 13-66.

$$g \frac{\bar{P}_f}{\omega} = \frac{\bar{P}_f}{\omega} \frac{C}{C_{cr}}$$

Equation 13-67.

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

Equation 13-68.

If PARAM,ALTRED,YES is specified, the diagonal extracted from $[L_{oo}] \rightarrow [D_{oo}]$ and the reduced load matrix is computed from

$$\{P_a\} = \{\bar{P}_a\} - [L_{ao}][D_{oo}][u_o^x]$$

Equation 13-69.

where $\{u_o^x\}$ is obtained on a forward pass from the equation

$$[L_{oo} \ D_{oo}]\{u_o^x\} = \{P_o\}$$

Equation 13-70.

for PARAM,ALTRED,NO, the SSG3 module (Static Solution Generation – Phase 3) solves the equation

$$[K_{oo}]\{u_o^o\} = \{P_o\}$$

Equation 13-71.

for $\{u_o^o\}$, the displacements of the omitted coordinates.

SSG3 also calculates the residual vector, δP_o , and the residual vector error ratio, ε_o , for the omitted coordinates

$$\{\delta P_o\} = [K_{oo}]\{u_o^o\} - \{P_o\}$$

Equation 13-72.

$$\varepsilon_o = \frac{\{u_o^o\}^T \{\delta P_o\}}{\{P_o\}^T \{u_o^o\}}$$

Equation 13-73.

Except for round-off error, the error ratio ε_o should be zero. Large values of these error ratios usually indicate singularities in the stiffness matrix. The residual load vector, RUOV, may be output by use of PARAM,IRES,1.

The quantity

$$1/2 \{P_o\}^T \{u_o\}$$

Equation 13-74.

is calculated by the SSG3 module and printed under the heading “External Work.” This component of strain energy includes effects of thermal loads, element deformations, and enforced displacements that may be subtracted later in the solution process. For example, an enforced displacement that causes zero strain will result in external work.

If a r-set is present, subDMAP SELRRS calculates determinate forces of reaction

$$[q_r] = -[D_{ar}]^T \{P_a\}$$

Equation 13-75.

where:

$$[D_{ar}] = \begin{bmatrix} D_{er} \\ I_{rr} \end{bmatrix}$$

Applied Dynamic Loads

Dynamic loads are produced in many subDMAPs. They are generally of the form:

$$P(x) = \sum_i PO_i \cdot F_i(x)$$

Equation 13-76.

The PO_i vectors are spatial functions that describe what DOFs are loaded and their scale factors. The F_i functions are temporal functions that define how the loads change with time in transient analysis $x = t$, or with excitation frequency in frequency response analysis $x = f$. All spatial loads except those on extra points are calculated in the static analysis load generator module, SSG1, to obtain the P_g vector described in the prior paragraphs.

Enforced motions are produced similarly.

$$Y(x) = \sum_i YO_i \cdot F_i(x)$$

Equation 13-77.

The YO_i vectors are again the spatial distribution, and the $F_i(x)$ temporal functions. The YO_i vectors are produced by the GP4 module to produce the Y_g vector. It is the same vector described for static analysis in [Eq. 13-66](#), except that it is merged to g-set size with null terms from Y_s .

The static load and enforced motion vectors are converted into dynamic loading tables in DPD module, then passed on to the dynamic load generators, FRLG and TRLG. These modules produce the applied loads and enforced motions at the p-set level, $Pp(x)$ and $Yp(x)$, as a function of excitation frequency or time. For transient analysis velocity and acceleration vectors are also provided, $\dot{Y}(x)$ and $\ddot{Y}(x)$.

A reduction matrix Rpx is made in the FDRMGGEN module, where the subscript "x" implies the d-set, for direct analysis, and h, for modal analysis. It combines the operations shown in Eq. 13-63 through Eq. 13-68 previously. In subDMAP CFORCE2 Eq. 13-66 is modified to add velocity and acceleration effects,

$$P_f = P_f - K_{fs} \cdot Y_s - B_{fs} \cdot \dot{Y}_s - M_{fs} \cdot \ddot{Y}_s$$

Equation 13-78.

In frequency response the velocity and acceleration effects can be computed from Y_s by multiplying it by $i \cdot \omega$ once or twice, where ω is the excitation frequency.

P_f is reduced to P_d or P_h by the same techniques used in static analysis.

For modal analysis with enforced motion it is essential that PARAM,RESVEC,YES be used. It causes static solutions to be formed from $K_{fs} \cdot Y_s$ -type terms. When the structure is tied to ground in a statically-determinate manner, these solutions are linear combinations of the rigid body modes that would exist if the ties to ground were removed. If the ties to ground are redundant, a condition named multi-base input, the resulting shapes will have significant stresses associated with them. They represent the effect of low-frequency excitation on the structure when PARAM,RESVEC,YES is not present, for the statically determinate case the element stresses will be correct but the displacement-type data will be missing the effect of the static shapes. For the multi-base input case, both stresses and displacement-type output will be incorrect.

Dynamic Reduction and Component Mode Synthesis in SubDMAP SEMR3

These capabilities use several subsets of the a-set not used by other solution techniques. The v-set matrix is formed in subDMAP SEKR

$$K_{vv} = \begin{bmatrix} K_{oo} & K_{or} & K_{oc} \\ K_{ro} & K_{rr} & K_{rc} \\ K_{co} & K_{cr} & K_{cc} \end{bmatrix}$$

Equation 13-79.

The v-set mass matrix is formed similarly in subDMAP SEMR3. The v-set degrees-of-freedom are free to vibrate during dynamic reduction and component mode computation. They consist of the omitted terms (o-set), the free boundary points used for rigid-body mode designation (r-set), and the other free boundary points (c-set). The $[M_v]$ matrix is formed in a similar manner.

If the METHOD command is specified for superelements, then the component modes are computed. The auto-omit operations described in “**Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS**” are carried out resulting in the reduced eigensolution equation

$$[K_{yy} - \lambda M_{yy}][\phi_{yz}] = 0$$

Equation 13-80.

where N_z is the number of eigenvectors requested by the EIGR or EIGRL Bulk Data entries. The back-transformations for the auto-omit-spc are performed, resulting in the expanded eigenvector matrix $[\phi_{yz}]$. This matrix is used to purify the approximate eigenvectors found in dynamic reduction,

$$[\phi_{vz}] = [\phi_{vy}][\phi_{yz}]$$

Equation 13-81.

The uncoupled eigensolution components are assembled from $[\phi_{vz}]$

$$[\phi_{vz}] = \begin{Bmatrix} \phi_{oz} \\ \phi_{rz} \\ \phi_{cz} \end{Bmatrix}$$

Equation 13-82.

$$[\phi_{az}] = \begin{Bmatrix} \phi_{rz} \\ \phi_{cz} \\ \phi_{bz} \end{Bmatrix}$$

Equation 13-83.

where $[\phi_{bz}] = [0]$

$[\phi_{oz}]$ and $[\phi_{az}]$ are stored for use with the uncoupled solution obtained using PARAM,FIXEDB,-1 as described in “**Data Recovery Operations in SubDMAP SEDISP**.”

The residual vector capability is requested with PARAM,RESVEC,YES. Details on this capability are given in “**Matrix Assembly Operations in SubDMAP SEMG**,” where real eigensolutions are discussed. The RESVEC SubDMAP returns auxiliary functions u_{o2} . They are appended to the eigenvectors, and are given all of the operations given to eigenvectors,

$$H_{vz} = \begin{bmatrix} \phi_{oz} & u_{o2} \\ \phi_{rz} & 0_{r2} \\ \phi_{cz} & 0_{c2} \end{bmatrix}$$

Equation 13-84.

The inertia relief mode shapes are calculated by statically applying the rigid-body inertial loads to the structure.

$$\begin{bmatrix} K_{oo} & K_{oa} \\ K_{ao} & K_{aa} \end{bmatrix} \begin{bmatrix} H_o^i \\ H_a^i \end{bmatrix} = \begin{bmatrix} M_{oo} & M_{oa} \\ M_{ao} & M_{aa} \end{bmatrix} \begin{bmatrix} V_o \\ V_a \end{bmatrix}$$

Equation 13-85.

Setting $[H_a^i]$ to zero and solving for $[H_o^i]$ results in,

$$[H_o^i] = [K_{oo}]^{-1} [[M_{oo}][V_o] + [M_{oa}][V_a]]$$

Equation 13-86.

The $[G_{oq}]$ matrix is formed,

$$[G_{oq}] \leftarrow [G_{oz}^1 \mid H_o^i \mid 0]$$

Equation 13-87.

The $[G_{oq}]$ matrix is truncated or padded with null columns to provide Nq columns, where Nq is the q-set size. $[G_{oq}]$ is expanded to a-size,

$$[G_{oq} \mid 0_{ot}] \rightarrow [G_{oa}^q]$$

Equation 13-88.

The total transformation matrix is formed by adding dynamic and static components and is reassembled whenever needed,

$$[G_{oa}] = [G_{oa}^t] + [G_{oa}^q]$$

Equation 13-89.

The generalized stiffness coefficients are formed from the dynamic transformation,

$$[K_{qq}] = [G_{oq}]^T [K_{oo}] [G_{oq}]$$

Equation 13-90.

The physical and generalized degrees-of-freedom are not stiffness coupled. $[K_{qq}]$ is merged to N_a size,

$$[K_{aa}^l] = \left[\begin{array}{c|c} K_{qq} & 0 \\ \hline 0 & 0 \end{array} \right]$$

Equation 13-91.

where superscript / refers to matrices formed from dynamic reduction. $[K_{qq}]$ is stored in the database. When the total stiffness boundary matrix is needed, it is formed from its partitions,

$$[K_{aa}^m] = \left[\begin{array}{c|c} K_{qq} & 0 \\ \hline 0 & \bar{K}_{tt} \end{array} \right]$$

Equation 13-92.

or

$$[K_{aa}] = [K_{aa}^m] + [K_{aa}^l]$$

Equation 13-93.

The physical and generalized degrees-of-freedom are mass-coupled. The terms are formed from the equations,

$$\begin{aligned} [M_{qq}] &= [G_{oq}]^T [M_{oo}] [G_{oq}] \\ [M_{qt}] &= [G_{oq}]^T [M_{ot} + M_{oo} \quad G_{oa}^t] \\ [M_{aa}^l] &= \left[\begin{array}{c|c} M_{qq} & M_{qt} \\ \hline M_{tq} & 0 \end{array} \right] \end{aligned}$$

Equation 13-94.

Virtual mass effects, if any, are added to $[M_{aa}^l]$ to form $[M_{aa}^l]$.

The total mass matrix is formed by:

$$[M_{aa}^m] = [M_{aa}] + [M_{aa}^l]$$

Equation 13-95.

For the rigid formats $[M_{aa}]$ is formed directly from its partitions,

$$[M_{aa}] = \left[\begin{array}{c|c} M_{qq} & M_{qt} \\ \hline M_{tq} & M_{tt} \end{array} \right]$$

Equation 13-96.

The a-set matrices do not contain static effects in the t-set components, and dynamic effects in the q-set components. Virtual mass effects, if any, are added to $[M_{aa}]$ to form $[M_{aa}^m]$.

The damping matrices $[B_{gg}]$ and $[K_{gg}^4]$ are reduced to the a-set by the MATREDU module which performs the following operations:

1. Eliminate multipoint constraints

$$[B_{gg}] \Rightarrow \left[\begin{array}{cc} \bar{B}_{nn} & B_{nm} \\ B_{mn} & B_{mm} \end{array} \right]$$

$$[B_{nn}] = [G_{mn}]^T [B_{mn} G_{mn} + B_{mn}] + [B_{mn}]^T [G_{mn}] + [B_{nn}]$$

2. Eliminate single point constraints

$$[B_{nn}] \Rightarrow \left[\begin{array}{cc} B_{ff} & B_{fs} \\ B_{sf} & B_{ss} \end{array} \right]$$

3. Partition omitted degrees-of-freedom

$$[B_{ff}] \Rightarrow \left[\begin{array}{cc} \bar{B}_{aa} & B_{ao} \\ B_{oa} & B_{oo} \end{array} \right]$$

4. Perform static condensation

$$[B_{aa}] = [G_{oa}]^T [B_{oo} G_{oa} + B_{oa}] + [\bar{B}_{oa}]^T [G_{oa}] + [\bar{B}_{aa}]$$

Formulation of Dynamic Equations in SubDMAP GMA

The tables prepared by the geometry processors are modified by DPD (Dynamic Pool Distributor) to include the extra points (e-points) introduced for dynamic analysis.

$[K^m_{aa}]$, $[M^m_{aa}]$, $[B_{aa}]$, and $[K^4_{aa}]$ are expanded to the d-set to form $[K^1_{dd}]$, $[M^1_{dd}]$, $[B^1_{dd}]$, and $[K^4_{dd}]$. The direct input matrices $[K^2_{pp}]$, $[M^2_{pp}]$, and $[B^2_{pp}]$ requested by the K2PP, M2PP, and B2PP Case Control commands are reduced to the d-set to form $[K^2_{dd}]$, $[M^2_{dd}]$, and $[B^2_{dd}]$, respectively.

The general dynamic equation used in the direct methods is

$$[[M_{dd}]p^2 + [B_{dd}]p + [K_{dd}]]\{U_d\} = \{P_d\}$$

Equation 13-97.

p is the derivative operator

$$\{U_d\} = \begin{Bmatrix} U_a \\ U_e \end{Bmatrix}$$

Equation 13-98.

where:

$\{U_d\}$ = the extra points

For direct frequency response and complex eigenvalue analysis the stiffness, damping, and mass matrices are generated as follows:

$$[K_{dd}] = (1 + ig)[K^1_{dd}] + [K^2_{dd}] + i[K^4_{dd}]$$

Equation 13-99.

$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}]$$

Equation 13-100.

and

$$[M_{dd}] = [M^1_{dd}] + [M^2_{dd}]$$

Equation 13-101.

For direct transient response the stiffness, damping and mass matrices are generated as follows:

$$[K_{dd}] = [K_{dd}^1] + [K_{dd}^2]$$

Equation 13-102.

$$[B_{dd}] = [B_{dd}^1] + [B_{dd}^{2x}] + \frac{g}{\omega_3}[K_{dd}^1] + \frac{1}{\omega_4}[K_{dd}^4]$$

Equation 13-103.

and

$$[M_{dd}] = [M_{dd}^1] + [M_{dd}^2]$$

Equation 13-104.

where:

- $[K_{dd}^1]$ is the reduced structural stiffness matrix plus the reduced direct input K2GG (symmetric).
- $[K_{dd}^2]$ is the reduced direct input matrix K2PP plus the reduced transfer function input (symmetric or unsymmetrical).
- $[K_{dd}^4]$ is the reduced structural damping matrix obtained by multiplying the stiffness matrix $[K_e]$ of an individual structural element by an element damping factor g_e and combining the results for all structural elements (symmetric) plus the reduced direct input K42GG (symmetric).
- $[B_{dd}^1]$ is the reduced viscous damping matrix plus the reduced direct input B2GG (symmetric).
- $[B_{dd}^2]$ is the reduced direct input matrix B2PP plus the reduced transfer function input (symmetric or unsymmetrical).
- $[M_{dd}^1]$ is the reduced mass matrix plus the reduced direct input M2GG (symmetric).
- $[M_{dd}^2]$ is the reduced direct input matrix M2PP plus the reduced transfer function input (symmetric or unsymmetrical).
- g is the structural damping coefficient on the PARAM Bulk Data entry G. The frequencies ω_3 and ω_4 are specified on the PARAM Bulk Data entries as W3 and W4, respectively.

For direct solutions $[K_{dd}]$, $[M_{dd}]$, and $[B_{dd}]$ are examined to identify rows and columns which are null in all three matrices. If the parameters ASING = 0 (the default) the singularities caused by the null rows and columns are removed as follows:

1. For direct frequency and direct transient response, the stiffness matrix $[K_{dd}^x]$ is formed from $[K_{dd}]$ by placing unity on the diagonal for each null row and column.
2. For direct complex eigenvalue analysis null rows and columns are discarded from $[K_{dd}]$, $[M_{dd}]$, and $[B_{dd}]$ to form $[K_{dd}^d]$, $[M_{dd}^d]$, and $[B_{dd}^d]$. This is performed in subDMAP DCEIGRS. See [“Complex Eigenvalue Analysis in SubDMAP CEIGRS.”](#)

If parameter ASING = -1 and null rows and/or columns exist, a fatal error will result.

In modal frequency response, and complex eigenvalue analysis, structural damping and viscous damping are included in the dynamic matrices as follows:

$$[K_{dd}^2] = [K_{dd}^{2x}] + ig[K_{dd}^1] + i[K_{dd}^4]$$

Equation 13-105.

and

$$[B_{dd}^2] = [B_{dd}^1] + [B_{dd}^{2x}]$$

Equation 13-106.

In modal transient response, the structural damping and viscous damping are included as follows:

$$[K_{dd}^2] = [K_{dd}^{2x}]$$

Equation 13-107.

and

$$[B_{dd}^2] = [B_{dd}^1] + [B_{dd}^{2x}] + \frac{g}{\omega_3}[K_{dd}^1] + \frac{1.0}{\omega_4}[K_{dd}^4]$$

Equation 13-108.

The general dynamic equation used in the modal method is:

$$[[M_{hh}]p^2 + [B_{hh}]p + [K_{hh}]]\{U_h\} = \{P_h\}$$

Equation 13-109.

where p is the derivative operator

$$\{U_h\} = \begin{Bmatrix} \zeta_i \\ U_e \end{Bmatrix}$$

Equation 13-110.

ζ_i are the modal coordinates.

The transformation from $\{U_h\}$ to $\{U_d\}$ is

$$\{U_d\} = [\phi_{dh}]\{U_h\}$$

Equation 13-111.

where:

$$[\phi_{dh}] = \begin{bmatrix} \phi_{di} & 0 \\ 0 & I_{ee} \end{bmatrix}$$

The dynamic matrices expressed in modal coordinates are assembled in GKAM (General K Assembler-Modal) as follows:

$$[K_{hh}] = [k] + [\phi_{dh}]^T [K_{dd}^2] [\phi_{dh}]$$

Equation 13-112.

$$[M_{hh}] = [m] + [\phi_{dh}]^T [M_{dd}^2] [\phi_{dh}]$$

Equation 13-113.

and

$$[B_{hh}] = [b] + [\phi_{dh}]^T [B_{dd}^2] [\phi_{dh}]$$

Equation 13-114.

where, if PARAM,KDAMP = 1 (default)

$$m_i = \text{modal mass}$$

$$b_i = 2\pi f_i g(f_i) m_i$$

$$k_i = 4\pi^2 f_i^2 m_i$$

If PARAM,KDAMP = -1 in complex eigenvalue analysis, modal frequency response, aerodynamic flutter, and aeroelastic response the viscous modal damping will be incorporated into the complex stiffness matrix as follows:

$$\begin{aligned}
 m_i &= \text{modal mass} \\
 b_i &= 0 \\
 k_i &= [1 + ig(f_i)]4\pi^2 f_i^2 m_i
 \end{aligned}$$

Equation 13-115.

Values of $g(f_i)$ are specified on the TABDMP1 Bulk Data entry, selected by the SDAMPING Case Control command. If KDAMP = 1 (default), the matrices $[b]$ and $[k]$ are formulated as in modal transient response.

In SubDMAP CMPMODE damping of CMS (o-set) modes is calculated by similar techniques. This q-set damping is expanded to g-set size, and placed in the B_{aa} matrix for summing into downtown superelements.

Data Recovery Operations in SubDMAP SEDISP

The dependent displacements and the single-point forces of constraint are calculated in SDR1 (Stress Data Recovery – Phase 1). The inputs to SDR1 are columns of the solution vectors corresponding to each eigenvalue, loading condition, or output time step. In the case of transient analysis, there are three columns, corresponding to displacement, velocity and acceleration, for each output time step.

Load Modification

If PARAM,INREL,-1 or PARAM,INREL,-2 are specified, then any component of the load vector is first multiplied by $[C_{rx}]$ to produce a set of loads that are a linear combination of applied and inertial loads called $\{P^i\}$

$$\{P^i\} \leftarrow [P_l][C_{rx}]$$

Equation 13-116.**Static Analysis**

In the case of static analysis and PARAM,ALTRED,NO, SDR1 recovers dependent displacements

$$\begin{Bmatrix} u_l \\ u_r \end{Bmatrix} \Rightarrow \{u_a\}$$

Equation 13-117.

$$\begin{Bmatrix} u_a \\ - \\ u_o \end{Bmatrix} \Rightarrow \{u_f\}$$

Equation 13-118.

where

$$\{u_o\} = [G_{oa}^t]\{u_a\} + \{u_o^o\}$$

Equation 13-119.

If PARAM,ALTRED,YES is specified, then the displacements $\{u_o\}$ are obtained from a backward pass from the equation

$$[L_{oo}]^T \{u_o\} = \{u_o^y\}$$

Equation 13-120.

where $\{u_o^y\} = \{u_o^x\} - [L_{ao}]^T \{u_a\}$

Enforced displacements are combined with u_f

$$\begin{Bmatrix} u_f \\ - \\ Y_s \end{Bmatrix} \Rightarrow \{u_n\}$$

Equation 13-121.

$$\begin{Bmatrix} u_n \\ - \\ u_m \end{Bmatrix} \Rightarrow \{u_g\}$$

Equation 13-122.

where

$$\{u_m\} = [G_{mn}]\{u_n\}$$

Equation 13-123.

and recovers single point forces of constraint

$$\{q_s\} = [K_{fs}]^T [u_f] + [K_{ss}]\{Y_s\} - \{P_s\}$$

Equation 13-124.

and multipoint forces of constraint

$$q_{mg} = \begin{bmatrix} I_{mm} \\ -G_{mn}^t \end{bmatrix} [[K_{mg}]\{u_g\} - \{P_m\}]$$

Equation 13-125.

In superelement static analysis, if PARAM,FIXEDB,-1 is present, then u_a is set to zero, and data recovery is performed using the modified equation

$$\{u_o\} = \{u_o^o\}$$

Equation 13-126.

This is equivalent to obtaining a solution with all exterior (“boundary”) degrees-of-freedom fixed. Displacements are due only to interior displacements due to interior loads, that is, the $\{u_o^o\}$ component.

Eigenvalue Analysis

In the case of eigenvalue analysis SDR1 recovers dependent components of the eigenvectors

$$\{\phi_o\} = [G_{oa}]\{\phi_a\}$$

Equation 13-127.

$$\left\{ \begin{array}{c} \phi_a \\ \phi_o \end{array} \right\} = \{\phi_f\}$$

Equation 13-128.

$$\left\{ \begin{array}{c} \phi_f \\ \phi_s \end{array} \right\} = \{\phi_n\}$$

Equation 13-129.

$$\{\phi_m\} = [G_{mn}]\{\phi_n\}$$

Equation 13-130.

$$\left\{ \frac{\phi_n}{\phi_m} \right\} = \{\phi_g\}$$

Equation 13-131.

and single-point forces of constraint

$$\{q_s\} = [K_{fs}]^T \{\phi_f\}$$

Equation 13-132.

If PARAM,DYNPCF,NEW (default) then the single-point forces are computed as follows:

$$\{q_s\} = [[K_{sf}] - \omega^2[M_{sf}]]\{\phi_f\}$$

Equation 13-133.

In superelement analysis, if PARAM,FIXEDB,-1 is present, the uncoupled eigenvectors are recovered from the modified equations

$$\{\phi_a\} = \{\phi_{az}\}$$

Equation 13-134.

$$\{\phi_o\} = \{\phi_{oz}\}$$

Equation 13-135.

with $\{\phi_{az}\}$ and $\{\phi_{oz}\}$ from “**Superelement Analysis.**”

The multipoint forces of constraint qm_q are recovered by

$$qm_q = \begin{bmatrix} I_{mm} \\ -G_{mn}^T \end{bmatrix} (-\omega^2 M_{mg} + K_{mg})$$

Equation 13-136.

Dynamic Analysis

In the case of dynamic analysis with the direct formulation the extra points are partitioned out of $\{u_d\}$.

$$\begin{Bmatrix} u_a \\ u_e \end{Bmatrix} = \{u_d\}$$

Equation 13-137.

SDR1 recovers the dependent components

$$\{u_o\} = [G_{oa}]\{u_a\}$$

Equation 13-138.

$$\begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \{u_f\}$$

Equation 13-139.

and recovers single point forces of constraint

$$\{q_s\} = [K_{fs}]^T \{u_f\} - \{P_s\}$$

Equation 13-140.

If PARAM,DYNSPCF,NEW (default) then the forces are computed as follows.

In transient response analysis, the formulation is

$$q_s = M_{sf}^x \ddot{u}_f + B_{sf}^x \dot{u}_f + K_{sf}^x u_f - P_s$$

Equation 13-141.

where:

$$M_{Sf}^x = M_{Sf} + M^{2pp}_{Sf}$$

$$B_{Sf}^x = B_{sf}^x + B_{sf}^{2pp} + \frac{g}{\omega_3} K_{sf} + \frac{1}{\omega_4} K_{sf}^{4gg}$$

$$K_{Sf}^x = K_{Sf} + K^{2pp}_{Sf}$$

In frequency response and complex eigenvalue analysis, the formulation is

$$q_s = (-\omega^2 M_{sf}^x + i\omega B_{sf}^x + K_{sf}^x)u_f - P_s$$

Equation 13-142.

where:

$$M_{sf}^x = M_{sf} + M_{2pp_{sf}}^{2pp}$$

$$B_{sf}^x = B_{sf} + B_{2pp_{sf}}^{2pp}$$

$$K_{sf}^x = (1 + ig)K_{sf} + K_{2pp_{sf}}^{2pp} + iK_{4gg_{sf}}^{4gg}$$

The multipoint forces of constraint qm_g are recovered

$$qm_g = \begin{bmatrix} I_{mm} \\ -G_{mn}^T \end{bmatrix} (X_m - P_m)$$

Equation 13-143.

where qm_g is the reactions expanded to the g-set and X_m and P_m are the internal and external loads on the m-set degrees-of-freedom, respectively. (See also Eq. 9 in Section 3.5.1 in *The NASTRAN Theoretical Manual*).

In transient response analysis,

$$X_m = M_{mg}^x \ddot{u}_g + B_{mg}^x \dot{u}_g + K_{mg}^x u_g$$

Equation 13-144.

where:

$$M_{mg}^x = M_{mg} + M_{2pp_{mg}}^{2pp}$$

$$B_{mg}^x = B_{mg} + B_{mg}^{2pp} + \frac{g}{\omega_3} K_{mg} + \frac{1}{\omega_4} K_{mg}^{4gg}$$

$$K_{mg}^x = K_{mg} + K_{2pp_{mg}}^{2pp}$$

In frequency response and complex eigenvalue analysis,

$$X_m = (-\omega^2 M_{mg}^x + i\omega B_{mg}^x + K_{mg}^x)u_g$$

Equation 13-145.

where:

$$\begin{aligned}
 M_{mg}^x &= M_{mg} + M_{mg}^{2pp} \\
 B_{mg}^x &= B_{mg} + B_{mg}^{2pp} \\
 K_{mg}^x &= (1 + igK_{mg} + K_{mg}^{2pp} + iK_{mg}^{4gg})
 \end{aligned}$$

For frequency response there is one complex vector for each loading condition. For complex eigenvalue analysis there is one complex eigenvector for each complex root and there is no loading term $\{P_s\}$ associated with the determination of the single point constraints. For transient analysis there is a displacement, velocity, and acceleration associated with each output time step.

In the case of dynamic analysis with the modal formulation, you may obtain improved element stresses by requesting the mode acceleration data recovery technique.

The use of the mode acceleration technique requires the decomposition of the I -set stiffness matrix $[K_{II}]$. This decomposition is not otherwise performed in dynamic analysis unless the model has rigid body modes. Also, the technique requires a forward/backward substitution operation with a number of righthand sides equal to the number of solution vectors. A subset of the solution vectors can be obtained by using the OTIME or OFREQUENCY entries in the Case Control Section. The subset of solution vectors is prepared by the MODACC functional module.

In the case of dynamic analysis with the modal formulation, two methods of data recovery are available. The default procedure is usually much more efficient for data recovery operations when the modal formulation is used for dynamic analysis. In this procedure, the dependent components of the eigenvectors used in the modal formulation are first recovered by SDR1 using the same sequence of operations described for problems in real eigenvalue analysis (after eliminating unwanted modes). The complete eigenvectors are used as the input to SDR2 in order to determine the requested forces and stresses in terms of modal coordinates.

In dynamic analysis, you may restrict the output request to the response of selected points in the solution set. In this case, the time-consuming operations in the recovery of the dependent degrees-of-freedom can be avoided. This solution set processing is performed by VDR (Vector Data Recovery).

The modal data generated by SDR1 and SDR2 are formed into a data matrix by DDRMM (Dynamic Data Recovery Matrix Method). The output quantities are determined by multiplying the data matrix by the modal solution vectors. The computer time for this matrix multiplication is proportional to the number of output times, the number of modes used in the modal formulation, and the number of output frequencies or output time steps. If both the number of output items and the number of modes are small, the computer time required will not be significant when compared to the total problem solution time. Although the DDRMM method of data recovery is more efficient, a complete $\{u_g\}$ is not formed. For this reason the method does not permit the creation of deformed structure plots, grid point force balance or the use of the mode acceleration technique.

The alternate procedure may be invoked using PARAM,DDRMM,-1. With this method the solution vectors are first transformed from modal to physical coordinates

$$\{u_d\} = [\phi_{dh}]\{u_h\}$$

Equation 13-146.

This operation is performed by MPYAD in subDMAP MODACC. Having determined the vectors $\{u_d\}$ in physical coordinates, the data recovery operations can be completed using the same sequence of operations described for the direct formulation of dynamics problems.

Most of the operations in the recovery of the dependent components do not require significant amounts of computer time. However, in the case of the multiply-add operation associated with the recovery of the omitted coordinates ($[G_{oa}] \{u_a\}$, $[G_{oa}] \{\varphi_a\}$) a significant amount of computer time will be required if there is a large number of vectors and a large number of components in the a-set. This operation is likely to be especially costly for transient problems because there is usually a large number of output time steps and there are three vectors for each time step.

Data Recovery Operations in SubDMAP SEDRCVR

NX Nastran calculates the element forces and stresses in SDR2 (Stress Data Recovery – Phase 2). The matrix operations in this phase of data recovery are similar to those performed in EMG for stiffness matrix generation. The computer time will be proportional to the number of elements for which output is requested.

If static loads are applied to intermediate points on the CBAR element, you can request the output of stresses and forces at a selected set of intermediate points. The additional calculations required for output at intermediate points on CBAR elements is performed in functional module SDRX.

The solutions and data recovery operations for transient analysis are initially sorted by the time step (SORT1). A transpose operation is performed by SDR3 (Stress Data Recovery – Phase 3) in order to prepare the requested output sorted by external point identification number or external element identification number (SORT2). In frequency response problems, you may request either SORT1 or SORT2 output. The SORT2 output will usually be desirable if the solution is made for a large number of frequencies. In any event, the SORT2 output must be prepared if XY-plots are requested.

Computer time for the preparation of SORT2 output will be directly related to the number of items requested for output. If a large number of items are requested for output, the transpose operation will require a significant amount of computer time.

You may request the output of a grid point force balance table for a selected set of points. This table lists the forces acting at each selected point due to element forces, single point forces of constraints and applied loads. The grid point force balance table is prepared by GPFDR (Grid Point Force Data Recovery).

You may also request the output of an element strain energy table and/or an element kinetic energy table for a selected set of elements. The tables include the energy for each element, the percentage of the total energy for each element, and the energy density for each element. The element energy is computed from the equations

$$V_e = \frac{1}{2} \{u_e\}^T [K_{ee}] \{u_e\}$$

Equation 13-147.

$$V_e = \frac{1}{2} \{u_e\} [M_{ee}] \{u_e\}$$

Equation 13-148.

where the subscript e implies the data for one element. The energy density is computed by dividing the element energy by the element volume. Some elements do not have enough data input to allow calculation of element volume, such as CELASi elements. The density output for these elements is set to zero. Simplified equations for other element volumes may be used. For example, offsets are ignored for bar-like elements, and midside node locations are ignored for plate and solid elements. Element volumes are calculated by the ELPRT module, and can be output by use of PARAM,EST,1. The strain energy table is prepared by functional module GPFDR.

13.3 Summary of Solution Sequence Operations

This section describes the data processing and algorithms that are performed by special purpose functional modules, sequences of functional modules (subDMAPs) and complete solution sequences.

See also

- *NX Nastran DMAP Programmer's Guide (for detailed descriptions of functional modules)*
- Chapter 4 of the *NX Nastran Programmer's Manual*

Static Solutions

There are several major paths for static solutions and some minor ones. For standard static analysis the stiffness matrix is decomposed in SubDMAP SEKRRS, in preparation for a linear solution in SubDMAP STATRS. There is also a form of static analysis called inertia relief analysis.

There are now several paths through inertia relief analysis, reflecting that it is still in a state of development, with more changes likely in future systems. This revision of this manual will discuss inertia relief in SOL 101 only. Each path has advantages and limitations. There is another method in SOL 144, Static Aeroelasticity. It is documented elsewhere.

All methods of inertia relief analysis take the standard static analysis equation

$$K_{aa} \cdot u_a = P_a$$

Equation 13-149.

where P_a are the applied loads, and add a set of constraint equations R_{ra} and automatically-computed inertial loads P_i . The constraints are required because K_{aa} is for a free structure and cannot be solved reliably without some form of constraint.

$$K_{aa} \cdot u_a = P_a - P_i = P_{ai}$$

Equation 13-150.

$$R_{ra} \cdot u_a = 0$$

Equation 13-151.

In theory, if P_{ai} is a set of balanced loads (no resultant) the solution of u_a is possible without the need for constraint equations. In practice, P_{ai} may have some unbalance due to numerical truncation effects. Even a small resultant load will provide a displacement solution at infinity in theory, or with very large numbers in practice, usually. The same truncation effects that unbalance the loads may provide enough “ground springs” in the stiffness matrix to allow a solution, but it will be a poor quality solution. Everything will first move 1.E5 units in the same direction. Stresses will be determined by subtracting numbers of this size to find element displacement perhaps on the order of 1.E-3. Ironically, if all goes well the forces in the support system are miniscule, but it must be present to control stability when truncation errors are present.

Types of Inertia Relief

Inertia Relief analysis is a quasi-dynamic analysis. It is assumed that the structure is not tied to ground and is in a state of steady acceleration caused by the applied loads. The displacements measured relative to ground are constantly increasing and cannot be determined. Displacements are therefore determined with respect to an accelerating reference frame that is attached to the structure. The applied loads are balanced by inertial loads that are calculated automatically from the steady acceleration acting on the mass properties of the model. Inertia relief analysis is requested with PARAM,INREL,[x], where [x] may have the values -1 or -2.

The reference frame may be obtained by two different methods.

- For the manual support method (PARAM,INREL,-1) you select 6 or fewer DOFs using SUPORTi entries. The reference frame is defined from these DOFs.
- For the auto support method (PARAM,INREL,-2) 6 constraint equations are written that constrain the structure to zero motion, on average, while leaving it free to “breathe” and deflect due to the applied loads and inertial loads.

Both methods find it convenient to have “rigid body modes shapes” that represent a set of vectors that represent all possible strain-free motion of the structure when not loaded. The technique used to obtain rigid body mode shapes differs between the two methods.

Rigid Body Modes Using Manual Support (PARAM,INREL,-1 and SUPORTi)

You prescribe a set of DOFs that constrain the structure in a statically-determinate manner on SUPORTi entries. These entries define the *r*-set. There must be six or less DOFs listed and they must constrain all rigid body modes that are not constrained by SPCs or other modeling devices. The rigid body modes are computed from the static analysis equation. There are no loads applied and I_{rr} is the identity matrix. Given

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{lr}^T & K_{rr} \end{bmatrix} \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix} = \begin{bmatrix} O_{lr} \\ O_{rr} \end{bmatrix}$$

Equation 13-152.

D_{lr} is found by solving the first row of this equation, then merged with I_{rr} to form D_{ar} .

$$K_{ll} \cdot D_{lr} \leq -K_{lr} \cdot I_{rr}$$

Equation 13-153.

$$[D_{ar}] = \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix}$$

Equation 13-154.

Rigid Body Modes Using Auto-Support (PARAM,INREL,-2)

Vectors of rigid body motion D_{gr} for six unit motions of a reference point are computed from grid point geometry in module VECPLOT. The D_{ar} component is partitioned from this matrix.

The Basic Equation of Motion for Inertia Relief

Inertia relief analysis starts with the a-set equation

$$K_{aa} \cdot u_a \leq P_{ai}$$

Equation 13-155.

and adds a constraint equation

$$R_{ar}^T \cdot u_a = 0$$

Equation 13-156.

The last equation implies a similar equation on a generalized constraint force variable vector, q_r

$$R_{ar} \cdot q_r = P_{ai} - K_{aa} \cdot u_a$$

Equation 13-157.

If P_{ai} represents balanced loads, q_r is zero. If there is a small unbalance due to numerical truncation effects, q_r is then small, but finite. There are two methods for defining R_{ar} , as discussed below. Both have the property of defining some part of u_a to have zero motion, so that the other parts of u_a are in effect measured relative to the constrained DOFs.

On of the solution methods partitions K_{aa} into its r -set and l -set components. The first equation is partitioned, and the a -set equations and the constraint equation may combine into one equation,

$$\begin{bmatrix} K_{ll} & K_{lr} & R_{rl}^T \\ K_{lr}^T & K_{rr} & R_{rr}^T \\ R_{rl} & R_{rr} & O_{rr} \end{bmatrix} \cdot \begin{bmatrix} u_l \\ u_r \\ -q_r \end{bmatrix} = \begin{bmatrix} P_{ail} \\ P_{air} \\ Y_r \end{bmatrix}$$

Equation 13-158.

The methods of computing P_r are described below. Y_r , the generalized displacement input, is set to zero.

Constraint Equations Using Manual Supports

In the constraint equation $R_{rr} = I_{rr}$, the identity matrix. R_{rl} is null. The last row can then be solved for $u_r = Y_r = 0$. The second equation can then be used to eliminate q_r ,

$$K_{lr}^T \cdot u_l - I_{rr} \cdot q_r = P_{air}$$

Equation 13-159.

$$q_r = K_{lr}^T \cdot u_l - P_{air}$$

Equation 13-160.

Because R_{rl} is null, the first equation then becomes

$$K_{ll} \cdot u_l = P_{ail}$$

Equation 13-161.

This equation can be solved for u_l reduced from u_a directly. It will be shown later that P_r terms are transferred to P_l terms in the inertial loads. u_a is formed from a merge of u_l and null terms for u_r .

Constraint Equations Using Automatic Supports

A constraint equation is written that states that the average weighted motion of the system is zero,

$$R_{ra} = D_{ar}^T \cdot M_{aa}$$

Equation 13-162.

$$R_{ra} \cdot u_{ar} = 0$$

Equation 13-163.

R_{ra} is generally a dense matrix, with non-zero terms at most DOFs with mass. It could be used to reduce the K_{aa} matrix with equations similar to those used for multi-point constraints, but that would produce a dense K_{ll} matrix, much more expensive to decompose and solve than the K_{ll} used in the manual support method. A solution method is used which increases solution costs only slightly.

The r -set l -set partitioning is not needed. The constraint equation is appended to the a -set equations to define the l -set equations in their augmented form,

$$K_{ll} \cdot u_l = P_l$$

Equation 13-164.

$$\begin{bmatrix} K_{aa} & R_{ra}^T \\ R_{ra} & 0_{rr} \end{bmatrix} \begin{bmatrix} u_a \\ -q_r \end{bmatrix} = \begin{bmatrix} P_{al} \\ 0_r \end{bmatrix}$$

Equation 13-165.

q_r are the unknown forces required on the system to produce equilibrium. They should be computational zeros, and are output with the SPC forces. The equation involving the augmented K_{ll} is solved directly for $u_l \cdot u_a$ and q_r are then partitioned from u_l . In superelement analysis the constraints are applied in the residual structure only.

Matrix Decomposition in SubDMAP SEKRSS

There are two major branches through this SubDMAP, standard static analysis and static analysis with inertia relief effects.

Inertia Relief Not Selected

When inertia relief is not selected and there are no SUPORTi entries present the a -set is decomposed directly in the DCMP module,

$$K_{aa} = L_{aa} \cdot D_{aa} \cdot L_{aa}^T$$

Equation 13-166.

L_{aa} is lower triangular, and D_{aa} is diagonal. Both are stored in an unconventional (“packed”) format, and output in a single data block named LLL. This name is used because the l -set and the a -set are synonymous when no SUPORTi entries exist. The factor matrices L_{aa} and D_{aa} are passed on to SubDMAP STATRS for solution of u_a .

When inertia relief is not selected but there are SUPORTi entries present, the a -set equations are partitioned into the r -set, defined on the SUPORTi entries, and the l -set, what is “left over” after the r -set is removed.

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{bmatrix} u_l \\ u_r \end{bmatrix} \geq \begin{bmatrix} P_l \\ P_r \end{bmatrix}$$

Equation 13-167.

In this context, the r -set is treated as an alternate form of single-point constraints. The partitions of K involving r are discarded. u_r is set to zero. u_l is found from the first row of this equation. Loads in P_r go directly to ground, and are output with the SPC forces, along with the product $-K_{rl} \cdot u_l$ from the second equation. K_{ll} is decomposed with the same equations used for conventional static analysis.

Decomposition of K_{ll}

For the manual support method K_{ll} is defined above. Its factors, used to compute the rigid body modes are also used later to compute the static solution. For the auto support method the factors of the expanded form of K_{ll} are passed on.

Computation of Inertial Loads

The basic assumption of inertia relief is that the structure is in a state of steady acceleration and that all transients due to dynamic effects are negligible. The solution is obtained in an accelerating reference frame from which all displacements are measured. Inertial loads are automatically computed that exactly balance the applied loads. A support system is applied to make the system numerically stable. The inertial loads assume rigid body accelerations when computing the loads.

The derivations given here assume that the system mass matrix M_{gg} has been reduced to M_{aa} and is available, as described elsewhere. In the actual DMAP, equivalent methods that never generate M_{aa} are used instead, in the interests of efficiency. Unit inertial loads are computed from M_{gg} , then reduced by load reduction techniques, avoiding the need for $M_{aa} \cdot M_{aa}$ can be a large size data block, and costly to compute. It can be shown that the results provided by the two derivations are identical. For example, the rigid body mass matrix M_{rr} , defined below as

$$M_{rr} = D_{ar}^T \cdot M_{aa} \cdot D_{ar}$$

Equation 13-168.

can just as easily be formed from the g -set equations

$$M_{rr} = D_{gr}^T \cdot M_{gg} \cdot D_{gr}$$

Equation 13-169.

D_{ar} is used to compute the rigid body mass M_{rr} about the reference point.

$$M_{rr} = D_{ar}^T \cdot M_{aa} \cdot D_{ar}$$

Equation 13-170.

M_{rr} is printed out with the matrix name of QRR.

The acceleration of the reference point a_r due to the applied loads is computed,

$$M_{rr} \cdot a_r = D_{ar}^T \cdot P_a$$

Equation 13-171.

The acceleration a_r are printed with the matrix name of URA.

The resulting inertial loads P_{ai} and total loads P_{ai} are

$$P_i = M_{aa} \cdot a_r$$

Equation 13-172.

$$P_{ai} = P_a - P_i$$

Equation 13-173.

For both methods, the mass matrix is reduced from M_{gg} size to M_{rr} size with load vector operations that can be shown to be identical to the M_{aa} operations described here, except for the order of operations. The load and mass transformations are based on rigid body modes produced from geometry.

A Comparison of Auto- and Manual-Support Solutions

When the same model is analyzed with both methods you should find that stresses, constraint forces, and other internal quantities are identical except for truncation errors, but the displacements may differ appreciably. This is because they use a different reference frame. For the manual supports, the reference frame is set up so that the motion at the *r*-set DOFs are identically zero. Think of a reference frame drawn through these points, and all displacements measured relative to it. This reference frame is moving through space at a constant acceleration rate, so that displacements are relative rather than inertial displacements. There are many sets of SUPORT *i* entries valid for one model. If you change SUPORT *i* DOFs you may also expect to see changes in displacements, but not in internal loads or element stresses.

For the auto-support option, one can visualize the reference frame as a set of axes connected to every point with mass with a very complicated set of levers that allow the structure to “breathe” as loads are applied to it, but keep the weighted average motion to a zero value. If the model happens to have a grid point at the reference point, you will observe very small motion there, computational zeros but not the binary zeros of the manual support method. This is because this motion is computed for the auto-support method, but assumed to be zero for the manual method.

The choice of an *r*-set as defined by SUPORT *i* DOFs requires meeting both a geometric requirement and a stiffness requirement. This geometric requirement is that the set should form a statically determinate tie to ground. This can be difficult to do with complicated geometries. The stiffness requirement is that the DOFs should be stiff enough to provide a stable solution. A method that has been used by some modelers is to pick points stiff points do not exist. The difficulties in picking a stable, reliable support system was one of the main motivations for development of the more modern auto-support system. SUPORT *i* entries are not allowed for the auto support method, and will cause a fatal error exit when present.

PARAM,GRDPNT,GID is optional for both methods when superelements are not present. It is required when superelements are present, and must be on a boundary point attached to all superelements. This same point should be used for the SUPORT *i* DOFs for the manual support method. It defines the reference point used for generating the geometry-based rigid body vectors. The default for GRDPNT is GID=-1, which is a flag to use the basic origin as the reference point when computing rigid body modes. Use of a GID near the c.g. of the structure produces more esthetically pleasing results. The rigid body accelerations produced by the solution are then measured near the center of the structure, rather than at a point that may be outside the structure.

Applications of Inertia Relief

Although this capability could be used in an environment where the reference frame is accelerating steadily, such as a free-falling elevator, the more common usage is when the modeler attempts to apply loads in equilibrium. He wishes the net acceleration of the structure to be zero. A ship floating in the sea, with pressure loads balancing gravity loads, is an example. Developing balanced load sets can be tedious on large models with complex geometry. The accelerations printed out in the URA matrix can be used as a measure of load quality. If the accelerations are negligibly small, the load balance is adequate. If the accelerations are large, some adjustment of the loads sets may be necessary. For good inertia relief analysis a good mass distribution is needed. As the structural weight is usually only a small proportion of the total weight, this means that a realistic payload mass distribution must be modeled also.

Another use of inertia relief analysis becoming more prevalent with the advent of auto support is in checkout of a new model. Although six and only six rigid body modes are provided, they are provided

in the form of a “flying waterbed”. To take an extreme example, suppose that a structure is made up of two components bolted together, but the bolts were not modeled, inadvertently. This model has twelve rigid body modes before the constraints are applied. Both components sit on the same waterbed. This means that all singularities will be constrained, and the model will run to completion on the first pass. If loads are applied to one component only, the constraints will cause the other component to move too, in a strain-free manner. A plot of the motion of the model should reveal that there is no displacement continuity between the two components. This should make the cause of the unexpected behavior apparent. As the goal of this analysis is model checkout only the structure mass distribution is usually adequate for stable solution. The payload masses may be added later.

Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS

The basic equation solved in these SubDMAPs is

$$[K - \lambda \cdot M] \cdot \phi = 0$$

Equation 13-174.

where K and M are real and symmetric, and M is positive semi-definite. This equation is defined for the a-set variables when called from MODERS for overall structural modes of vibration, and for o-set variables when called from SEMR3 during CMS (Component Mode Synthesis) analysis. Many steps are taken in DMAP to better automate the removal of DOFs that are ill-defined, and to augment the mode shapes with residual flexibility effects. The SubDMAPs that perform these operations are described below.

Although a knowledge of DMAP is not necessary for dynamic analysis, such a study can increase the understanding of the steps that are performed.

Massless Mechanism Identification in SubDMAP MMFIND

SubDMAP MMFIND identifies potential instabilities in the matrices used for real eigensolutions. The K and M matrices presented to the READ module for real eigensolutions may contain massless mechanisms (MMs). A massless mechanism has a constraint shape X such that

$$X^T \cdot K \cdot X = \varepsilon_k$$

Equation 13-175.

$$X^T \cdot M \cdot X = \varepsilon_m$$

Equation 13-176.

$$X^T \cdot X > 0.0$$

Equation 13-177.

If ϕ contains the eigenvectors of the system, then

$$X^T \cdot \phi = 0$$

Equation 13-178.

$$X^T \cdot M \cdot \phi = 0$$

Equation 13-179.

The constraint shapes have non-zero terms only where the eigenvectors have zero terms. The massless mechanism shapes can be thought of as "deficient" eigenvectors describing parts of the system that are unconnected (or very weakly connected) to other parts of the structure that have both stiffness and mass that are active in the well-conditioned eigenvectors. k and m may be binary zeroes, or computational zeroes due to truncation effects. An "eigenvalue" λ associated with massless mechanism shapes is

$$\lambda = \varepsilon_k / \varepsilon_m$$

Equation 13-180.

λ is indeterminate in the limit as the i approach zero. It can be a very large number, a very small number, and either negative or positive. It is likely to change a large amount when small changes are made to the model. This is the definition of an unstable eigensolution. The assumption taken here is that the cause of this condition is either stiffness and/or mass effects inadvertently left out of this part of the model, with the result that the load paths are ill-defined. This load path is constrained, and the software issues a warning message. That is, this mode of motion is later removed from the K and M matrices prior to eigensolution by steps equivalent to single point constraint elimination.

The massless mechanisms are defined from the shifted stiffness matrix $A = K + s \cdot M$, where s is a shift that you may change from its default value with the parameter PARAM, SHIFT1, (default=1.234 Hz.). The default is chosen to make detection of rigid body modes as MMs unlikely. A is decomposed into its factors,

$$A = L \cdot D \cdot L$$

Equation 13-181.

The ratios of the diagonal terms of A over D are stored in a vector Ry. Terms of D approach a value of zero at the ends of massless mechanisms. Terms of Ry larger in magnitude than user parameter MAXRATIO are reset to unity and are kept in vector R1. It can be shown that each of these terms is at the end of a massless mechanism constraint shape. R1 is expanded into a set of loading vectors P , with one unit term per column. P is used along with the factors to find the constraint shapes,

$$A \cdot u \leq P$$

Equation 13-182.

The constraint shapes are normalized, and small numbers are discarded. Each column of u is then a potential constraint mode. It could also be a rigid body mode. The rigid body modes are detected by computing the diagonal terms of the generalized mass matrix for these shapes,

$$Mg = u \cdot M \cdot u$$

Equation 13-183.

Terms of $Mg(i, i)$ greater than PARAM, MECHFIL, default value = 1.e-6 are an indication that the corresponding column is a rigid body mode. Columns of this type in u are discarded to produce $u1$, the matrix of constraint mode shapes. The location of the end of the constraint shapes is passed back in R . The constraint mode shapes themselves are passed back in MECHU. R is used later to remove rows and columns in K and M to constrain the massless mechanisms. The nonzero terms in MECHU are used to indicate which DOFs will discard loads applied to them, because they will have zero coefficients in the true eigenvectors of the system, leading to a null contribution to generalized force for all modes. MECHLD contains P , and is intended for a feature not yet implemented.

Massless Mechanism Control in SubDMAP XREAD

At present, the auto-omit DMAP is duplicated at several places, followed by a call to XREAD. It is therefore an implicit part of the XREAD operations now, and may be moved to XREAD in a later version to reduce the duplication of code.

SubDMAP XREAD modifies the input matrices used for eigensolution, when needed, then restores the modified solutions back to the original size of the matrices. Solutions for both structural vibration and buckling analyses are provided. This discussion describes vibration analysis first, with a concluding section on changes made for a buckling solution.

Auto-Omit Operation

All the eigensolution methods require that each column of $[K]$ plus $[M]$ have some nonnull terms. Any degree-of-freedom that has only null terms in both matrices is regarded as undefined and causes a fatal error. The HOU method of eigenvalue extraction also requires that the mass matrix be positive-definite. The MHO and the Lanczos methods allow null columns in the mass matrix but are generally faster if the massless degrees-of-freedom are removed by the static reduction method described in “**Static Solutions in SubDMAP SEKRRS.**” However, the static reduction tends to increase the number of active columns in the a-set matrices. The SIN and Lanczos methods, by contrast, are generally slower if the static reduction is performed on massless degrees-of-freedom, because these methods take advantage of sparse matrices, whereas the other methods do not.

Three options are provided to deal with the various null column possibilities. If PARAM,ASING,-1 is used, any degrees-of-freedom with null columns in both the stiffness and mass matrices (“undefined degrees-of-freedom”) are identified and a fatal error exit is taken. This is an option to use to identify modeling errors before performing an expensive eigenvalue solution.

If ASING = 0 (the default) the method used depends on the eigenvalue extraction method. In the tridiagonal methods of eigenvalue extraction, degrees-of-freedom with null columns in the mass matrix are identified and placed in the w-set.

$$[M] = \begin{bmatrix} M_{xx} & | & M_{xw} \\ \hline M_{wx} & | & M_{ww} \end{bmatrix}$$

Equation 13-184.

where $[M_{ww}]$ and $[M_{wx}]$ are null by definition. Degrees-of-freedom in the r-set (SUPPORT Bulk Data entry) must have mass, or a fatal error will result. The stiffness matrix is similarly partitioned.

$$[K] = \begin{bmatrix} \bar{K}_{xx} & | & K_{xw} \\ \hline K_{wx} & | & K_{ww} \end{bmatrix}$$

Equation 13-185.

with unit diagonal terms placed on null columns of $[K_{ww}]$. The static condensation matrices are formed

$$[K_{ww}][G_{wx}] = -[K_{wx}]$$

Equation 13-186.

and

$$[K_{xx}] = [\bar{K}_{xx}] + [K_{wx}]^T [G_{wx}]$$

Equation 13-187.

This method will prevent errors due to mass matrix singularity in the tridiagonal methods if the singularity is caused by null columns in $[M]$. It will not detect singularities due to mechanisms (caused, for example, by a point mass offset from a grid point). This type of singularity does not necessarily prevent obtaining eigensolutions, but will cause poor numerical stability. The MHOU method does not suffer from numerical instability due to singular or nearly singular mass matrices. They are therefore more reliable than the HOU method. In well conditioned problems, however, it is unnecessary to ? automatically select the appropriate method. The AHOU method is the most modern, reliable method.

The static reduction will reduce the cost of solution and minimize the number of uninteresting high roots found at the value of computed infinity due to the singularity of the mass matrix. Note that the static reduction used here does not introduce any approximation, as the "omitted" that is w-set terms have null mass values. The introduction of unit diagonal terms on null columns of $[K_{ww}]$ does not introduce approximations because the terms that multiply these diagonal terms (the corresponding rows of $[G_{wx}]$ are null. The same results can be achieved at higher cost by first eliminating null rows

and columns of $[K_{ww}]$ by the single point constraint operations of “Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB” followed by static reduction.

The solution for the system $[K_{xx} - \lambda \cdot M_{xx}] \cdot \phi_x = 0$ is provided. Rigid body shapes D_{xr} are provided when SUPORTi entries exist, causing an r-set. The factor of K_{xx} , named LLL, is used for computing approximate solutions for buckling analysis, to better set the range of eigenvalues to be searched. For the tridiagonal methods, K and M have DOFs with zero mass removed by an auto-omit method in a prior subDMAP. For other methods of analysis, K_{aa} and M_{aa} are used directly for residual structure modes, and K_{vv} and M_{vv} for CMS (Component Mode Synthesis) modes.

The modes of the system are computed in the READ eigensolver module. If the solution appears to be of good quality the subDMAP returns with no other operations performed. If the modes can not be found, or appear to be of low quality, the module sets a flag (Neigv parameter). This flag causes the MMFIND subDMAP to be called. Given K and M , it finds a high-ratio vector R and a matrix of constraint mode shapes MMECHA. These terms are defined in the MMFIND section above.

There are non-zero terms in R at the end of massless mechanisms. These mechanisms can be removed with SPC-like operations using R as a partitioning vector,

$$K_{xx} = \begin{bmatrix} K22 & K21 \\ K12 & K11 \end{bmatrix}$$

Equation 13-188.

M_{xx} is partitioned similarly, to obtain $M22$. Partitions involving the "1" subscript are discarded. The modified eigenproblem solved is then $[K22 - \lambda \cdot M22] \cdot \phi_2 = 0$. ϕ_2 is merged to the size of the original K matrix by inserting null rows for the "1-set" DOFs,

$$\phi_x = \begin{bmatrix} \phi_2 \\ 01 \end{bmatrix}$$

Equation 13-189.

The auto-omit reduction is backed out by the operations

$$\phi_w = G_{wx} \cdot \phi_x$$

Equation 13-190.

$$\phi = \begin{bmatrix} \phi_x \\ \phi_w \end{bmatrix}$$

Equation 13-191.

Real Eigenvalue Analysis in the READ Module

Real eigenvalues and their associated eigenvectors are determined in READ (Real Eigenvalue Analysis – Displacement). In the case of normal mode analysis, READ extracts real eigenvalues from the equation

$$[K - \lambda M]\{u\} = 0$$

Equation 13-192.

There are seven basic eigensolution methods. The Inverse (SINV) Method obtains eigensolutions by iterations based on the equation

$$[K - \lambda_i M]\{\phi_{i+1}\} = [M][\phi_i]$$

Equation 13-193.

where λ_i is an estimate of the eigenvalue. This method is best suited to large problems with sparse matrices, where only a few eigenvectors are desired.

The tridiagonal methods, Householder (HOU and MHOU), first mass-scale the equation

$$[M] = [L][L]^T$$

Equation 13-194.

where $[L]$ is a lower-triangular matrix. The $[K]$ and $[M]$ matrices are pre- and post-multiplied by $[L]^{-1}$ and $[L^{-1}]^T$, respectively, to obtain the standard form

$$[J - \lambda I]\{\bar{\phi}\} = 0$$

Equation 13-195.

where $[J] = [L^{-1}][K][L^{-1}]^T$ and $[I]$ is the identity matrix. Note that the mass matrix is in effect inverted, so that it must be nonsingular. The $[J]$ matrix is transformed to tridiagonal form for economical computation of eigenvalues and eigenvectors. The eigenvectors in the tridiagonal basis are then back-transformed to the initial basis. The Modified Householder (MHOU) Method uses the shifted matrix

$$[K + \lambda_s M]$$

Equation 13-196.

in place of $[M]$ in the discussion above, thereby allowing the mass matrix to be singular if there are compensating terms in the stiffness matrix. The MHOU method is best suited to small problems, or for

large problems where many eigenvectors are required, after extensive static reduction or dynamic reduction is performed.

The Lanczos method uses a block shifted algorithm. Sets of vectors obtained by a recursive form are used to reduce the problem to a reduced block tridiagonal form. The eigensolutions are computed in the reduced basis by a QL algorithm, then back-transformed to the original basis. This is currently the most modern method, and should be considered for all large-size problems. Since the method takes advantage of sparsity in the input matrices, it is most economical when used without static or dynamic reduction. It is requested with either the EIGRL or EIGR Bulk Data entries.

The AHOU method combines the best features of all of the tridiagonal methods, and is the recommended method at present. It first decomposes M . If the factors are stable, the HOU method is used. If the factors may be unstable, the MHOU method is used instead.

Modal Solution Preparation in the MODERS subDMAP

MODERS is called when solving for the eigensolution of the residual structure, with a-set operations. It is used for both structural mode and buckling analysis.

In buckling, the r-set caused by SUPORTi entries are treated as equivalent SPCs. The rows and columns associated with r-set DOFs are discarded before eigensolution.

For structural analysis, the auto-omit operations take place for the tridiagonal methods. See the XREAD Section for details.

Table indexing is a method to allow direct access to data in large sequential files, to speed up interactive analysis. This work is being done as part of the API project.

Residual Flexibility in the SubDMAP RESVEC

Residual flexibility augments the mode shapes with shape functions found from static analysis with various types of loading functions. These auxiliary displacement shapes reduce errors due to modal truncation, particularly near the area of load application. These shapes are modified or discarded to make them linearly independent of the mode shapes. They are then made orthogonal to the mode shapes with a second eigensolution in a reduced basis. This augmented system provides increased accuracy for modal dynamic analysis.

The RESVEC subDMAP is called in several contexts. When used in the residual structure, it operates on a-set matrices such as K_{aa} . When used in CMS, it operates on o-set matrices such as K_{oo} . The naming convention used in the subDMAP is to use the generic set name x . For example, K_{xx} implies K_{aa} when called from the residual structure operations, and K_{oo} during CMS. In this section we will use the a-set nomenclature, to make the discussion less abstract. The notation "In CMS" below implies that we are then talking about operations unique to CMS-only in that paragraph, that is, o-set operations. Other operations for CMS operations are similar to those for the a-set.

As in other subDMAPs, fluid-structure analysis of modal data is broken into two parts, a part for structural modes only, and a part for fluid modes only. These effects are coupled after RESVEC. A review of the subDMAP compilation shows the complication resulting from this partitioning and the looping through several operations, once for fluid and once for structure. The complexity is ignored in this section by largely ignoring fluid analysis. The equations describe the structural path. The fluid path can be inferred to be similar.

Finally, some of the steps described in this section are actually performed in other SubDMAPs. They are described here for clarity, with a notation of the SubDMAP where the actual DMAP statements may be found.

Parameters Unique To Residual Flexibility Operations

The comment section of the DMAP source code is reproduced here, with extensions such as the default value of NDDL parameters. The recommended parameters to use are param, resvec, yes, and param, resviner, yes. The first is required to select any type of residual vector calculation. The second adds load cases for loads due to inertia effects from rigid body acceleration to the applied load cases. It has been found to improve results for a small incremental cost.

```
$ resvec default='no'
$ The main user parameter. Selects all resvec operations, as qualified
$ by the other resv* parameters. Set to 'yes' for any resvec
$ operations.
```

Tuning parameters are used only for unusual models. The defaults were set after running many test problems.

```
$ resvrat default = 1.e8
$ filter value for discarding trial vectors with little
$ independent content. Make it smaller to discard more vectors
$ resvpgf default = 1.e-6
$ filter on partitioning vector. Make it larger to discard more vectors
$ resvofxs default = .001
$ [in the DMAP, but no longer in use]
```

Intermediate Output Control

```
$ resvse default='no'
$ The default 'NO' will skip the printing of the strain
$ energy of the static shapes (labeled EXTERNAL WORK).
$ Use RESVSE='YES' to print the strain energy.
```

Miscellaneous

```
$ resvsli default = 'yes'
$ The default 'YES' will test for linearly
$ independent shapes. Use RESVSLI='NO' to skip the tests.
$ resviner default = 'no'
$ The default (NO) is to NOT use inertial loads to obtain trial vectors.
$ Use 'yes' to use inertial loads as a source for
$ trial vectors
$ resvso default = 'yes'
$ The default 'YES' will clean up the static vectors to
$ insure orthogonality to modes. If the stiffness
$ matrix is well conditioned this is unnecessary, use
$ RESVSO='NO' to skip the reorthogonalization.
$ resmeth default = 'ahou'
$ method used for eigensolution in reduced basis.
$ 'lan' is an alternate method, somewhat more robust but slower for this
$ class of reduced basis problem.
```

The research legacy of this capability is evident in the many user parameters for specific functions, some of which appear only in TYPE statements. Most are used only when you suspect problems in your analysis. Their defaults, and even names, are likely to change from version to version. The res*-type parameters current with the version you are using can be determined by compiling the RESVEC SubDMAP, and searching for, y, parameter names in TYPE statements. The default values of NDDL parameters can be determined compiling the NDDL statements with an input file like this:

```

$ file nddl*.dat
compile nddl=nddl nolist noref deck $
cend
begin bulk
enddata

```

The .pch file output format, provided when "deck" appears on the compile statement, is more readable with an editor than the .f06 output file. An example of some lines of this output is:

```

PARAM RESVEC  = 'NO'          TYPE=CHAR8  PATH=SEID  LOCATION=MASTER
PARAM RESVSE  = 'NO'          TYPE=CHAR8  PATH=SEID  LOCATION=MASTER
PARAM RESVSLI = 'YES'         TYPE=CHAR8  PATH=SEID  LOCATION=MASTER
PARAM RESVINER= 'NO'          TYPE=CHAR8  PATH=SEID  LOCATION=MASTER
PARAM RESVSO  = 'YES'         TYPE=CHAR8  PATH=SEID  LOCATION=MASTER
PARAM RESVRAT = 1.E+8         TYPE=RS      PATH=SEID  LOCATION=MASTER
PARAM RESVPGF = 1.E-6         TYPE=RS      PATH=SEID  LOCATION=MASTER

```

Load Generation

Three types of loads are used to generate auxiliary shapes. Their selection is controlled by user parameters and other bulk data input. The parameter name and its default are listed by each load type. Adding new load columns has little incremental cost, so provisions are made for many types of loading conditions, to avoid the need for re-calculation of residual flexibility effects on restarts. The types of loads chosen are those most likely to contain effects that may be truncated by modal analysis.

- Point Loads (USET,U6 Bulk Data entry; PARAM, RESVEC,YES)

This is a provision for cases where the DOFs to be loaded in later runs are known *a priori*, but the magnitudes of the loads are not known. This type of loading is described not by conventional load bulk data entries, but by an entry used to define a special set, the u6 set. This option is best suited to loads on a few DOFs. This set is not used for other purposes in the solution sequences, so it is used in this context to define places where unit point loads will be applied later. Each u6 DOF causes one additional loading condition consisting of a point load at DOF u6. These loads are labeled P_{g1} in the subDMAP.

- Applied Loads (Conventional dynamic loading entries; PARAM, RESVEC,YES)

All dynamic loads defined on DAREA Bulk Data entries are converted to equivalent FORCE entries at the start of the program. All loads selected from a DLOAD Case Control command that apply to grid and scalar points are processed for static loads. These load entries are processed by the static load generator (SSG1 module) to produce the P_g load vectors. These are the spatial loads that will be multiplied later by time-varying functions in transient analysis, or frequency-varying functions in frequency response analysis.

Appended, Reduced Loads

Loads from the prior two possible sources are appended into one load matrix,

$$P_{g2} = \left[P_{g1} \mid P_g \right]$$

Equation 13-197.

The loads are partitioned and reduced,

$$P_{g2} = \begin{bmatrix} P_o \\ P_{ax} \end{bmatrix}$$

Equation 13-198.

$$P_a = R_{ga}^T \cdot P_{g2}$$

Equation 13-199.

where R_{ga} symbolizes the several-step load reduction described for static loads earlier. P_a is carried forward when processing residual structure modes. P_o is carried forward when processing CMS modes. They will be given the generic name P_a in the remaining discussion.

Inertia loads are due to unit rigid body acceleration (param,RESVNER, YES; PARAM, RESVEC,YES).

Rigid body displacements D_{gr} for 6 unit motions of a reference point are computed from geometry. The a-set DOFs are partitioned out to form D_{ar} . The inertia loads due to unit accelerations of these shapes are computed,

$$P_{aa} = M_{aa} \cdot D_{ar}$$

Equation 13-200.

These loads are appended to all prior selected loads to make the final load matrix P_{a1} , consisting of u6-set effects, applied load effects, and rigid body inertia load effects.

Free Boundary Effects (CMS only)

When free boundary points are defined by CSETi entries, the mode shapes are computed for the v-set, which includes the o-set and the c-set. It is necessary to reduce the v-set effects to o-set effects. The v-set may include rigid body modes that the c-set constrains. The derivation and motivation for these steps is described elsewhere. The resulting steps implemented in the subDMAP are:

- Discard any rigid body modes by counting the number of r-set points.
- Reduce the mode shapes from v-set to o-set size,

$$P_{o2} = P_{o1} - G_{ox} \cdot \phi_x$$

Equation 13-201.

where x symbolizes the union of the c- and r-sets, locally.

P_{o2} replaces P_{a1} in the ensuing discussion.

Load Sweeping

Later steps will discard shape functions that are not linearly independent of the mode shapes and prior shape functions. Some of these shapes that would be discarded can be identified by inspecting the loads that produce them. These loads can then be discarded before their residual vectors are calculated, saving the expense of calculating shapes that will eventually be discarded.

Loads in P_{a1} are normalized to produce P_{an} . Some portion of the loads in P_{an} can be expressed as modal loads, and therefore do not need to be carried further. The derivation of the sweeping equation used to eliminate these loads starts by expressing the loads as a linear combination of the inertial modal loads

$$P = M_{aa} \cdot \phi_a \cdot y$$

Equation 13-202.

where y is unknown. Multiplying both sides by ϕ_a^T , and solving for y ,

$$y = M_i^{-1} \cdot \phi_a^T \cdot P$$

Equation 13-203.

where $M_i^{-1} = [\phi_a^T \cdot M_{aa} \cdot \phi_a]^{-1}$, the inverse of the modal mass.

These equations are used to solve for the swept loads P_{a2} , based on the normalized loads P_{an} ,

$$P_{a2} = P_{an} - M_{aa} \cdot \phi_{a2} \cdot M_i^{-1} \cdot \phi_{a2}^T \cdot P_{an}$$

Equation 13-204.

If the modes adequately express the generalized load caused by a column of P_{an} , the corresponding column of P_{a2} will be numerical noise, consisting of numbers near computational zero. Your input used to set this filter is PARAM,RESVPGF, with a default value of 1.E-6. Columns with little content are discarded from P_{a2} , leading to the swept load vectors P_{af} .

Static Solution for Shape Functions

If the structure contains rigid body modes it is not stable for static analysis. This problem is solved by one of two methods:

Manual SUPORT (SubDMAP SEKRSS)

If you supply SUPORTi entries, the r-set DOFs it produces are used as "temporary" SPCs, applied to allow solving for the static shapes, but removed before computing modes. Partitions involving "r" are discarded.

$$K_{aa} \geq \begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix}$$

Equation 13-205.

$$P_{a2} \geq \begin{bmatrix} P_{l2} \\ P_{r2} \end{bmatrix}$$

Equation 13-206.

$$K_{ll} \cdot ul \leq P_{l2}$$

Equation 13-207.

$$u_{a2} = \begin{bmatrix} u_{l2} \\ 0_{r2} \end{bmatrix}$$

Equation 13-208.

Solution for Trial Vectors by Static Analysis

Inertia relief analysis is NOT performed, despite the presence of SUPORTi entries, usually used in the context of inertia relief. In this context, SUPORTi data or its equivalent for the auto-support feature is used for SPC-type operations. In CMS at present, K_{oo} may not contain rigid body modes. There is no provision to constrain them with SUPORT-type inputs. DOFs may be removed from the o-set to make it stable. For example, in superelement analysis, an interior point active in the rigid body mode, which by default is in the o-set, can be made an exterior point. This changes it to the a-set, which is held fixed when computing CMS modes. If the rigid body modes still exist downstream, they can be constrained in residual structure calculations.

The u_{a2} trial vectors, or "raw static shapes", as they are called in the SubDMAP comments, may not be linearly independent of the mode shapes, and are certainly not orthogonal to the mode shapes or each other. An orthogonalization process is now used to insure that the resulting shapes are orthogonal to the mode shapes and each other. Shapes that have little information after this process are discarded.

Orthogonalization

The trial vectors u_{a2} may not be linearly independent of each other or the mode shapes. They are processed to find a set of combined mode shapes and linear combinations of the trial shapes that are all orthogonal to each other with respect to the mass matrix. This reduces the risk of ill-conditioning when using these matrices for solution later.

The trial vectors are appended to ϕ_a to form the combined set ϕ_{ua1}

$$\phi_{ua1} = \left[\phi_a \mid u_{a1} \right]$$

Equation 13-209.

An optional pre-sweeping of trial vectors is done when requested by PARAM, RESVSO, YES. The derivation of the sweeping equation is similar to that for sweeping loads above, with the equation for x , the vector of linear combination coefficients, being

$$x = M_{i-1} \cdot \phi_a \cdot M_{aa} \cdot u_{a2s}$$

Equation 13-210.

The swept displacement shape vectors ϕ_{ua1} are then determined by eliminating x ,

$$\phi_{ua1} = u_{a2s} - \phi_{a2} \cdot M_i^{-1} \cdot \phi_{a2} \cdot M_{aa} \cdot u_{a2s}$$

Equation 13-211.

If the optional sweeping is not selected ϕ_{a1} is merely equal to ϕ_{a2} .

The u_{a1} vectors are mass-normalized into ϕ_{ua} such that the diagonal terms of the product on the left side are unity

$$\phi_{ua} \cdot M_{aa} \cdot \phi_{ua} = M_{zz}$$

Equation 13-212.

$$\text{diag}(M_{zz}) = I_{zz}$$

Equation 13-213.

M_{zz} is generally fully coupled at this point. It is decomposed with an LTL factorization, $M_{zz} = L \cdot D \cdot L$, where L is lower triangular and D is diagonal.

The ratio of D over the diagonal terms of M_{zz} is calculated in the vector R ,

$$R = \left\{ \frac{\text{diag}(M_{zz})}{\text{diag}(D)} \right\}$$

Equation 13-214.

Rows of R whose value exceeds PARAM, RESVRAT (default=1.e8) indicate columns of ϕ_{ua} that have little new content, and are discarded to produce U_{a3} and M_{zz3} . M_{zz3} is diagonal in the region of the eigenvectors, but coupled in the region of the auxiliary shapes. It is then re-named as M_{zz} .

The generalized stiffness corresponding to M_{zz} is calculated,

$$K_{zz} = u_{a3} \cdot K_{aa} \cdot u_{a3}$$

Equation 13-215.

The eigensolution of the z-set is obtained,

$$[K_{zz} - \lambda_z \cdot M_{zz}] \cdot \phi_z = 0$$

Equation 13-216.

The eigenvectors ϕ_z of this reduced basis are used as a transformation to orthogonalize U_{a3} ,

$$\phi_a = \phi_{ua} \cdot \phi_z$$

Equation 13-217.

The eigenvalue table of the combined system, λ_z , replace the prior eigenvalue table that contained modes only. λ_z includes both structural modes and the effects of orthogonalized residual flexibility vectors.

Model Partitioning in the MODEFSRS subDMAP

MODEFSRS is used in the residual structure to separate the fluid and structural components of the a-set. The modes of these components are computed independent of one another to find ϕ_{sa} and ϕ_{fa} , the structural and fluid modes, respectively. Once computed, residual flexibility effects are added with calls to the RESVEC SubDMAP.

The eigenvectors with residual flexibility effects appended to them λ_a are then merged together to produce one eigenvector matrix,

$$\phi_a = \begin{bmatrix} \phi_{sa} & 0 \\ 0 & \phi_{fa} \end{bmatrix}$$

Equation 13-218.

For the cases where only fluid or only structural modes are requested, the identity matrix is substituted for the eigenvector matrix that is not calculated.

Complex Eigenvalue Analysis in SubDMAP CEIGRS

Complex eigenvalues and the associated eigenvectors are calculated in CEAD (Complex Eigenvalue Analysis – Displacement) using the inverse power method, the determinant method, the Hessenberg method, or the Lanczos method, as you request on the EIGC Bulk Data entry. For direct complex eigenvalue analysis and if PARAM,ASING = 0 (default) then null rows and columns are discarded from $[K_{dd}]$, $[M_{dd}]$, and $[B_{dd}]$ to form $[K_{xx}^d]$, $[M_{xx}^d]$, and $[B_{xx}^d]$.

In the case of a direct formulation CEAD extracts the eigenvalues from the following equation.

$$[M_{xx}^d p^2 + B_{xx}^d p + K_{xx}^d] \{u_x^d\} = 0$$

Equation 13-219.

$\{u_x^d\}$ is then augmented with null rows to form $\{u_d\}$.

In the case of a modal formulation the following equation is used:

$$[M_{hh} p^2 + B_{hh} p + K_{hh}] \{u_h\} = 0$$

Equation 13-220.

CEAD also normalizes the eigenvectors according to one of the following user requests:

- Unit magnitude of a selected coordinate (“POINT”).
- Unit magnitude of the largest component (“MAX”).

as specified on the EIGC Bulk Data entry.

The complex Lanczos method is recommended for large problems with sparse matrices. The HESS method is best suited to small dense matrices, typical of small models or those typical in a modal formulation.

Frequency Response and Random Analysis in SubDMAP FREQRS

The frequency-dependent loads are generated in FRLG (Frequency Response Load Generation) from the RLOAD1 and RLOAD2 Bulk Data entries. Constraints are applied to reduce the loads to the d-set, and in the case of a modal formulation, the loads are transformed to the h-set.

The solution for the response is performed in FRRD1 or FRRD2 (Frequency Response Solution). In the case of a direct formulation the following equation is solved.

$$[-M_{dd} \omega^2 + i B_{dd} \omega + K_{dd}] \{u_d\} = \{P_d\}$$

Equation 13-221.

and in the case of a modal formulation, the following equation is used in:

$$[-M_{hh}\omega^2 + iB_{hh}\omega + K_{hh}]\{u_h\} = \{P_h\}$$

Equation 13-222.

The arithmetic used in the solution may be real or complex and the solution procedure may be symmetric or unsymmetrical. The choice of arithmetic and solution procedure is made by the program depending on the form of the dynamic matrices.

Power spectral density functions and autocorrelation functions are computed in RANDOM (Random Analysis) according to the information on RANDPS and RANDT1 Bulk Data entries. Random analysis calculations are made for selected displacements, loads, single point constraint forces, element stresses and element forces. The selection of items for random analysis is made in the XY-output request part of the Case Control Section.

Transient Response in SubDMAPs SEDTRAN and SEMTRAN

The linear time-dependent loads are generated in TRLG (Transient Response Load Generation) from the TLOAD1 and TLOAD2 Bulk Data entries. The nonlinear loads are generated from the NOLINI Bulk Data entries. Constraints are applied to reduce the loads to the d-set, and in the case of a modal formulation, the loads are transformed to the h-set.

The solution for the response is performed in TRD1 (Transient Response Solution). In the case of a direct formulation the following equation is integrated over the time periods specified via the TSTEP Bulk Data entry in subDMAP DTRANRS:

$$[M_{dd}p^2 + B_{dd}p + K_{dd}^x]\{u_d\} = \{P_d\} + \{P_d^{nl}\}$$

Equation 13-223.

in order to determine the displacement velocity and acceleration response.

In the case of a modal formulation, the following equation is used in subDMAP MTRANRS:

$$[M_{hh}p^2 + B_{hh}p + K_{hh}]\{u_h\} = \{P_h\} + \{P_h^{nl}\}$$

Equation 13-224.

If there are no direct input matrices and no nonlinear loads, an analytical integration procedure is used rather than numerical integration. The numerical integration procedure may use either symmetric or unsymmetrical routines. The choice is made by the program depending on the form of the dynamic matrices.

Buckling Analysis in SubDMAP MODERS

In linear buckling, a static load is applied to the structure and the following eigenvalue problem is solved subsequently,

$$[K_{aa} + \lambda_i K_{aa}^d] \{\phi_i\} = 0$$

Equation 13-225.

where $[K_{aa}]$ is the linear stiffness, $[K_{aa}^d]$ is the differential stiffness, λ_i are the eigenvalues and $\{\phi_i\}$ are the eigenvectors (buckling modes). The critical buckling loads are

$$\{P_i^{crit}\} = \lambda_i \{P_a\}$$

Equation 13-226.

where $\{P_a\}$ is the vector of applied static loads including reaction forces.

The basic assumption in linear buckling is that the differential stiffness $[K_{aa}^d]$ is proportional to the applied load $\{P_a\}$. The assumption implies a linear material law and small deformations. For buckling in nonlinear problems see the *NX Nastran Basic Nonlinear Analysis User's Guide*.

The linear stiffness $[K_{aa}]$ is derived from the variation of the stresses, the differential stiffness $[K_{aa}^d]$ is derived from the variation of the strains keeping the stresses constant.

SOL 105 is the recommended solution sequence for linear buckling in static analysis. The Case Control Section must contain at least two subcases. A static loading condition is applied in the first subcase. The load is defined with a LOAD, TEMP(LOAD), or DEFORM Case Control command. The load may also be defined with enforced deformations on the SPC Bulk Data entries. In the second subcase, a METHOD command must appear to solve the eigenvalue problem. The Lanczos method (EIGR Bulk Data entry) is the recommended eigenvalue solver for linear buckling.

Linear buckling is available in the following solution sequence.

SOL 105 Linear buckling in statics with superelements.

The linear buckling capability is available for the following elements: CONROD, CROD, CTUBE, CBAR, CBEAM, CBEND, CQUAD4, CTRIA3, CTRIA6, CSHEAR, CHEXA, CPENTA, CPYRAM, AND CTETRA.

The differential stiffness formulation for the CQUAD4 and CTRIA3 elements was changed in Version 68. Numerous spurious modes appeared in linear buckling of thin shell structures using versions prior to Version 68. The improved results of Version 68 can be produced with a version earlier than Version 68 by overlapping two CQUAD4 or CTRIA3 elements, one with bending stiffness only and one with membrane stiffness only. The pre-Version 68 method of calculating the differential stiffness may be activated in Version 68 by setting the system cell NASTRAN SYSTEM(170)=1 in the NASTRAN statement. The Version 68 method is the default, NASTRAN SYSTEM(170)=0. The pre-Version 68 method remains in the code to recover old results.

The following recommendations apply to linear buckling analysis.

- Use at least 5 elements per half sine wave to get reasonable results in the buckling load.
- For buckling of 3-D shell structures, use PARAM,K6ROT,100. to assign a stiffness to the sixth degree-of-freedom. The default is PARAM,K6ROT,0.

The following restrictions apply to linear buckling analysis.

- Offsets should not be used in beam, plate or shell elements. The buckling loads for structures with offsets are incorrect.
- Follower force effects are not included in the differential stiffness. The calculated buckling loads are incorrect. The Bulk Data entries FORCE1, FORCE2, MOMENT1, MOMENT2, PLOAD, PLOAD2, PLOAD4, RFORCE, TEMP, TEMPD, TEMPP1, TEMPP3, TEMPRB describe follower forces.

Superelement Analysis

The theoretical aspects of superelement analysis are discussed in the *NX Nastran Superelement User's Guide*. Superelements are mathematically equivalent to substructures. The major difference is in a more convenient user interface and in the ability to process a much broader class of problems than is practical for most other substructuring capabilities.

There are now two distinct paths through superelements in the solution sequences. There are also two major types of superelements.

- When a BEGIN SUPER entry is present the program uses the more modern SEP1X module to make the SEMAP table used to control partition of superelements.
- When there is a BEGIN BULK entry but no BEGIN SUPER entries a parallel path using the older SEP1 module is used instead.

While the newer module supports most features of both part and grid list superelements, it does not have some of the advanced features supported in the older module. When starting a new project you should decide at modeling initiation which path is most suitable because transitioning from one to another can be painful, particularly when the model is in development. If one starts on the old path, for example, and then adds a new feature that causes the program to automatically switch to the new path, some of your modeling input may be invalidated. This may cause considerable, confusing, and unnecessary work to transition to the new path.

The short answer is that the new path should be the best path for most projects, based on the appeal of its advanced features and the likelihood that future improvements in NX Nastran may require use of the new path. The new path is forced by replacing the BEGIN BULK entry with a BEGIN SUPER entry in your Main Bulk Data file. It should be the first entry in your bulk data. It allows grid list superelements, but they need not be present.

The exception for the path chosen is when your project requires the use of features available in the old path but not yet enabled in the new path. Some of these features may be used in the new paths by special avoidance techniques, but these techniques can be cumbersome. It may be more convenient to stay with the older methods until the newer methods catch up.

Terminology

- **Newest Option:** The partitioned bulk data file superelements (parts) are the form introduced in Version 69. They are sometimes called substructures. A major feature of parts is ID independence. That is, element IDs and grid points IDs may be duplicated across different parts, although they must be unique within a part.
- **Intermediate:** The Main Bulk Data Superelements (superelements) introduced in Version 47 are sometimes called grid list superelements (list s.e.s). When used in the new path, they have more restrictive rules than in the old path.

- **Oldest Path:** List s.e.s only are used in the old path. There are no formal names at present to differentiate the intermediate and old flavors. Some advanced features are available only in this path, at present.

A subtle difference between the two paths is that the new path always uses a single level configuration for list s.e.s if there is no SETREE-type entry. It does this by defining new boundary points in the residual structure when some are needed but not supplied by you. The old path automatically converts to a multilevel configuration when required, regardless of the presence of DTI, SETREE entries. It has no ability to generate new boundary points. This type of change when switching from the old path to the new, on purpose or by accident can be confusing.

Criteria for Selecting Part Versus List Superelements

Each of the two major forms has overwhelming advantages in certain contexts. Some of these advantages are of a temporary nature, as the rest of the technology in supporting FE tools catch up to the newest methods. However, other relative advantages are due to the inherent difference in concept between the two major branches that will be relevant into the foreseeable future. It is likely that both branches will be maintained and extended because of their unique capabilities. The following criteria should be considered when making a choice:

- Converting an existing non-superelement model to a part configuration is a labor-intensive process at present. No migration aides are provided. Converting it to a list superelement requires addition of only SESET entries, a trivial task by comparison.
- When debugging a new model it is much easier to comprehend and more efficient when it is in a superelement configuration. Errors in one area will not obscure errors in other areas, and the model solved is smaller, leading to faster turnaround. It may be more economical to run it in a non-superelement configuration for production runs after the model is stable and requires no more changes. If it is a list superelement this means merely removing the SESET entries, after which it is automatically an exactly equivalent non-superelement model if advanced features such as those provided by the RELEASE and CSUPEXT entries are avoided. If it is a part superelement there can be a large labor cost for conversation, as all duplicated IDs will need to be changed, duplicate points at boundaries will need to be resolved, etc. A list s.e. model, to make virtue of necessity, enforces rules such as unique IDs when updating the model that make re-conversion to a non-s.e. configuration an almost automatic process.
- When starting a new modeling project where more than one person provides input, and most particularly when more than one organization contributes to the model, the higher degree of data control for part superelements is attractive. There is no need to legislate acceptable ID ranges for each part. It is possible to lock the description of a part into its boundary matrices, in a data base marked with a time stamp, so that there is little questions about who is responsible for the behavior of each part.
- Many GUIs can be used to modify list superelements because the rules for ID independence are the same for GUIs as they are for list s.e.s Part superelements can be modeled in GUIs only one at a time. This makes boundary matching, in particular, cumbersome.

Features Not Supported in the New Path

- **Fluid Structure Analysis:** This capability is supported in the new path with the “old” path rules:
 1. All fluid elements must be in a special list superelement, and/or in the residual structure

2. This special superelement may have fluid elements only; structural elements or grid points are not allowed
 3. One or several fluid cavities may be in the fluid superelement
 4. Fluid elements may be connected to structural elements in list superelements, or in the residual structure.
 5. Elements in part superelements may not be connected to fluid elements.
- p-Elements: Not supported at all. It may be possible to segregate the portions of the model containing p-elements into external superelements, but this will be a labor-intensive process. No documentation is available on this topic at present.
 - RELEASE Entries: Allowed only in the old path. There are semi-heroic modeling techniques that may be used to simulate this feature in the new path, but they are not documented.
 - SExSET Entries: Most set entries for non-superelement models such as ASET have synonyms for use in list s.e.s such as SEASET. The synonym is not accepted in PARTs, but the simpler format of ASET, for example, is accepted.

Features Not Supported in the Old Path

In general, features associated with automatic boundary control are not supported. The QRG is not explicit on this issue. A list of entries not supported in the old path is:

SETREE, SENQSET, SEBNDRY, SEBULK, SECONCT, SEEXCLD, SELABLE, SELOC, SEMPLN

The DTI, SETREE entry is used in place of SETREE. The action of the SEELT entry is somewhat different. For the new path, the SEMAP will be changed to account for the action of SEELT entries. For the old path, it is assumed that the element being moved is one that can be moved without changing the SEMAP, such as an element attached only to boundary points. If the SEELT requires change of the SEMAP the program processes conventionally until it encounters the s.e. with the “illegal” SEELT entry, then gives a fatal error exit stating that the connectivity of the superelement is incorrect.

Check List for Transitioning an Old Model to the New Path

- Replace the BEGIN BULK entry with a BEGIN SUPER entry.
- Check the SEMAP table from an old run to determine if it is in a multilevel state. When it is, add SETREE entries that define the tree in the old run, or convert it to a single level model if multilevel is not essential. If you have many multilevel models to convert, there is a parameter that will request that SETREE entries be made from the SEMAP, and placed in the .pch (punch) file.
- Discard all RELEASE and CSUPEXT entries and p-elements. If you cannot find reasonable replacements for them you may be forced to stay in the old path.

Decide if this is the right time to convert to a PART model, or whether the list s.e. capability in the new path is more advantageous, based on the discussion above.

The Case Control commands for superelement processing selection all begin with the letters SE (SEMG, SELG, SEKR, SEMR, SELR, SEALL, and SEEX). The SEFINAL command is used to control the order of processing of the superelements. A subcase structure is required that defines

constraints, loads, and data recovery options for each superelement. Each subcase includes the Case Control command SUPER in order to relate the subcase to a particular superelement and load condition. Examples of Case Control (including plot requests) are given in the *NX Nastran Superelement User's Guide*. A set of plotting commands is required for each superelement to be plotted, separated by the plotter commands SEPLOT and SEUPLOT.

Each superelement is given an identification number (SEID). You assign each grid point to a superelement on a GRID or SESET Bulk Data entry. Identical image, mirror image, and external superelements are defined by the CSUPER and optional SEQSEP Bulk Data entries. The SEELT Bulk Data entry may be used to change the superelement membership of boundary elements. The RELEASE Bulk Data entry may be used to disconnect selected degrees-of-freedom from exterior grid points of a superelement.

The mesh generator option, called by MESH on the NASTRAN statement, supports the superelement capability fully, in that all GRID entries generated for a field can be given the same SEID, and the SESET Bulk Data entry can change these SEIDs at boundaries. The automatic grid point resequencing features are fully implemented for superelement models.

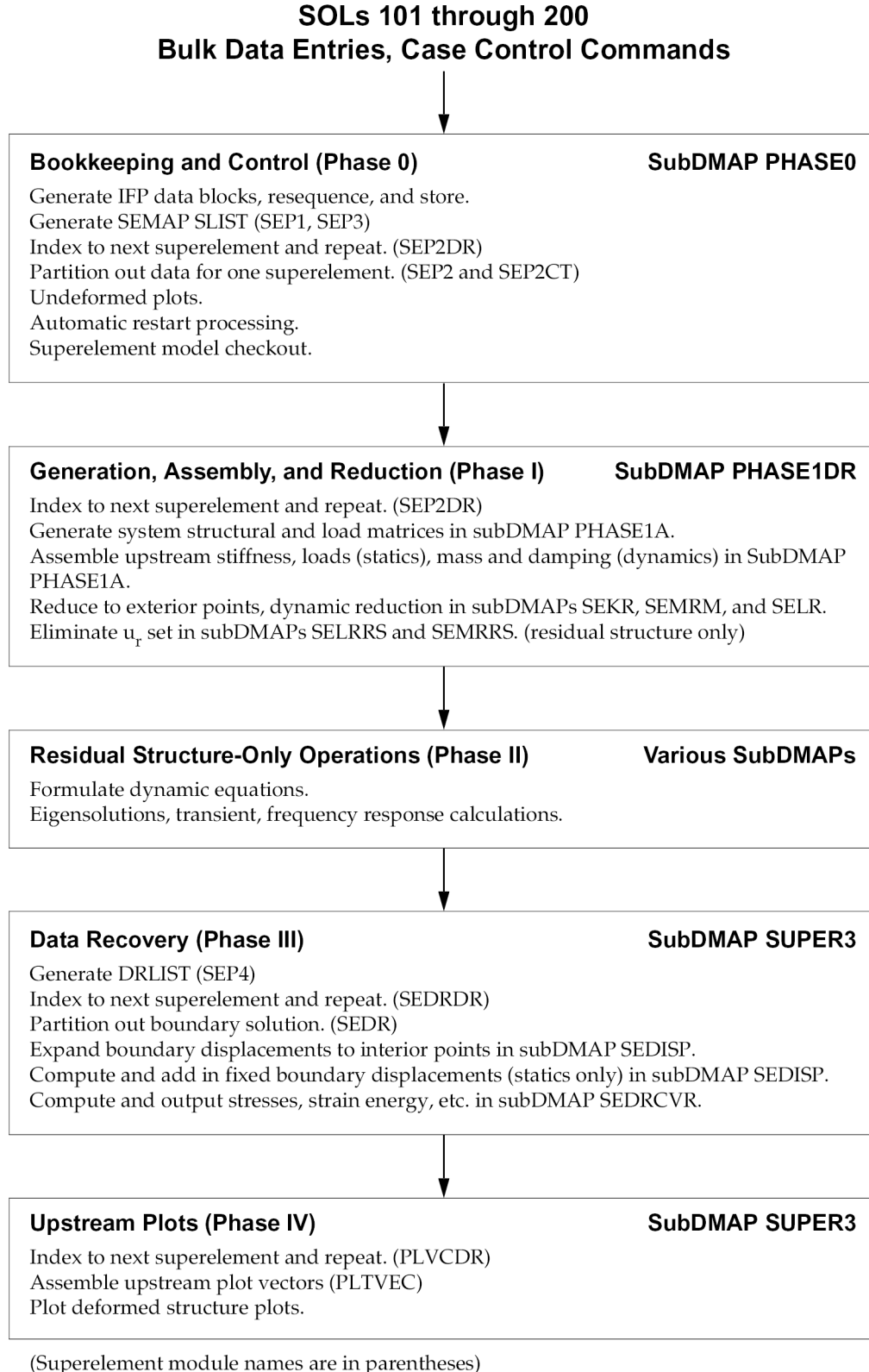
The superelement partitioning module passes on resequencing data from SEQGP Bulk Data entries only for grid points of a superelement which are not exterior points for any superelement. This class of grid points can be determined from the SEMAP table printed by the program, but may change if the order of processing is changed. The automatic resequencing option processes each superelement separately.

Component mode synthesis uses SEBSETi, SECSETi, SESUP and SEQSETi entries to define the boundary condition when computing component modes. Uncoupled solutions for statics and modes can be obtained by use of PARAM,FIXEDB,-1.

Superelement Bulk Data entries may be specified in nonsuperelement solution sequences, but those entries pertaining to the superelement capability (e.g., the CSUPER entry) are ignored, as well as some fields on other entries (e.g., Field 9 on the GRID entry). However, better automatic resequencing may be possible if the SEID data is removed.

Superelement Processing in the Structured Solution Sequences

An overview of the superelement processing in SOLutions 101-200 is shown in [Figure 13-3](#). Phase 0 performs the preparation and bookkeeping function. The input data is stored on the database and scanned to prepare the SEMAP table. This table lists the superelement to which each Bulk Data entry is to be applied. Phase 0 also partitions out the data that defines each superelement and performs automatic restart processing. In Phase I this data is used to generate the boundary matrices to be assembled when combining the superelements into the total structure. Phase II performs calculations done only on the residual structure (the superelement processed last), such as transient response calculations or eigensolutions. Phase III expands boundary solutions to the interior grid points of the superelement, performs the data recovery operations for internal forces, stresses, and strain energy, and outputs these quantities. Phase IV combines plot vectors from upstream superelements to provide deformed structural plots for all or part of the structure.

**Figure 13-3. Structured Solution Sequence Flow Chart**

The input file processor operations are carried out in a conventional manner. The grid point resequencer module resequences each superelement individually.

Some of the geometry processor operations of “**Geometry Processing in SubDMAP PHASEO**” are carried out to prepare tables of all grid points in the entire model, useful for generation of the SEMAP table. After the SEMAP is available, the SLIST table is assembled. This list controls the operations to be performed in that particular run for generating boundary matrices, as selected with Case Control commands. The GP4 module generates constraint data in the form of the USET table, as described in “**Geometry Processing in SubDMAP PHASEO**.” A separate USET table is generated for each superelement. It is similar to the USET table of the older rigid formats except that the exterior points are automatically placed in the a-set.

Assembly and Reduction (Phase I) in SubDMAP Phase1DR

The following operations may be carried out, as you select with superelement-type Case Control commands. By default, all superelements and the residual structure are given these operations serially. If requests for the SEMA (or SEMR) operation are made on an upstream superelement and the residual structure, all superelements in the tree between the selected superelement and the residual structure will also be given the SEMA operation.

Superelement Matrix Generation (SEMG) in SubDMAP PhaseIA

The element structural matrices (stiffness, structural damping, mass and viscous damping) are generated and stored. The direct matrix input option (DMIG Bulk Data entry) can be used to add stiffness, structural damping, mass or viscous damping terms to the system matrices in this operation. These terms are selected by the Case Control commands K2GG, K42GG, M2GG, and B2GG. The parameters CB1, CB2, CK1, CK2, CK41, CK42, CM1 and CM2 can be used to scale the structural matrices.

The SEMA module adds in stiffness terms from any upstream superelements. The SEEX Case Control command may be used to eliminate stiffness contributions from selected upstream superelements. The superelements that are upstream are defined by the current SEMAP. The number of rows and columns of the upstream superelements must be consistent with the number of exterior grid points, as defined by the SEMAP. In order to complete execution of the SEMA module, the boundary matrices of all upstream superelements must be present in the database, and consistent with the current SEMAP.

The SELA module adds loads from upstream superelements to the load vector for the current superelement. The SEEX Case Control command may be used to eliminate load contributions from upstream superelements. The requirements for conformity of load data in the database with the current state of the SEMAP are similar to those for stiffness, except that only rows, not columns of the load vectors are mapped using data in the SEMAP table. The requirement on columns of all load matrices is that they be equal to the number of columns (number of load condition) in the residual structure. If the number of columns in all superelements and the residual structure is not the same, a user fatal error results. Static loads may be applied to any grid point, whether exterior or interior to a superelement. Dynamic loads may be applied directly to residual structure points. Dynamic load entries may also reference static load sets applied to the superelements via the LOADSET Case Control command and LSEO Bulk Data entries.

Superelement Load Generation (SELG) in SubDMAP PHASE1A

The load generation module SSG1 is executed in the static and dynamic response solutions. This operation is not executed in the normal modes and complex eigenvalue solution sequences. As loads may depend on structural matrices (i.e., thermal or inertial loads), an SELG operation must be preceded by an SEMG operation. The resultants of these loads about the grid point listed on the PARAM,GRDPNT Bulk Data entry are output.

Superelement Stiffness Reduction (SEKR) in SubDMAPs SEKR and SEKRSS

You may change SPCs and/or MPCs between subcases (“boundary condition changes”). The GPSP module identifies grid point singularities, and, optionally, provides SPCs for singular degrees-of-freedom. This is followed by the constraint eliminations described in “[Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB](#),” “[Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB](#),” and “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#).” If the omitted degrees-of-freedom contain mechanism-type singularities, the grid and component number of all downstream ends of the mechanism are output in the matrix named MECHS, and a user fatal error stops execution. The approximate fixed-boundary natural frequencies of the superelement are printed in the FAPPROX matrix. The boundary stiffness matrix $[K_{aa}]$ is stored in the database. Mass and damping matrices are not processed in this phase.

SUPPORTi Bulk Data entries, which define the r-set, may reference only grid points in the residual structure. The rigid body operations for the stiffness are performed as described in “[Static Solutions in SubDMAP SEKRSS](#).”

Superelement Load Reduction (SELR) in SubDMAP SELR

This operation processes loads in the static and dynamic response solution sequences. This operation is not executed in the normal modes and complex eigenvalue solution sequences. The static load reduction of “[Static Condensation in SubDMAPs SEKR and SEMR2](#)” is applied, stopping after generation of the $[P_a]$ matrix for superelements and $[P]$ for the residual structure.

Superelement Mass Assembly and Reduction (SEMR) in SubDMAP SEMRM

In the dynamics solution sequences, the constraint operations of “[Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB](#),” “[Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB](#),” and “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#)” are applied to the mass and damping matrices. The rigid body operations of “[Static Solutions in SubDMAP SEKRSS](#)” are applied to the mass of the residual structure only. Dynamic reduction and component mode calculations are performed. This allows a mass-change-only type of restart by use of the SEMG and SEMR options and avoids the costly reduction of the stiffness matrix in the SEKR operation if no stiffness changes are made.

The optional Bulk Data entry PARAM,ERROR,0 affects operations in Phase I. If it is not used, the default action is to terminate the run after the first user fatal error, as in the older rigid formats. If the parameter is used, a user fatal error causes a branch to the end of Phase I for the superelement, and continuation to the generation, assembly, and reduction operations for any remaining superelements. The five SE-type commands may be executed on one run, or may be done separately. The following sequence must be observed if these commands are executed separately:

SEKR after SEMG
SELG after SEMG

SELR after SEKR, SELG, SEMR
SEMR after SEKR

Residual Structure-Only (Phase II)

The operations on the superelements described above are also performed on the residual structure. This is followed by operations unique to the residual structure. Eigensolutions and direct and modal forced response calculations take place at this point. Direct input matrices, when selected by the Case Control commands M2PP, K2PP, and B2PP are constrained and added as in the “[Formulation of Dynamic Equations in SubDMAP GMA](#).”

Bulk Data entries used for static loads (i.e., FORCE, GRAV, LOAD, etc.) may be used in direct and modal dynamic analysis. They are interchangeable with DAREA Bulk Data entries and may be referenced by LSEQ Bulk Data entries.

Data Recovery Operations (Phase III) in SubDMAP SUPER3

The total solution vectors of the downstream superelement are partitioned to generate the boundary solution vectors for the superelement being processed. The boundary solution is expanded to all interior points of the superelement. The fixed boundary component is computed and added.

If SPC force output is requested, the resultants of the SPC forces about PARAM,GRDPNT or the basic coordinate system origin are also output. The data recovery operations of “[Static Solutions in SubDMAP SEKRRES](#)” are carried out. (Note that solution sequences involving complex eigenvalue or frequency response analysis may produce complex displacement vectors. GPFDR module output (grid point force balance, element strain energy) may be requested for complex vectors, but only the real component of the vector is processed.) Deformed structure plots of only elements in the superelement are generated here.

Upstream Plots (Phase IV) in SubDMAP SUPER3

The displacement vectors of a superelement and all of its upstream members are combined and plotted at your request.

Superelement Processing

An overview of the superelement processing is shown in [Figure 13-4](#). Phase 0 performs the preparation and bookkeeping function. The input data is stored on the database and scanned to prepare the SEMAP table. This table lists the superelement to which each Bulk Data entry is to be applied. Phase I partitions out the data that defines one superelement and processes this data to generate the boundary matrices to be assembled when combining the superelements into the total structure. Phase II performs calculations done only on the residual structure (the superelement processed last), such as transient response calculations or eigensolutions. Phase III expands boundary solutions to the interior grid points of the superelement, performs the data recovery operations for internal forces, stresses, and strain energy, and outputs these quantities. Phase IV combines plot vectors from upstream superelements to provide deformed structural plots for all or part of the structure.

The input file processor operations are carried out in a conventional manner. The grid point resequencer module resequences each superelement individually.

Some of the geometry processor operations of “Geometry Processing in SubDMAP PHASE0” are carried out to prepare tables of all grid points in the entire model, useful for generation of the SEMAP table. After the SEMAP is available, the SLIST table is assembled.

This list controls the operations to be performed in that particular run for generating boundary matrices as selected with Case Control options.

Assembly and Reduction (Phase I)

The following operations may be carried out as you select on superelement-type Case Control commands. By default, all superelements and the residual structure are given these operations serially. If requests for the SEMA (or SEMR) operation are made on an upstream superelement and the residual structure, all superelements in the tree between the selected superelement and the residual structure will also be given the SEMA operation.

Superelement Matrix Generation (SEMG)

The element structural matrices (stiffness, structural damping, mass and viscous damping) are generated and stored. The direct matrix input option (DMIG Bulk Data entry) can be used to add stiffness, structural damping, mass or viscous damping terms to the system matrices in this operation. These terms are selected by the Case Control commands K2GG, K42GG, M2GG, and B2GG. The parameters CB1, CB2, CK1, CK2, CK41, CK42, CM1, and CM2 can be used to scale the structural matrices.

Superelement Load Generation (SELG)

The load generation module SSGI is executed in the statics and dynamic response solutions. This operation is not executed in the normal modes and complex eigenvalue solution sequences. As loads may depend on structural matrices (i.e., thermal or inertial loads), an SELG operation must be preceded by an SEMG operation. The resultants of these loads about the grid point listed on the PARAM,GRDPNT Bulk Data entry are output.

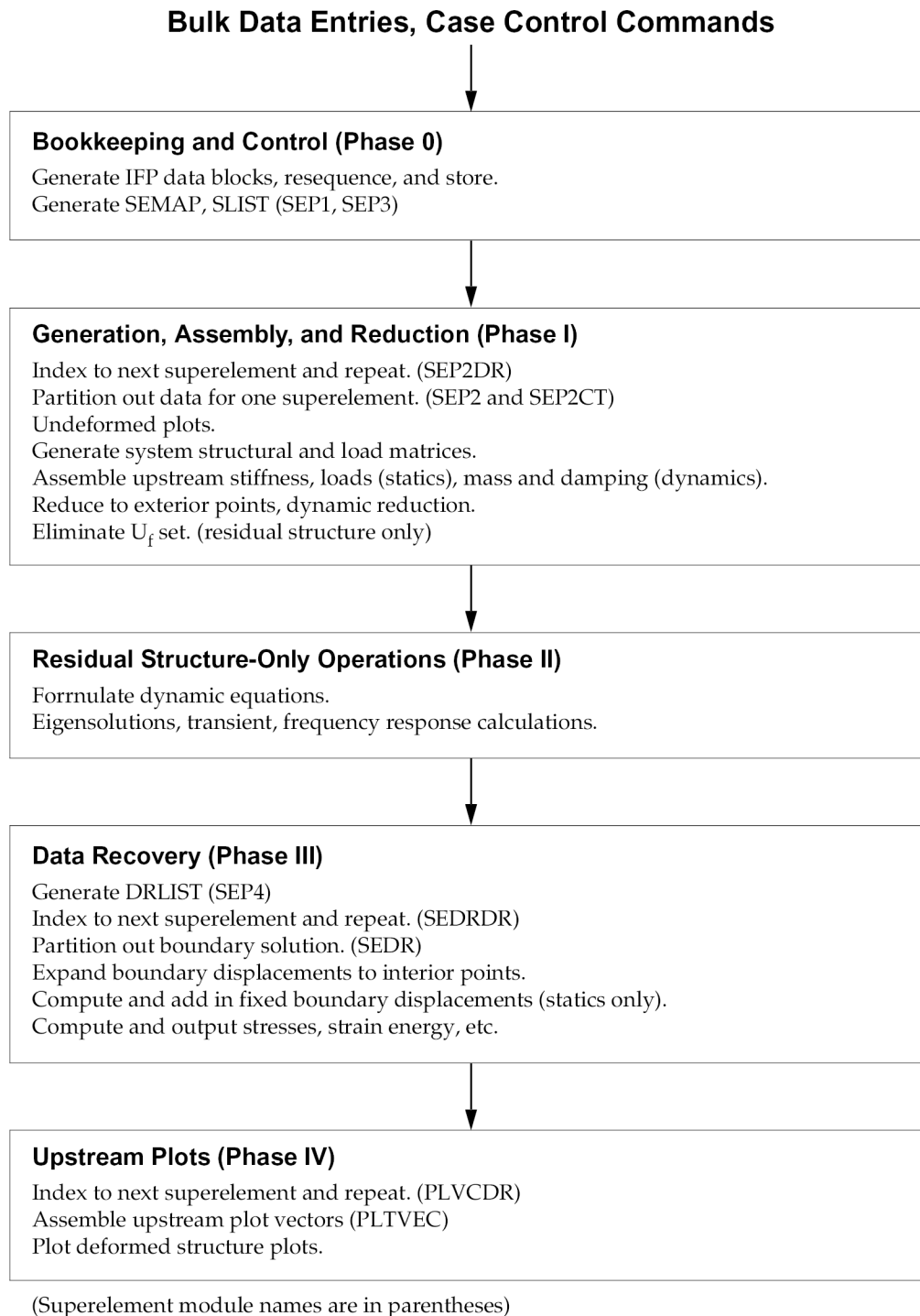


Figure 13-4. Solution Sequence Flow Chart

Superelement Stiffness Matrix Assembly and Reduction (SEKR)

The SEMA module adds in stiffness terms from any upstream superelements. The SEEX Case Control command may be used to eliminate stiffness contributions from selected upstream

superelements. The superelements that are upstream are defined by the current SEMAP. The number of rows and columns of the upstream superelements must be consistent with the number of exterior grid points, as defined by the SEMAP. In order to complete execution of the SEMA module, the boundary matrices of all upstream superelements must be present in the database, and consistent with the current SEMAP. The GP4 module generates constraint data in the form of the USET table, as described in “[Geometry Processing in SubDMAP PHASEO](#).” A separate USET table is generated for each superelement. It is similar to the USET table of the older rigid formats except that the exterior points are automatically placed in the a-set.

You may change SPCs and/or MPCs between subcases (“boundary condition changes”) in SOLs 101 and 103 only. The GPSP module identifies grid point singularities, and at user's option provides SPCs for singular degrees-of-freedom. This is followed by the constraint eliminations described in “[Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB](#),” “[Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB](#),” and “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#).” If the omitted degrees-of-freedom contain mechanism-type singularities, the grid and component number of all downstream ends of the mechanism are output in the matrix named MECHS, and a user fatal error stops execution. The approximate fixed-boundary natural frequencies of the superelement are printed in the FAPPROX matrix. The boundary stiffness matrix $[K_{aa}]$ is stored in the database. Mass and damping matrices are not processed in this phase.

SUPPORT Bulk Data entries, which define the r-set, may reference only grid points in the residual structure. The rigid body operations for the stiffness are performed as described in “[Static Solutions in SubDMAP SEKRRS](#).”

Superelement Load Assembly and Reduction (SELR)

This operation processes loads in the static and dynamic response solution sequences. This operation is not executed in the normal modes and complex eigenvalue solution sequences. The SELA module adds loads from upstream superelements to the load vector for the current superelement. The SEEX Case Control command may be used to eliminate load contributions from upstream superelements. The requirements for conformity of load data in the database with the current state of the SEMAP are similar to those of the SEKR operation above, except that only rows, not columns of the load vectors are mapped using data in the SEMAP table. The requirement on columns of all load matrices is that they be equal to the number of columns (number of load condition) in the residual structure. If the number of columns in all superelements and the residual structure is not the same, a user fatal error results. Static loads may be applied to any grid point, whether exterior or interior to a superelement. Dynamic loads may be applied directly to residual structure points. Dynamic load entries may also reference static load sets applied to the superelements via the LOADSET Case Control command and LSEQ Bulk Data entries. The static load reduction of “[Static Condensation in SubDMAPs SEKR and SEMR2](#)” is applied, stopping after generation of the $[P_a]$ matrix for superelements and $[P]$ for the residual structure.

Superelement Mass Assembly and Reduction (SEMR)

In the dynamics solution sequences, the constraint operations of “[Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB](#),” “[Single Point Constraint Operations in SubDMAPs SEKR, SEMR2 and SEMRB](#)” and “[Constraint and Mechanism Problem Identification in SubDMAP SEKR](#)” are applied to the mass and damping matrices. The rigid body operations of “[Static Solutions in SubDMAP SEKRRS](#)” are applied to the mass of the residual structure only. Dynamic reduction and component mode calculations of “[Static and Dynamic Load Generation](#)” are performed. This allows

a mass-change-only type of restart by use of the SEMG and SEMR options and avoids the costly reduction of the stiffness matrix in the SEKR operation if no stiffness changes are made.

The optional Bulk Data entry PARAM,ERROR,0 affects operations in Phase I. If it is not used, the default action is to terminate the run after the first user fatal error, as in the older rigid formats. If the parameter is used, a user fatal error causes a branch to the end of Phase I for the superelement, and continuation to the generation, assembly, and reduction operations for any remaining superelements.

The five SE-type commands may be executed in one run, or may be done separately. The following sequence must be observed if these commands are executed separately:

SEKR after SEMGSELG after SEMGSELR after SEKR, SELG, SEMRSEMR after SFKR

Residual Structure-Only (Phase II)

The operations on the superelements described above are also performed on the residual structure. This is followed by operations unique to the residual structure. Eigensolutions and direct and modal forced response calculations take place at this point. Direct input matrices, when selected by the Case Control commands M2PP, K2PP, and B2PP are constrained and added as in “**Formulation of Dynamic Equations in SubDMAP GMA.**”

Bulk Data entries used for static loads (i.e., FORCE, GRAV, LOAD, etc.) may be used in direct and modal dynamic analysis. They are interchangeable with DAREA Bulk Data entries and may be referenced by LSEQ Bulk Data entries.

Data Recovery Operations (Phase III)

The total solution vectors of the downstream superelement are partitioned to generate the boundary solution vectors for the superelement being processed. The boundary solution is expanded to all interior points of the superelement. The fixed boundary component is computed and added.

If SPC force output is requested, the resultants of the SPC forces about PARAM,GRDPNT or the basic coordinate system origin are also output. The data recovery operations of “**Data Recovery Operations in SubDMAP SEDISP**” are carried out. (Note that solution sequences involving complex eigenvalue or frequency response analysis may produce complex displacement vectors. GPFDR module output (grid point force balance, element strain energy) may be requested for complex vectors, but only the real component of the vector is processed.) Deformed structure plots of only elements in the superelement are generated here.

Upstream Plots (Phase IV)

The displacement vectors of a superelement and all of its upstream members are combined and plotted at user's request.

Superelement Alternate Statics Solution

SOL 101 provides an alternate method of performing the superelement static condensation. PARAM,ALTRED,YES must be entered in the Bulk Data Section. In some cases, the alternate solution can result in significant reduction in computer cost for static analysis.

The stiffness and load reduction and displacement recovery are described in:

- “**Static and Dynamic Load Generation**”
- “**Static Condensation in SubDMAPs SEKR and SEMR2**”

- “Data Recovery Operations in SubDMAP SEDISP”

Static Superelement Analysis With Inertia Relief

Inertia relief analysis is available in SOL 101. PARAM,INREL,-1 or PARAM,INREL,-2 must be present for either of these options. The following method transforms static loads based on model geometry rather than the stiffness matrix as described in “Static Solutions in SubDMAP SEKRRS.”

The additional operations required to perform inertia relief are described in “Static and Dynamic Load Generation” and “Static Solutions in SubDMAP SEKRRS.”

Assumptions and Limitations

1. The model must have six and only six rigid body modes for a continuous, three-dimensional structure.
2. Masses on scalar points are ignored in calculating inertia loads.
3. The structural model, when given enforced displacements at the reference point, is assumed to produce the same motion as a rigid body model. This will be true for any structural or rigid element. It is possible, if unconventional, to use CELASi or general elements, MPC equations, or DMIG coefficients that will not meet this assumption.
4. Identical and mirror image superelements are not processed correctly.

Nonlinear Heat Transfer Analysis

In NX Nastran, you can perform:

- Steady state heat transfer analysis
- Transient heat transfer analysis

Steady State Heat Transfer Analysis

The steady state heat balance equation is given by

$$[K]\{u\} + [\mathcal{R}]\{u + T_{abs}\}^4 = \{P\} + \{N\}$$

Equation 13-227.

where:

- $[K]$ = a heat conduction matrix
- $[\mathcal{R}]$ = a radiation exchange matrix
- $\{P\}$ = a vector of applied heat flows that are constant
- $\{N\}$ = a vector of nonlinear heat flows that depend on temperature
- $\{u\}$ = a vector of grid point temperature
- T_{abs} = the absolute temperature

The components of the applied heat flow vector, $\{P\}$, are associated either with surface heat transfer or with heat generated inside the volume heat conduction elements. The vector of nonlinear heat flows, $\{N\}$, results from boundary radiation, surface convection, and temperature dependent thermal loads.

The equilibrium equation is solved by a Newton iteration scheme, where the tangential stiffness matrix is approximated by

$$[K_T]^i \approx [K]^i + 4[\mathcal{R}]^i \{u^i + T_{abs}\}^3 - \left\{ \frac{\partial N}{\partial u} \right\}^i$$

Equation 13-228.

and the residual vector is

$$\{R\}^i = \{P\} + \{N\}^i - [K]^i \{u\}^i - [\mathcal{R}]^i \{u^i + T_{abs}\}^4$$

Equation 13-229.

Transient Heat Transfer Analysis

The general equation solved in transient analysis has the form

$$[B]\{\dot{u}\} + [K]\{u\} + [\mathcal{R}]\{u + T_{abs}\}^4 = \{P\} + \{N\}$$

Equation 13-230.

To take phase change into consideration, the heat diffusion equation is converted into

$$\{\dot{H}\} + [K]\{u\} + [\mathcal{R}]\{u + T_{abs}\}^4 = \{P\} + \{N\}$$

Equation 13-231.

where:

- $[B]$ = heat capacity matrix
- $[K]$ = heat conduction matrix
- $[\mathcal{R}]$ = radiation matrix
- $\{P\}$ = vector of applied heat flows that are constant or functions of time
- $\{N\}$ = vector of nonlinear heat flows that depend on temperature
- $\{H\}$ = enthalpy vector
- $\{\dot{H}\}$ = $\{dH / dt\}$
- $\{u\}$ = grid point temperatures
- $\{\dot{u}\}$ = $\{du / dt\}$

T_{abs} = absolute temperature scale factor

The equilibrium equation is solved by Newmark's method with adaptive time stepping. Base on this one-step integration scheme, the time derivative of the nodal temperatures at the $(i + 1)$ th iteration of the time step $(n + 1)$ is expressed as

$$\{\dot{u}_{n+1}\}^{i+1} = \frac{1}{\theta \Delta t} \{u_{n+1}^{i+1} - u_n\} + \left(1 - \frac{1}{\theta}\right) \{\dot{u}_n\}$$

Equation 13-232.

where

$$\{u_{n+1}\}^{i+1} = \{u_{n+1}\}^i + \{\Delta u_{n+1}\}^i$$

Equation 13-233.

and

$$\frac{1}{\theta} = 2 - 2\eta$$

Equation 13-234.

The parameter η is specified on the PARAM,NDAMP Bulk Data entry. When $\eta = 0$ ($\theta = 0.5$), no numerical damping is requested. In this case, Newmark's method is equivalent to the Crank-Nicolson method.

Chapter 14: Modeling Guidelines

- *Introduction to Modeling Guidelines*
- *Choosing the Right Element*
- *Determining Mesh Density*
- *Creating Mesh Transitions*
- *Reviewing Grid Point Stresses*
- *Defining Consistent Loading*
- *Symmetry*

14.1 Introduction to Modeling Guidelines

With finite element modeling, the quality of your results depends directly upon the quality of your model. One of the more common errors that a beginning finite element analyst makes in modeling is to simply simulate the geometry rather than to simulate both the geometry and the physical behavior of the real structure. This section covers the following finite element modeling topics and guidelines:

- Choosing the right element
- Mesh density
- Mesh transitions
- Grid point stresses
- Consistent loading
- Symmetry

It is also good modeling practice to simulate and validate a new capability or a feature that you haven't used before with a small prototype model before applying this feature to your production model. Model verification techniques are covered in [“Model Verification.”](#)

14.2 Choosing the Right Element

NX Nastran contains a large library of structural elements. Often, you could use several elements to model the same structural effects. The criteria for the selecting an element may include its capabilities (for example, whether it supports anisotropic material properties), its cost (in general, the more DOF an element has, the more expensive it is), and/or its accuracy.

In many cases, the choice of the best element for a particular application may not be obvious. For example, in the model of a space frame, you may choose to use CROD elements if end moments are unimportant or to use CBAR elements if end moments are important. You may choose to use CBEAM elements with warping if the members have open cross sections and torsional stresses are estimated to be significant. You may even choose to represent the members with assemblies of plate or solid elements. The choice of which type and number of elements to use depends primarily on your assessment of the effects that are important to represent in your model and on the cost and accuracy you are willing to accept.

Given this, you should have a fairly good idea of how the structure will behave prior to generating your finite element model. In other words, understanding the load path is crucial in the selection of the appropriate element. Additionally, a few hand calculations can usually provide a rough estimate of stress intensities. If you don't have a fairly good idea of how the structure will behave, you may be misled by incorrect results due to errors or incorrect assumptions in your input data preparation.

General Guidelines for Element Selection

Always experiment with a small test model when using elements that you aren't familiar with. This practice is cheaper than experimenting with a large production model, and it gives you a better understanding of an element's capabilities and limitations prior to applying it to a large production model.

Avoid using obsolete elements. Elements are classified as obsolete only if their utility is completely superseded by newer elements and after the newer elements are in service long enough to be reliable. In most cases, they are retained in NX Nastran only to satisfy the requirements for upward compatibility. Elements that are considered obsolete are no longer included in any updated documentation.

Zero-dimensional Elements

When you use CELASi elements to represent concentrated springs between two components of translation, the directions of the two components must be coaxial. Even small deviations in direction can induce a significant moment to your model that does not exist in your physical structure. When you use a CELASi element, the locations of the two end points should be coincident to avoid this type of problem. If the two end points aren't coincident, consider using a CROD or CBUSH element instead.

One-dimensional Elements

If only an axial and/or torsion load is to be transmitted in an element, then the CROD is the easiest element to use.

A CBAR is easier to use than a CBEAM element. The I1 and/or I2 values can be set to zero. Use the CBEAM element instead of the CBAR element if any of the following features is important:

- The cross-sectional properties are tapered.
- The neutral axis and shear center do not coincide.
- The effect of cross-sectional warping on torsional stiffness is critical.
- The difference in the mass center of gravity and the shear center is significant.

The formulation for the CBEAM element is based on a flexibility approach; the element stiffness matrix is generated by inverting the flexibility matrix. For this reason, I1 and I2 must not be zero for the CBEAM element.

Two-dimensional Elements

In general, quadrilateral elements (CQUAD4 and CQUAD8) are preferred over the triangular elements (CTRIA3 and CTRIA6). The CTRIA3 element is a constant strain element. It is excessively stiff, and when used alone, it is generally less accurate than the CQUAD4 element, particularly for membrane strain. Whenever feasible, you should use the CQUAD4 element instead of the CTRIA3 element. CTRIA3 should only be used when necessary for geometric or topological reasons, for example, mesh transition between regions of quadrilateral elements with different meshes or near the polar axis of a spherical shell.

Additionally, you should avoid using CTRIA3s in locations where the membrane stresses are changing rapidly, for example, in the web of an I-beam. Since CTRIA3 has constant membrane stresses, a large number of them may be needed to obtain acceptable accuracy. It is better to use quadrilateral elements or CTRIA6 elements, if possible.

Don't use plate or shell elements (CQUADi, CTRIAi) in stiffened shell structures with very thin panels that can buckle. You should use shear panels (CSHEAR) in this case or in any situation where direct stresses cannot be supported, such as in a very thin curved panel.

Avoid highly skewed elements (see [Figure 14-1](#)). The angle α should be as close to 90 degrees as possible.

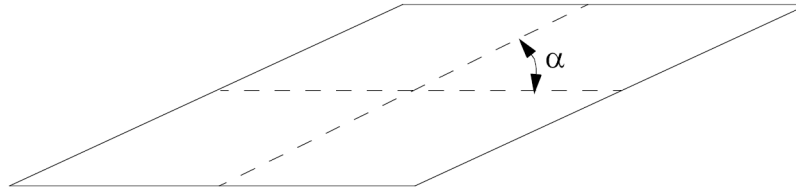


Figure 14-1. Highly Skewed Element

For the CTRIA3 element, the skew test is based upon the three vertex angles.

Aspect ratio is defined as l/ω (length/width). Very high aspect ratio (see [Figure 14-2](#)) should also be avoided, although it is no longer true that accuracy degrades rapidly with aspect ratios as it once did with some of the obsolete elements.

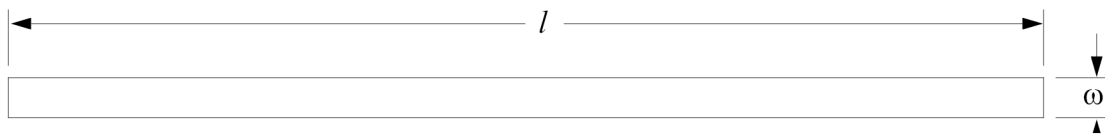


Figure 14-2. Element with High Aspect Ratio

Warping is a measure of the amount the element deviates from being planar (see [Figure 14-3](#)). Element warping should be minimized.

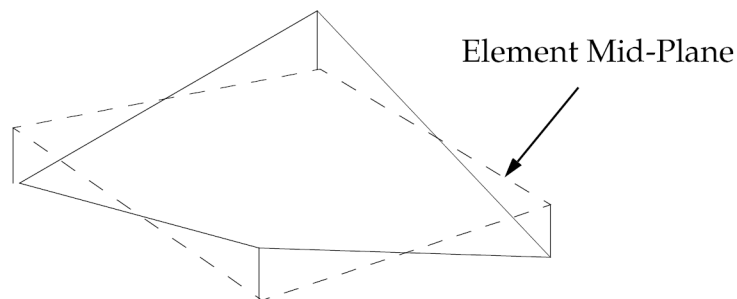


Figure 14-3. Highly Warped Element

For the CQUAD8 elements, if midside nodes are present, they should be located within the middle third of the edge. If a midside node is located at one-fourth the distance of the edge as measured from either corner node on the edge, the internal strain field becomes singular at the corners of the element. For best results, the midside node should be located as close to the center of the edge as possible. If you want to use midside nodes, you should include all of them. A CQUAD8 element with midside nodes deleted is excessively stiff and inferior to a CQUAD4 element.

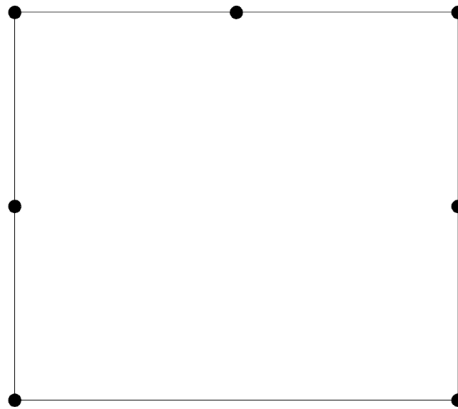


Figure 14-4. CQUAD8 with Missing Midside Node

For single curved structures (e.g., a cylinder), the CQUAD8, in general, yields better results than the CQUAD4 element. For doubly curved structures (e.g., a spherical dome), the CQUAD4 element, in general, performs better than the CQUAD8.

You should turn on the shell normal (PARAM,SNORM,X) option when using the CQUAD4, CTRIA3, CQUADR, or CTRIAR element. See the next section on shell normals for further details.

The membrane properties for the CQUADR and CTRIAR elements are less sensitive to the element shape than the CQUAD4 and CTRIA3. The same thickness should be used for all four corners ($T1 = T2 = T3 = T4$, or use the T Field on the PSHELL entry). The CQUADR and CTRIAR elements can be used in conjunction with each other, but not with other elements.

Shell Normals

By default, the direction of the normal rotation vector for flat plate elements is assumed to be perpendicular to the plane of each element. If the model is curved, the shell bending and twist moments must change direction at the element intersection. If transverse shear flexibility is present, the deformations may be too large. (Because elements using low-order formulations ignore the edge effect, this rarely causes any problems—a large value of the parameter K6ROT partially cures the problem.) With the unique normal (SNORM) option, the rotational degrees of freedom at each corner of an element are measured relative to the specified normal vector direction. Thus, all elements connected to a grid point will use a consistent direction for defining shell bending and twisting moments.

In CQUAD4 and CTRIA3 elements, the stiffness matrices of the elements are modified to eliminate the undesirable small stiffness in the rotational motions about the shell normal vector. In effect, the transformation replaces the normal moments with in-plane forces. No changes were made to the basic element stiffness matrix, and therefore flat plate models will not be affected. The objective of the new transformation was to remove a potential weakness in curved shell models and allow the automatic constraint process to remove the true singularity in the assembled stiffness matrix.

The CQUADR and CTRIAR elements are also improved for some types of shell problems. Because of their extra degrees of freedom, these elements are more sensitive to the coupling between in-plane and out-of-plane motion due to curvature. The new formulation provides more consistency between adjacent elements in a curved shell.

Shell normals are available for CQUAD4, CQUADR, CTRIA3, and CTRIAR elements. Normals are activated if the actual angle between the local element normal and the unique grid point normal is

less than 20° , the default value for β (see [Figure 14-5](#).) The default for β can be changed by setting `PARAM,SNORM, β` to the desired real value up to 89 degrees. The unique grid point normal is the average of all local shell element normals at a specific grid point. Generated grid point normals may be overwritten by user-defined normals.

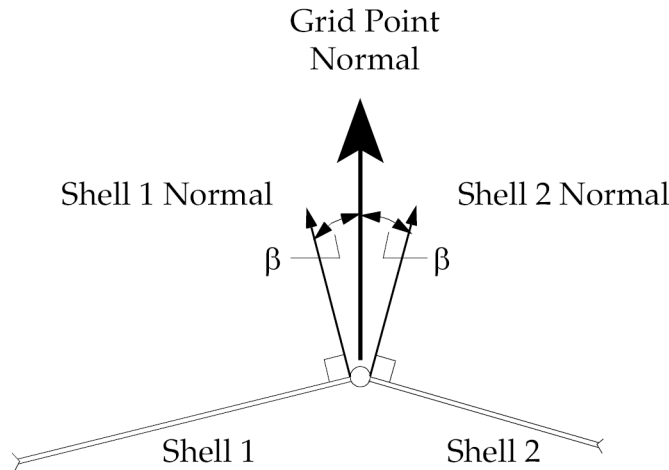


Figure 14-5. Unique Grid Point Normal

A shell normal defines a unique direction for the rotational degrees of freedom of all adjacent elements. A shell normal vector is created by averaging the normal vectors of the attached elements. If the actual angle is greater than the value defined on `PARAM,SNORM`, (default = 20.) the edge is assumed to be a corner, and the old method is used. Shell normals improve the accuracy of the results in curved shells where in-plane shear and twisting moments act together.

If the actual shell normal vector is known, such as in a cylinder or sphere, the automatically calculated values may be superseded by the actual vectors using the Bulk Data entry

```
SNORM,GID,CID,V1,V2,V3
```

where:

- GID is a unique grid point,
- CID is the coordinate system for defining the shell normal vector,
- V1, V2, and V3 are unscaled components of the vector in the coordinate system.

A second parameter, `SNORMPRT`, controls the print or punch of values of the internal shell normals. The output format is the same as the `SNORM` Bulk Data so that the individual values may be used and modified on a subsequent job.

Remarks and Recommendations for Shell Normals

1. The type of structure that exhibits the most change in results is a thick curved shell with large in-plane shear forces and twisting moments.

2. Most other problems, such as flat plates and curved shells with pressure loads, show changes in results of less than 1%. More degrees of freedom may be constrained using this improved formulation. This formulation results in zero in-plane rotational stiffness values.
3. It has been observed from testing that the automatic normal vector calculations produced answers nearly equal to and as accurate as those using the explicit SNORM vector input. In other words, the results were insensitive to small differences in the direction of the vectors. The important fact is that the shell normal vector requires the connected elements to use a consistent normal direction.
4. The CQUAD8 and CTRIA6 elements are not included in the shell normal processing. If they are modeled correctly, they do not require shell normal processing. Connecting these elements to the lower-order flat elements is not recommended.
5. Curved shell elements have no stiffness in the rotational degree of freedom when the new formulation is used-consequently, mechanisms may be introduced. Mechanisms occur when elements, RBEs, or MPCs are connected to the out-of-plane rotation of the shell grid points. Note that PARAM,AUTOSPC,YES does not constrain these mechanisms.
6. In linear solution sequences, the values of "param,k6rot,0." and "param,snorm,20." are the default.
7. Transverse shear flexibility (MID3 on the PSHELL property entry) should be left on when normals are used.

Three-dimensional Elements

Although the CHEXA and CPENTA elements are designed to behave reasonably well as thin shell elements, in general, you shouldn't use them in this capacity. The high ratio of extensional stiffness in the direction normal to the effective transverse shear stiffness can produce significant round-off errors.

As in the case of plate elements, if you include midside nodes for the solid elements, you should locate them as close to the center of the edge as possible. Additionally, if midside nodes are desired, you should include all of them.

R-type Elements

A high degree of precision must be maintained when specifying coefficients for MPCs to avoid introduction of unintentional constraints to rigid body motions. You should use rigid elements (e.g., the RBE2, RBAR, etc.) whenever possible because their constraint coefficients are internally calculated to a high precision. Furthermore, these R-type elements require much less user interaction.

14.3 Determining Mesh Density

The mesh density in a finite element model has important implications for both accuracy and cost. Frequently, the minimum number of elements is set by topological considerations, for example, one element per member in a space frame or one element per panel in a stiffened shell structure. Historically, when problem size was more severely limited, it was not uncommon to lump two or more frames or other similar elements in order to reduce the size of the model. With computers becoming faster and cheaper, the current trend is to represent all major components individually in the finite element model.

If the minimum topological requirements are easily satisfied, the question remains as to how fine to subdivide the major components. The question is particularly relevant for elastic continua, such as slabs and unreinforced shells. In general, the accuracy of your results increases with the mesh density. The mesh density required can be a function of many factors. Among them are the stress gradients, the type of loadings, the boundary conditions, the element types used, the element shapes, and the degree of accuracy desired.

The grid point spacing should typically be the smallest in regions where you expect stress gradients to be the steepest. **Figure 14-6** shows a typical example of a stress concentration near a circular hole. The model is a circular disk with an inner radius = a and an outer radius = b . A pressure load p_i is applied to the inner surface. Due to symmetry, only half of the disk is modeled. In the example, both the radial stress and the circumferential stress decrease as a function of $1/r^2$ from the center of the hole. The error in the finite element analysis arises from differences between the real stress distribution and the stress distribution within the finite elements.

Three different mesh densities are used in this example as shown in **Figure 14-6**.

- The first one is a coarse mesh model with the elements evenly distributed.
- The second model consists of the same number of elements; however, the mesh is biased toward the center of the hole.
- The third model consists of a denser mesh with the elements evenly distributed.

These three models are then analyzed with three different element types-CQUAD4, CQUAD8, and CQUAD4 with the corner stress option. The circumferential stress at the inner radius is always greater than p_i , which is the applied pressure load at the inner radius, and approaches this value as the outer radius becomes larger. The theoretical circumferential stress σ_θ (see Timoshenko and Goodier, *Theory of Elasticity*, McGraw-Hill Book Company, Third Ed., 1970) is given by the following equation:

$$\sigma_\theta = \frac{p_i a^2 \left(1 + \frac{b^2}{r^2} \right)}{(b^2 - a^2)}$$

where:

- a = inner radius
- b = outer radius
- r = radial distance as measured from the center of the disk
- p_i = pressure applied at the inner radius

The stresses are then plotted as a function of the radius in a nondimensional fashion-stress p_i versus r/a . The results are summarized in **Table 14-1** and **Figure 14-7**.

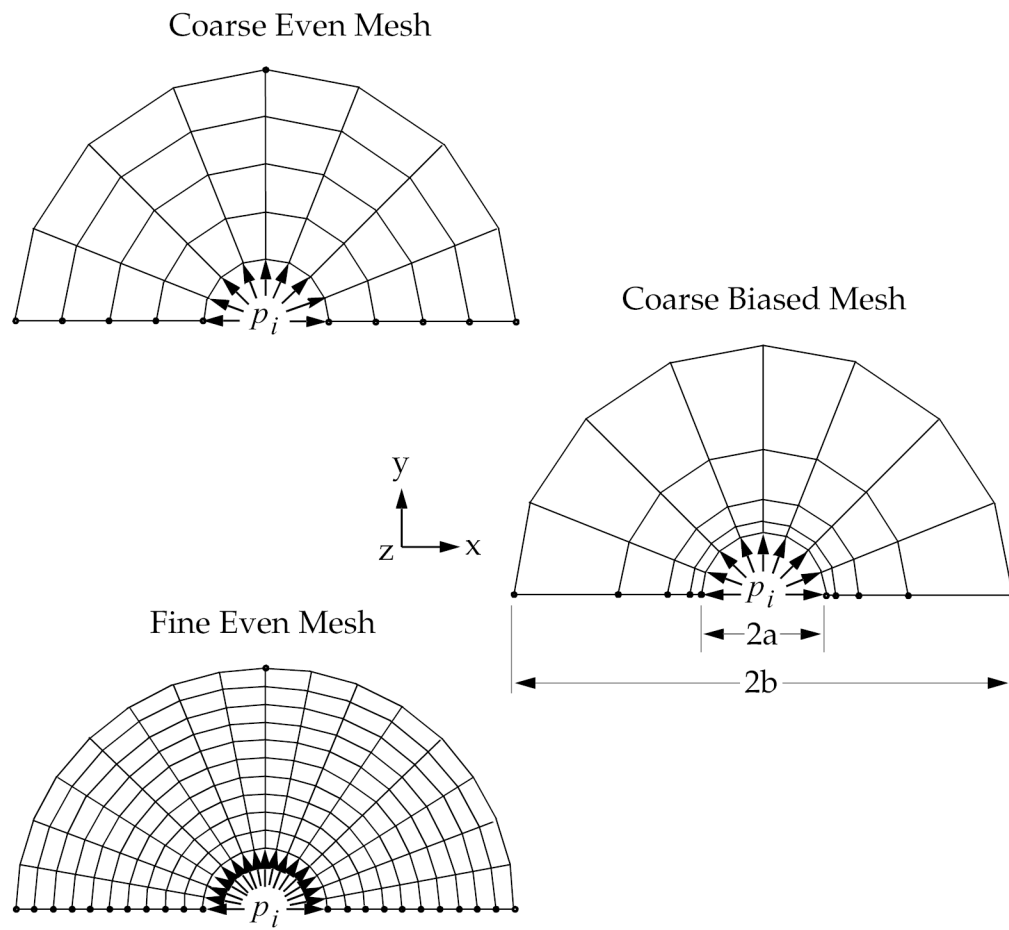


Figure 14-6. Circular Disk with Different Meshes

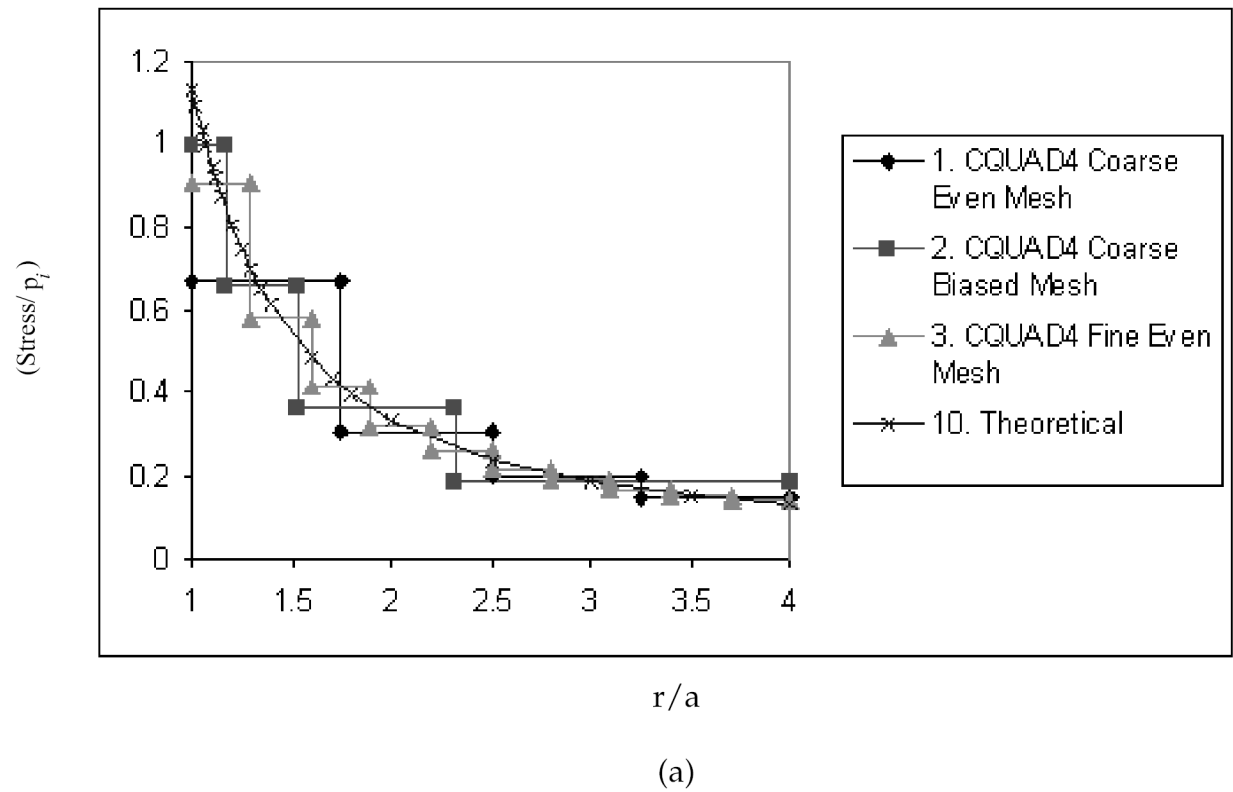


Figure 14-7. Stresses with Different Elements and Meshes

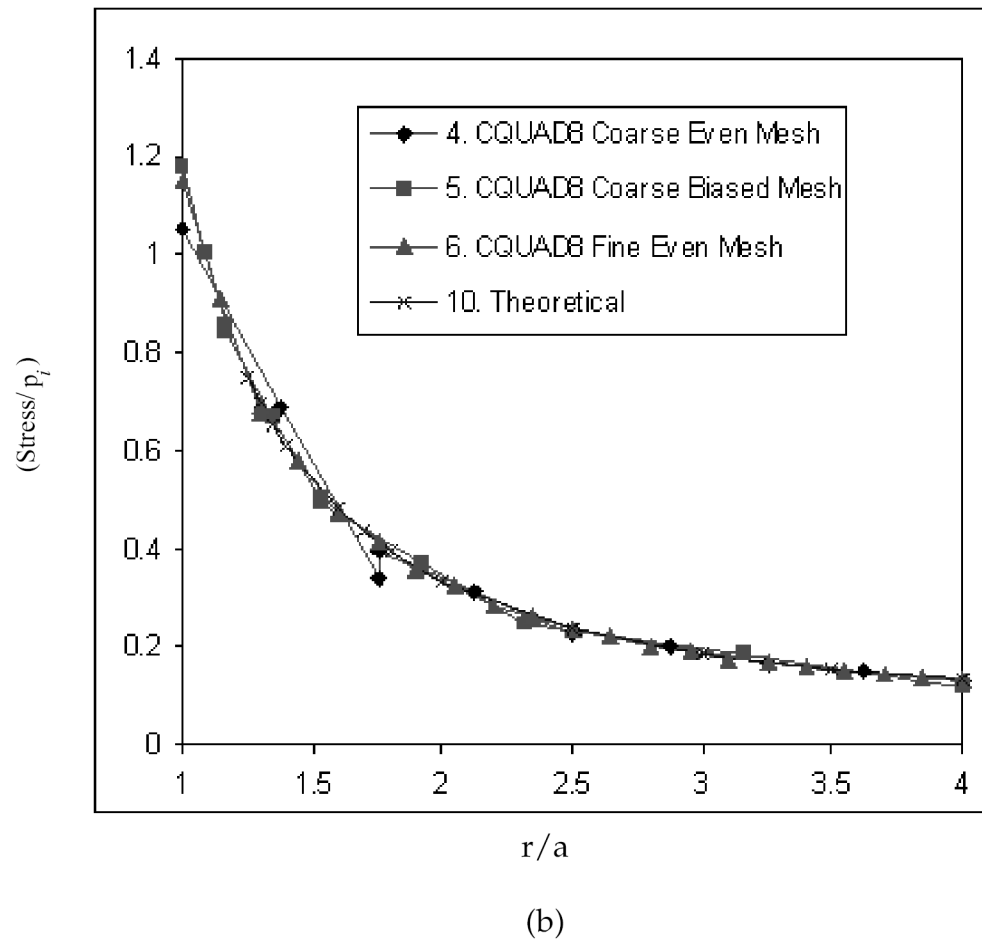


Figure 14-8. Figure 9-7 Stresses with Different Elements and Meshes (continued)

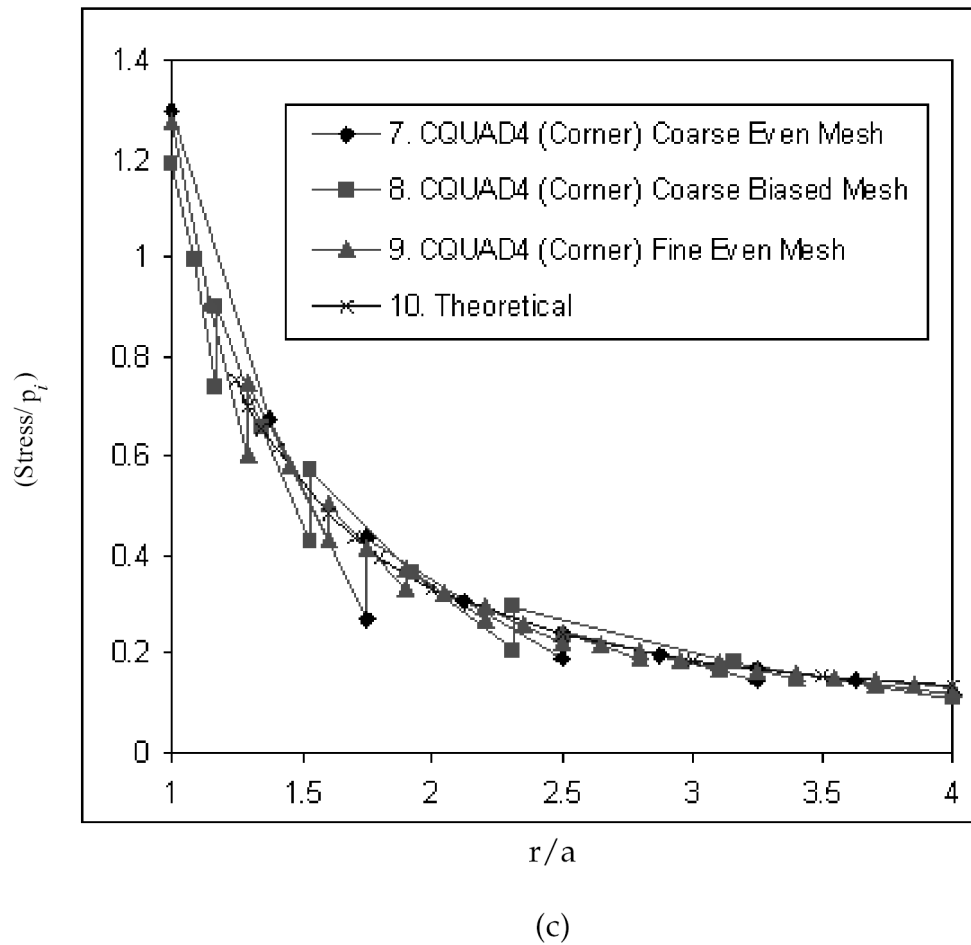


Figure 14-9. Figure 9-7 Stresses with Different Elements and Meshes (continued)

Table 14-1. Stresses Close to $r = a$ for a Circular Disk				
Case No.	Element Type	DOF(L-Set)	Description	Stress Theoretical
1	CQUAD4	194	Coarse, Even Mesh	0.594
2	CQUAD4	194	Coarse, Biased Mesh	0.881
3	CQUAD4	868	Fine, Even Mesh	0.801
4	CQUAD8	550	Coarse, Even Mesh	0.929
5	CQUAD8	550	Coarse, Biased Mesh	1.041
6	CQUAD8	2538	Fine, Even Mesh	1.015
7	CQUAD4 with Corner Option	194	Coarse, Even Mesh	1.142
8	CQUAD4 with Corner Option	194	Coarse, Biased Mesh	1.047
9	CQUAD4 with Corner Option	868	Fine, Even Mesh	1.127
10	THEORETICAL	—	—	1.000

For this particular case, since the stresses are proportional to $1/r^2$, you expect the highest stress to occur at the inner radius. In order to take advantage of this piece of information, the obvious thing to do is to create a finer mesh around the inner radius. Looking at the results for the first two cases in [Table 14-1](#), it is quite obvious that just by biasing the mesh, the results are 30% closer to the theoretical solution with the same number of degrees of freedom.

A third case is analyzed with a finer but unbiased mesh. Note that for case number 3, even though it has more degrees of freedom, the result is still not as good as that of case number 2. This poor result is due to the fact that for the CQUAD4 element, the stresses, by default, are calculated at the center of the element and are assumed to be constant throughout the element. This assumption is best illustrated by the results shown in [Figure 14-7\(a\)](#). Looking at [Figure 14-6](#), it is also obvious that the centers of the inner row of elements are actually further away from the center of the circle for case number 3 as compared to case number 2. The results for case number 3 can, of course, be improved drastically by biasing the mesh.

You can request corner outputs (stress, strain, and force) for CQUAD4 in addition to the center values. Corner results are extrapolated from the corner displacements and rotations by using a strain rosette analogy with a cubic correction for bending. The same three models are then rerun with this corner option—their results are summarized in cases 7 through 9. Note that the results can improve substantially for the same number degrees of freedom.

You select corner output by using a corner output option with the STRESS, STRAIN, and FORCE Case Control commands. When you select one of these options, output is computed at the center and four corners for each CQUAD4 element, in a format similar to that of CQUAD8 and CQUADR elements.

There are four corner output options available:

- CORNER
- CUBIC
- SGAGE
- BILIN

These different options provide for different approaches to the stress calculations. The default option is CORNER, which is equivalent to BILIN. BILIN has been shown to produce better results for a wider range of problems.

To carry it a step further, the same three models are then rerun with CQUAD8 (cases 4 through 6). In this case, the results using CQUAD8 are better than those using the CQUAD4. This result is expected since CQUAD8 contains more DOFs per element than CQUAD4. Looking at column three of [Table 14-1](#), you can see that due to the existence of midside nodes, the models using CQUAD8 contain several times the number of DOFs as compared to CQUAD4 for the same number of elements. The results using CQUAD4 can, of course, be improved by increasing the mesh density to approach that of the CQUAD8 in terms of number of DOFs.

It is important to realize that the stresses are compared at different locations for Cases 1 through 3 versus Cases 4 through 9. This difference occurs because the stresses are available only at the element centers for Cases 1 through 3, but the stresses are available at the corners as well as the element centers for Cases 4 through 9. When looking at your results using a stress contour plot, you should be aware of where the stresses are being evaluated.

How fine a mesh you want depends on many factors. Among them is the cost you are willing to pay versus the accuracy you are receiving. The cost increases with the number of DOFs. The definition of cost has changed with time. In the past, cost is generally associated with computer time. With both hardware and software becoming faster each day, cost is probably associated more with the time required for you to debug and interpret your results. In general, the larger the model is, the more time it takes you to debug and interpret your results. As for acceptable accuracy, proceeding from case 8 to case 6, the error is reduced from 4.7% to 1.5%; however, the size of the problem is also increased from 194 to 2538 DOFs. In some cases, a 4.7% error may be acceptable. For example, in cases in which you are certain of the loads to within only a 10% accuracy, a 4.7% error may be acceptable. In other cases, a 1.5% error may not be acceptable.

In general, if you can visualize the form of the solution beforehand, you can then bias the grid point distribution. However, this type of information is not necessarily available in all cases. If a better assessment of accuracy is required and resources are available (time and money), you can always establish error bounds for a particular problem by constructing and analyzing multiple mesh spacings of the same model and observe the convergences. This approach, however, may not be realistic due to the time constraint.

You can use the stress discontinuity feature, as described in “**Model Verification**” in the *NX Nastran User's Guide*, to assess the quality of the mesh density for the conventional h-version elements.

With the software's p-version element and p-version adaptivity capabilities, you can perform the mesh refinement in NX Nastran.

See also

- “**CQUAD4 and CTRIA3 Elements**” in the *NX Nastran Element Library*

h-elements

In traditional finite element analysis, as the number of elements increases, the accuracy of the solution improves. The accuracy of the problem can be measured quantitatively with various entities, such as strain energies, displacements, and stresses, as well as in various error estimation methods, such as simple mathematical norm or root-mean-square methods. The goal is to perform an accurate prediction on the behavior of your actual model by using these error analysis methods. You can modify a series of finite element analyses either manually or automatically by reducing the size and increasing the number of elements, which is the usual h-adaptivity method. Each element is formulated mathematically with a certain predetermined order of shape functions. This polynomial order does not change in the h-adaptivity method. The elements associated with this type of capability are called the h-elements.

p-elements

A different method used to modify the subsequent finite element analyses on the same problem is to increase the polynomial order in each element while maintaining the original finite element size and mesh. The increase of the interpolation order is internal, and the solution stops automatically once a specified error tolerance is satisfied. This method is known as the p-adaptivity method. The elements associated with this capability are called the p-elements.

See also

- “**p-Elements**” in the *NX Nastran User's Guide*

14.4 Creating Mesh Transitions

You can use mesh transitions, for example, to refine the mesh in a particular area, connect different element types (for example, a CBAR element to a solid element), or provide transitions required to model the geometry of the structure.

- Never place a mesh transition in an area of interest or in an area where there is a large variation in stress.
- Mesh transitions should be located away from the areas of interest in a region.

Due to incompatibilities between finite element types, any transition between different element types (even a transition from CQUAD4 to CTRIA3 elements) can result in local stress anomalies. Normally, these stress anomalies are localized and dissipate quickly as you move away from the transition. However, a problem arises when the transition occurs in an area of interest. In this case, the local stress rises (or decreases) due to the effect of the transition; in other words, the results may be conservative (or unconservative) in an area near a transition. However, if this localized stress variation occurs away from areas of interest, the increase (or decrease) in stress caused by the transition should cause no concern.

Transition from a Coarse Mesh to a Fine Mesh

Creating a transition between coarse and fine meshes can be challenging. One common method of performing a transition is to use an intermediate belt of triangular elements as shown in [Figure 14-10\(a\)](#). It is also very tempting to selectively delete midside nodes of higher-order elements (e.g., CQUAD8) and use them as transition elements as shown in [Figure 14-10\(b\)](#). However, this method isn't recommended because it severely distorts the stress distribution in the elements adjacent to the change in mesh size.

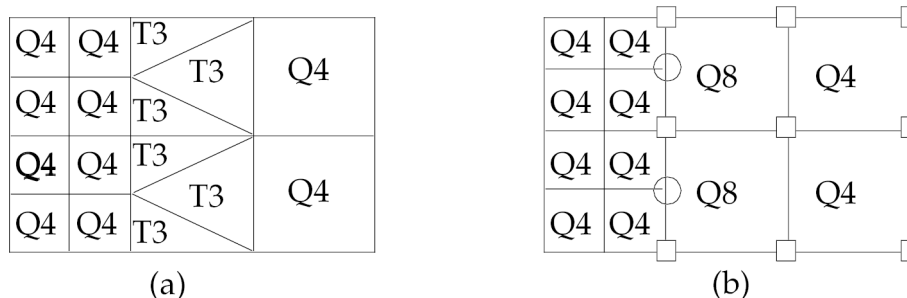


Figure 14-10. Transition Methods

When using a higher-order element, such as a CQUAD8 or a CHEXA (nine to 20 nodes) for any purpose, you should include all midside nodes. A CQUAD8 element with all midside nodes deleted is excessively stiff and therefore is inferior to a CQUAD4. For the CHEXA element, you should use it with eight nodes (no midside nodes), or use it with 20 nodes (all midside nodes). The midside nodes, if used, should be located as close to the center of the edge as possible.

As an alternative mesh transition method, a spline element can be used. In NX Nastran, the RSPLINE element is an elastic interpolation element that connects N end points. Displacements for intermediate grid points are interpolated from the equations of an elastic beam passing through these grid points. The spline element assumes a linear interpolation for displacement and torsion along the

axis of the spline, a quadratic interpolation for rotations normal to the axis of the spline, and a cubic interpolation for displacements normal to the axis of the spline.

Consider an example problem as shown in [Figure 14-11](#). This problem involves a plate fixed at the left end with a dimension of 14 in by 9 in, and modeled with 1 in by 1 in CQUAD4 elements. For clarity, only grid points of interest are labeled. Five different loadings—as shown in [Table 14-2](#)—are applied to this structure along the free edge (right end). For convenience, a RBE3 element is created along the free edge by defining a dependent point (grid point 500) at the center. This grid point is connected to all the edge grid points. The load can then be applied to grid point 500 and distributed to the rest of the edge grid points.

Hypothetically, suppose that you're interested in the stresses along the line one inch from the fixed edge. Since you're not interested in the stresses outside of this region, you can represent the rest of the region with a coarser mesh. Two different transition methods will be used for this purpose—one with CTRIA3s ([Figure 14-12](#)) and one with RSPLINEs ([Figure 14-13](#)). The transitions are performed along the line five inches away from the fixed edge. An additional transition is performed at $x = 11$ inches in order to maintain the same grid point density on the right edge for all three models. This condition assures identical load applications for all three models. Normally, this additional transition is seldom done in a real model.

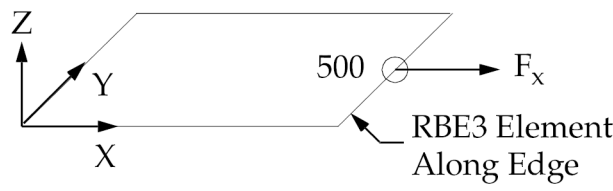
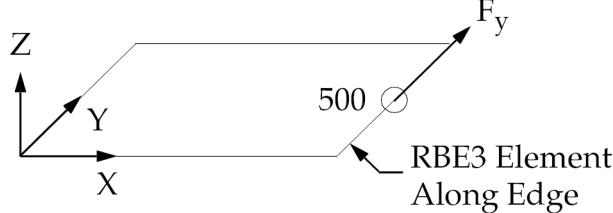
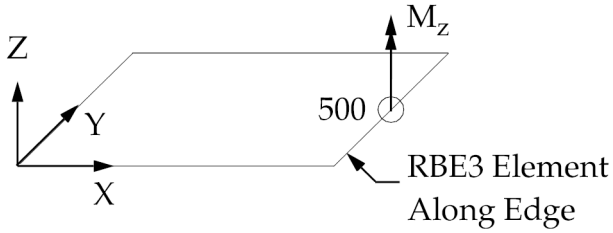
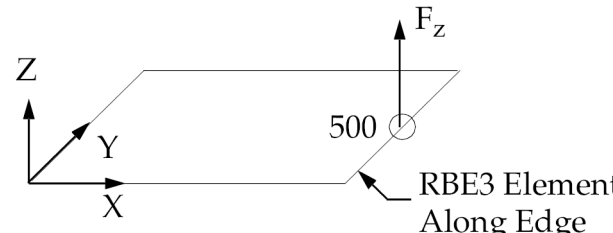
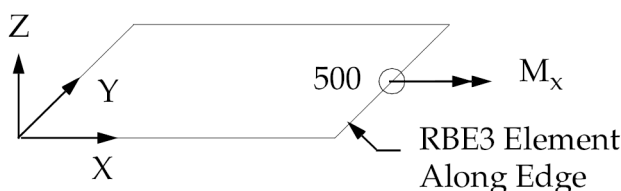
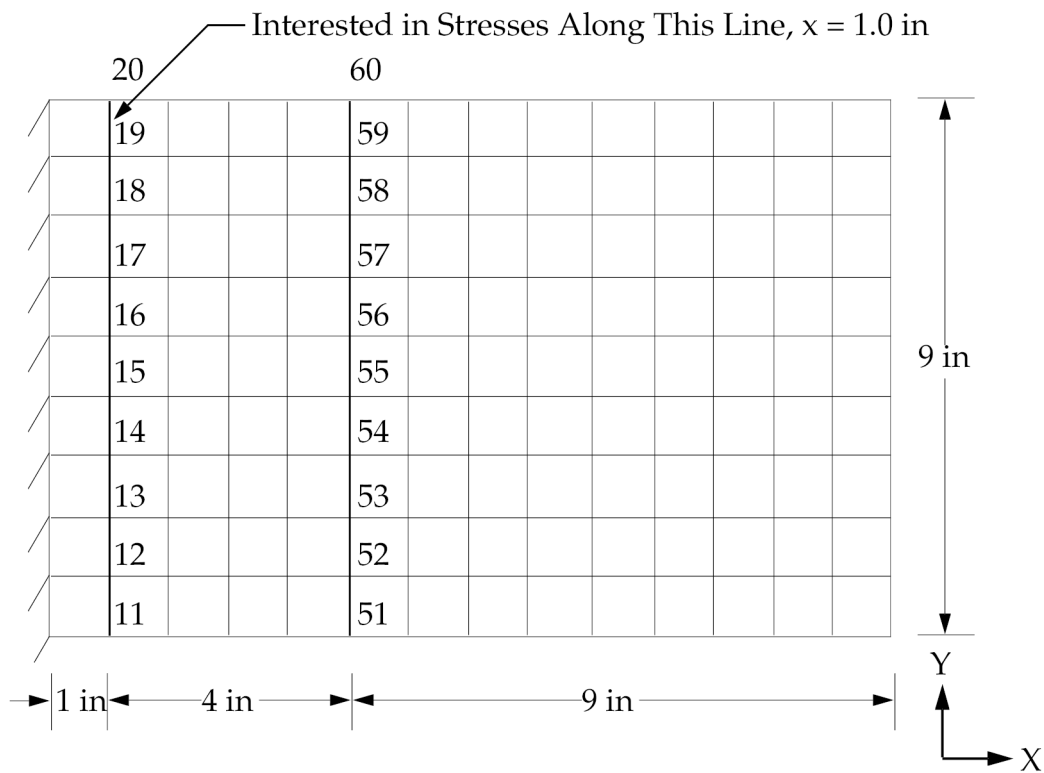
Table 14-2. Loadings Used for the Mesh Transition Test	
Loading Description	
1. Membrane Loads	
2. In-Plane Shear Loads (parabolic distribution)	
3. In-Plane Bending Loads	
4. Vertical Loads	

Table 14-2. Loadings Used for the Mesh Transition Test

Loading Description	
5. Twisting Loads	 <p>500 M_x RBE3 Element Along Edge</p>

**Figure 14-11. Baseline Model**

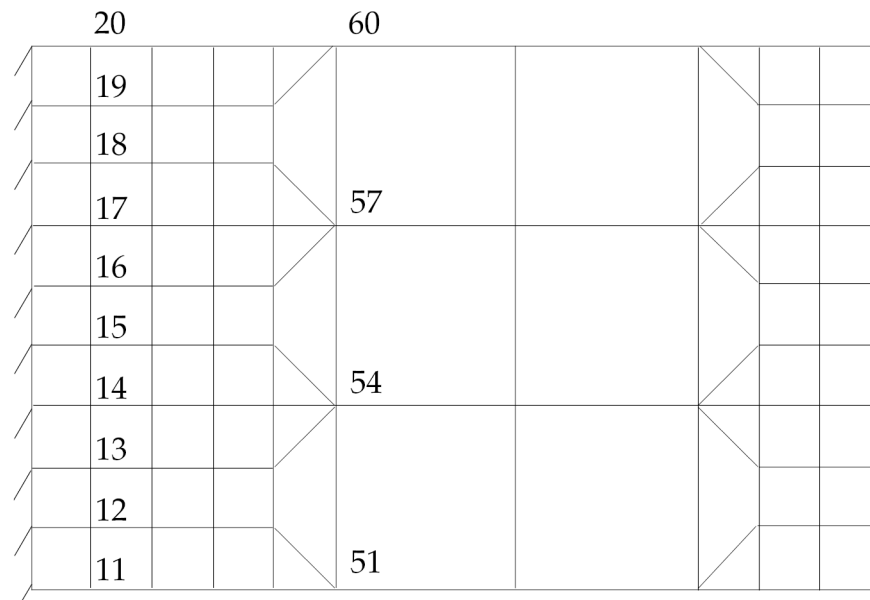


Figure 14-12. Transition with CTRIA3

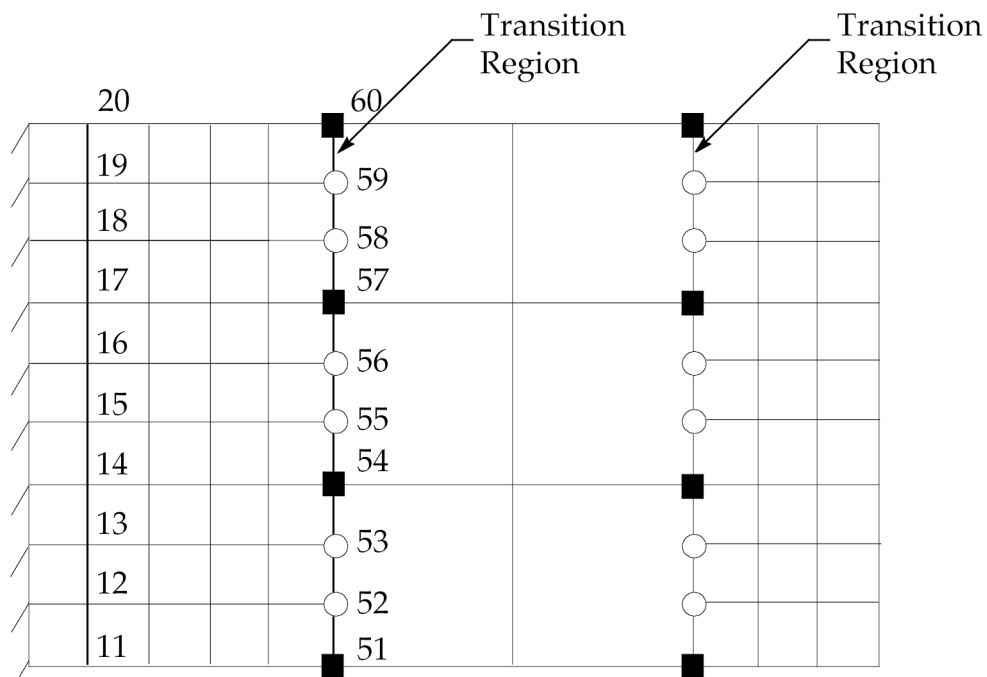


Figure 14-13. Transition with RSPLINE

The model with the mesh transition using CTRIA3s, as shown in [Figure 14-12](#), is quite straightforward and needs no further explanation. However, further discussion is warranted in the case of mesh transition using the RSPLINE element.

The input format for the RSPLINE element is as follows:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

Field	Contents
EID	Element identification number.
Gi	Grid point identification number.
Ci	Components to be constrained.

See also

- “[The RBE3 Element](#)” in the *NX Nastran Element Library*

Remarks

1. The first and last grid points must be independent.
2. Displacements are interpolated from the equations of an elastic beam passing through the grid points.

A single RSPLINE is used to connect grid points 51 through 60 as shown in [Figure 14-13](#). Grid points 51, 54, 57, and 60 are considered to be independent, while the other six points are dependent (grid points 52, 53, 55, 56, 58, and 59). This RSPLINE interpolates the motions at the intermediate points, indicated by circles in [Figure 14-13](#) (grid points 52, 53, 55, 56, 58, and 59), from the cubic equations of a beam spline passing through the points indicated by squares (grid points 51, 54, 57, and 60). The input for the RSPLINE element connecting grid points 51 through 60 is as follows:

1	2	3	4	5	6	7	8	9	10
RSPLINE	1000		51	52	12345	53	12345	54	
		55	12345	56	12345	57		58	
	12345	59	12345	60					

The results using these two transition methods are compared with the baseline model with no transition. These results are evaluated at two locations: at $x = 1.0$ in (location of interest) and at $x = 5.0$ in (transition region). The five different loading conditions mentioned previously were used. The Hencky-von Mises stresses using the grid point stress feature (“[Reviewing Grid Point Stresses](#)”) and the CQUAD4 corner stress option are used for comparison. The results are shown in [Figure 14-14](#) through [Figure 14-18](#). As you can see, the results compare well, using either method of transition, with the baseline model at the area of interest (away from the transition region) for all five loading conditions.

At the transition region, depending on the loading and location, the transition using CTRIA3 elements compares better with the baseline model in some cases, whereas the transition using RSPLINE elements compares better with the baseline model in other cases. Note that there are only four stress values available in the case of using CTRIA3s since there are only four grid points along the line joining grid points 51 through 60 in [Figure 14-12](#).

The CTRIA3 elements used in this model are all regularly shaped elements; in most real life problems, it is very likely that the transition areas may consist of CTRIA3 elements that are more distorted. In

this case, the transition using the distorted CTRIA3 elements does not perform as well as the model in **Figure 14-11** consisting of regularly shaped CTRIA3 elements.

In either case, there are noticeable differences in the stresses at the transition region, as expected. However, at the area of interest and away from the transition region, the stresses are virtually the same for both cases as compared to the baseline model. Once again, mesh transitions should always be modeled away from the area of interest.

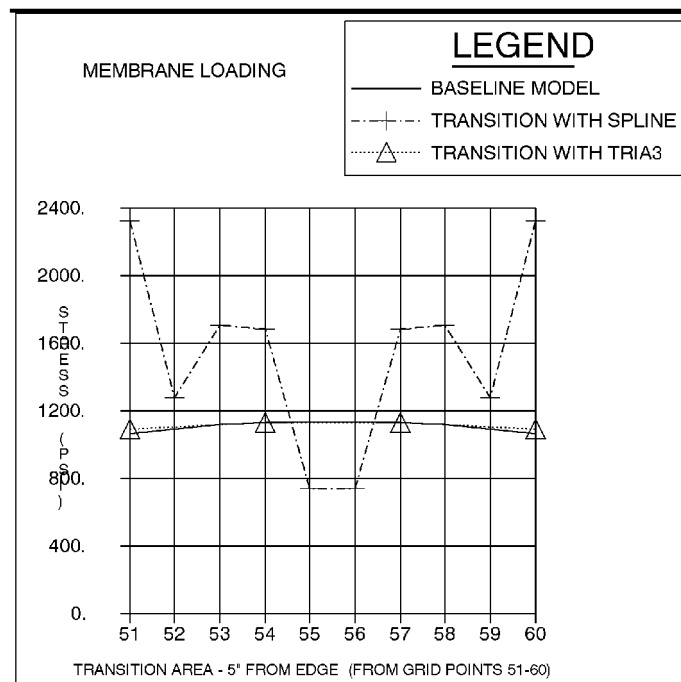
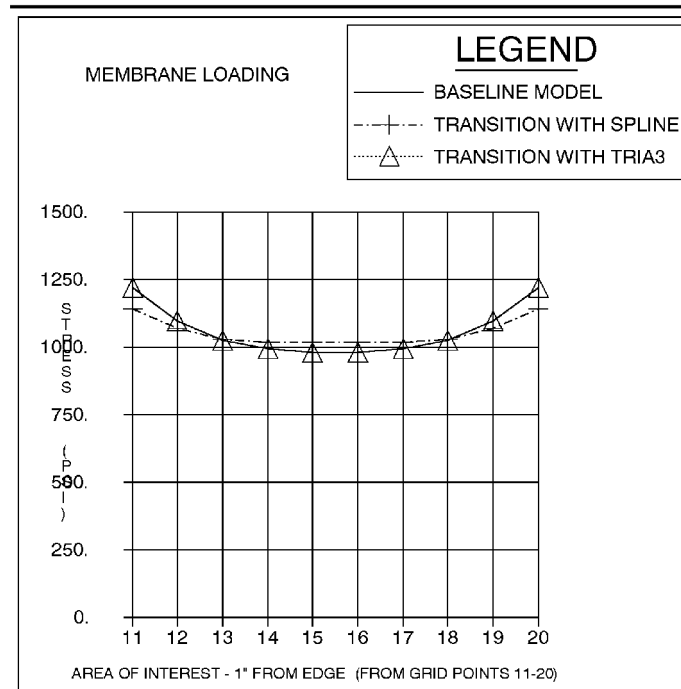


Figure 14-14. Resultant Stresses for Membrane Loads

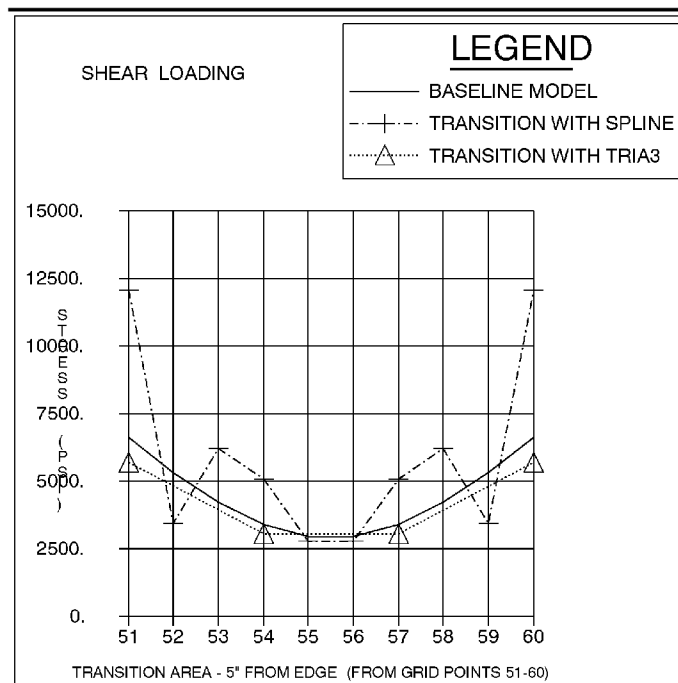
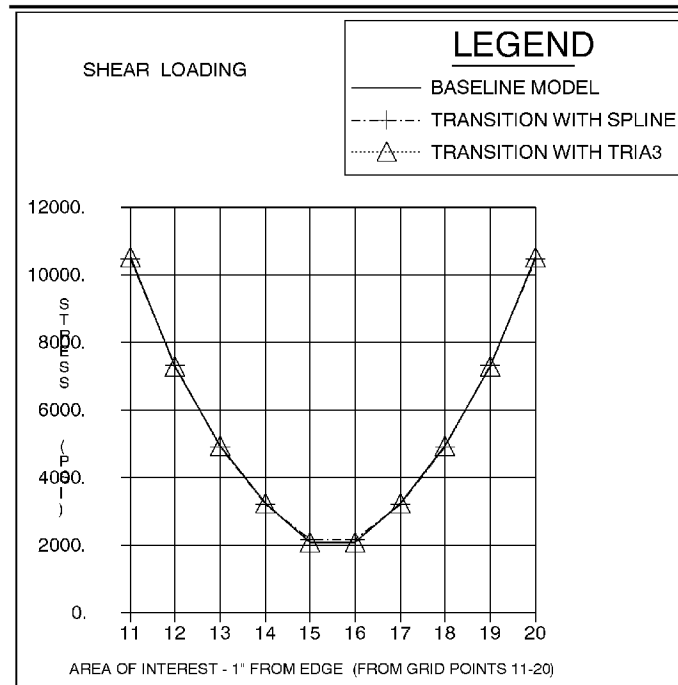


Figure 14-15. Resultant Stresses for In-Plane Shear Loads

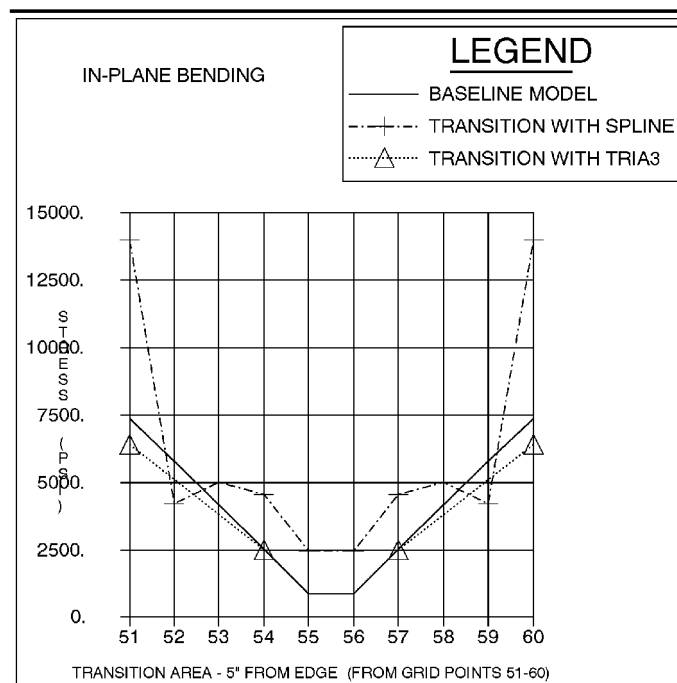
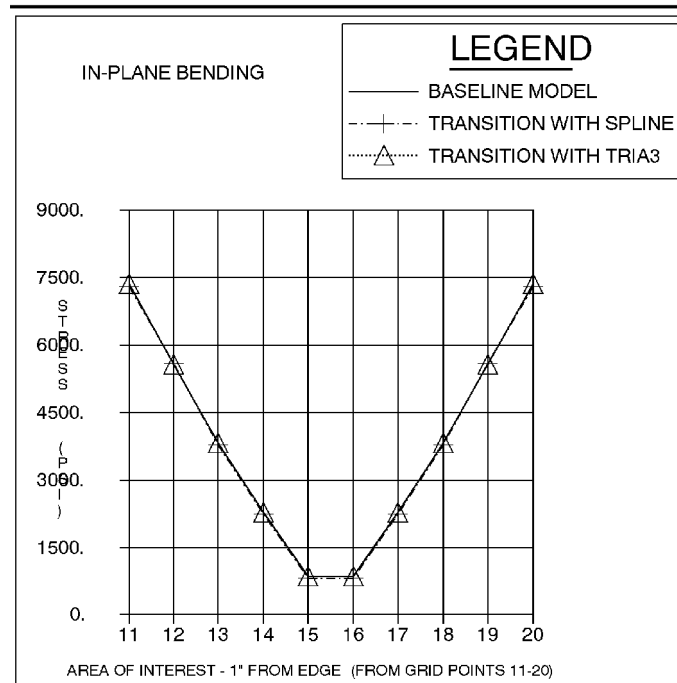


Figure 14-16. Resultant Stresses for In-Plane Bending Loads

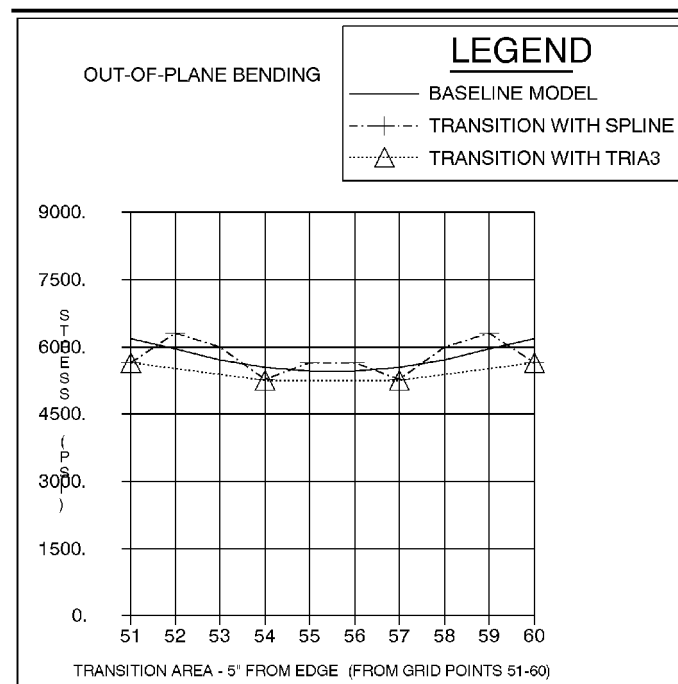
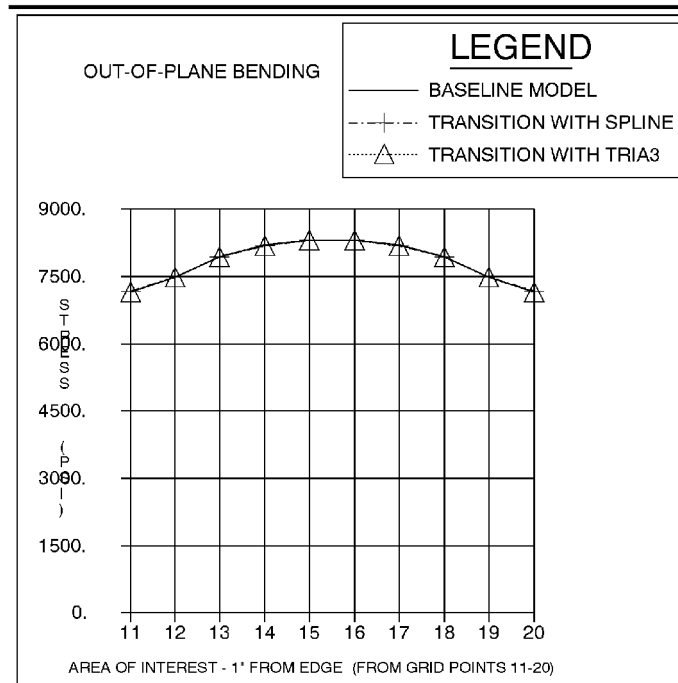


Figure 14-17. Resultant Stresses for Vertical Loads

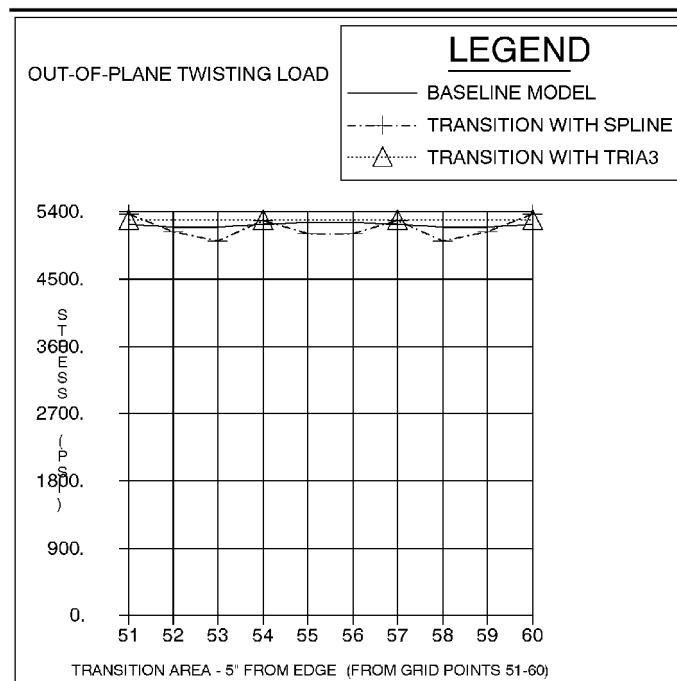
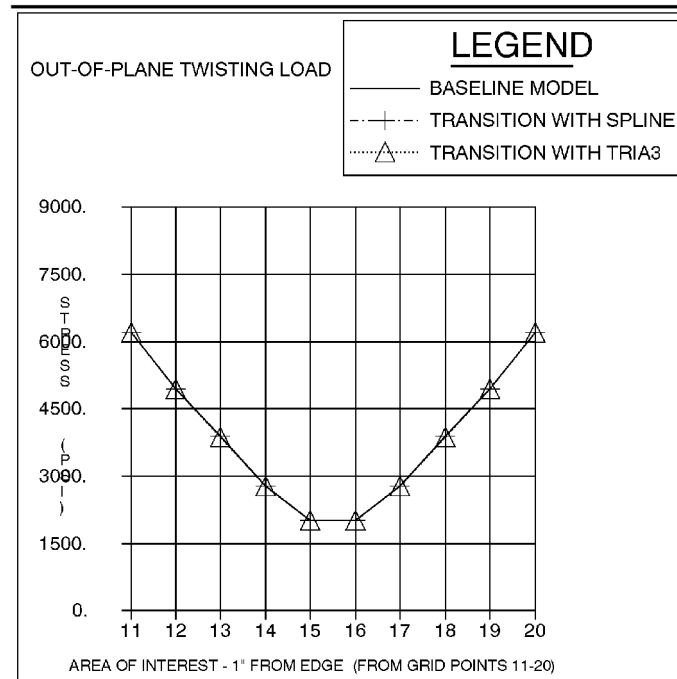


Figure 14-18. Resultant Stresses for Twisting Loads

Mesh Transition Between Dissimilar Element Types

Attaching a plate or bar element to a solid element is a case of transition between dissimilar element types. This process is more involved than it appears at first glance. Solid elements have stiffness

only in the translational DOFs at the attachment grid points; they have no stiffness for rotational DOFs. A simple visualization is to think of the attachment of a solid element to a grid point as a “ball-and-socket” joint, that is, translational forces may be transmitted, but no moment may be transmitted through the connection.

This incompatibility of the element stiffness matrices represents a modeling problem whenever plate or bar elements are attached to solid elements. Both plate and bar elements have stiffness for rotational DOFs (although the plate element may not have a stiffness for the normal rotation). Therefore, special modeling must be performed whenever a plate or bar is connected to a solid element. Otherwise, the connection becomes a hinge (for plate elements) or a pinned connection (for bar elements).

Several methods are available to handle the transition between these elements. These methods range from adding extra elements (for example, adding an additional plate or bar that continues into the solid element) to using special (R-type) elements for the transition.

One method of handling this transition is to use RBE3 elements. The RBE3 is an interpolation element, which is ideally suited for this application. By using RBE3s, the rotations of the attached grid points is simply slaved to the translations of the adjacent grid points.

Examples of using RBE3 elements to connect a solid element to a plate element and bar element are shown in **Figure 14-19**. The RBE3 elements attach the rotational DOFs to the translational DOFs on the solid element.

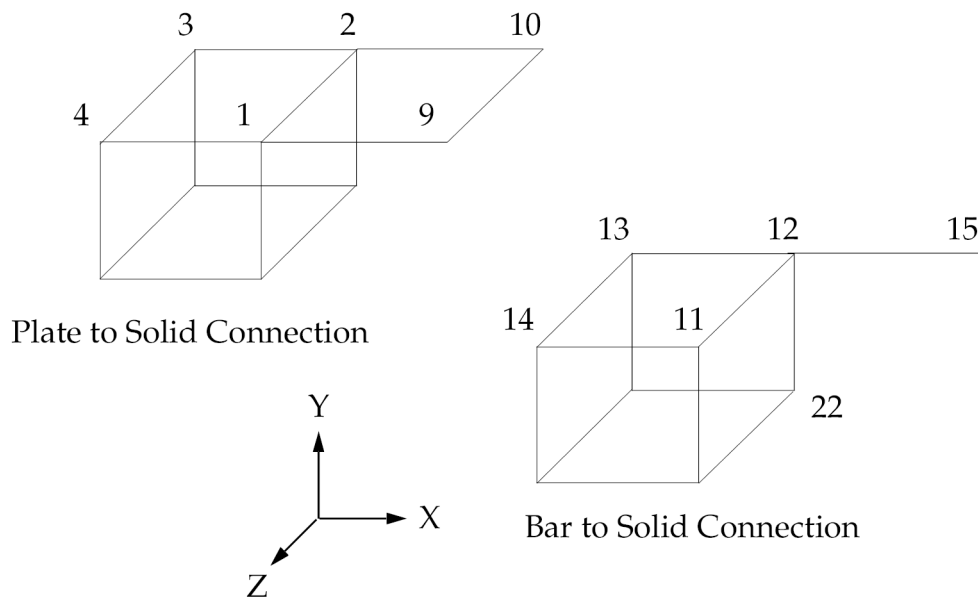


Figure 14-19. Typical Transition Between Dissimilar Elements

For the plate to solid connection, two RBE3 elements suffice:

1	2	3	4	5	6	7	8	9	10
\$RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	901		1	456	1.0	123	2	3	
	4								

RBE3	902		2	456	1.0	123	1	3	
	4								

For the bar to solid connection, one RBE3 element can make the connection:

RBE3	903		12	456	1.0	123	11	13	
	22								

These RBE3 elements transmit the loads to the independent DOFs. If RBE2 elements are used, then the connection is “rigid.”

Remember, when handling these connections, the solid elements have no stiffness for rotational DOFs, whereas the real structure does. This means that a special modeling effort is needed when any element with bending stiffness is connected to a solid element.

When using the RBE3 element, care must be taken to ensure that the independent DOFs are sufficient to transfer any applied loads. For the bar-to-solid connection in [Figure 14-18](#), if only two independent grid points are used, the element is “unstable,” that is, since only the translational DOFs are used as independent DOFs in the sample, the element is unstable for rotation about the axis connecting the two points. Therefore, three non-colinear grid points are used. A simple way to remember this is to ask, “If I constrain the DOFs that I list as independent on the RBE3, can I prevent any possible rigid body motion?” If the answer to this question is “yes,” then the RBE3 element is capable of transferring any applied loads. In this way, you can avoid possible problems in processing the RBE3 elements.

See also

- [“The RBE3 Element”](#) in the *NX Nastran Element Library*

Shell-to-solid Transition Element (RSSCON)

The RSSCON method is another way to model shell-to-solid transitions (see [Figure 14-20](#)). This capability conveniently eliminates the need to define RBE3s or MPCs (multipoint constraints) to constrain a shell element's translational and rotational degrees of freedom to a solid element's translational degrees of freedom. When using the RSSCON capability, the shell element mesh must line up with the solid element mesh so that there is an exact element-to-element correspondence.

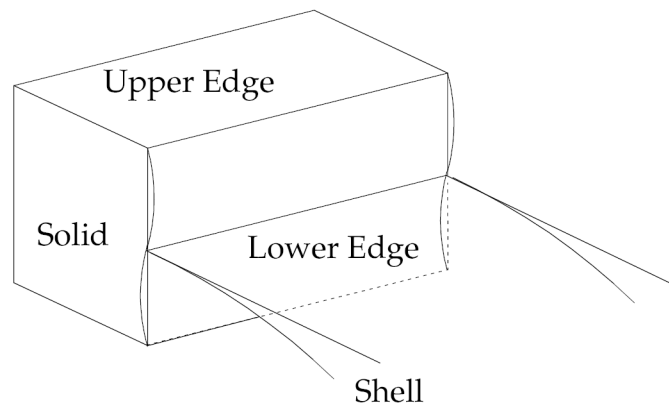


Figure 14-20. Clamped Connection of a Shell to a Solid

The RSSCON Bulk Data entry supports both p-adaptive elements and h-elements. The p-adaptive elements are outside the scope of this user's guide. Through the RSSCON Bulk Data entry, the shell elements CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR can be connected to the solid elements CHEXA, CPENTA, CPYRAM, and CTETRA. Elements with midside nodes are also supported.

RSSCON generates a multipoint constraint, which puts the shell degrees of freedom in the dependent set (m-set). The three translational degrees of freedom and the two rotational degrees of freedom of the shell edge are connected to the three translational degrees of freedom of the upper and lower solid edge. Poisson's ratio effects and temperature loads are modeled correctly. The generated multipoint constraints produce six zero-energy modes for rigid-body motion.

The RSSCON Bulk Data entry defines the connection of a shell element to a solid element. Within NX Nastran, however, there are two options for making this connection using the RSSCON Bulk Data entry, as shown in [Figure 14-21](#):

1. The ELEM option, which allows you to specify the element ID of the shell and the element ID of the solid to which the shell is to be connected.
2. The GRID option, which allows you to specify the grid point ID of the shell and the upper and lower grid point IDs of the solid. Two triplets of grid point IDs may be specified on one RSSCON Bulk Data entry.

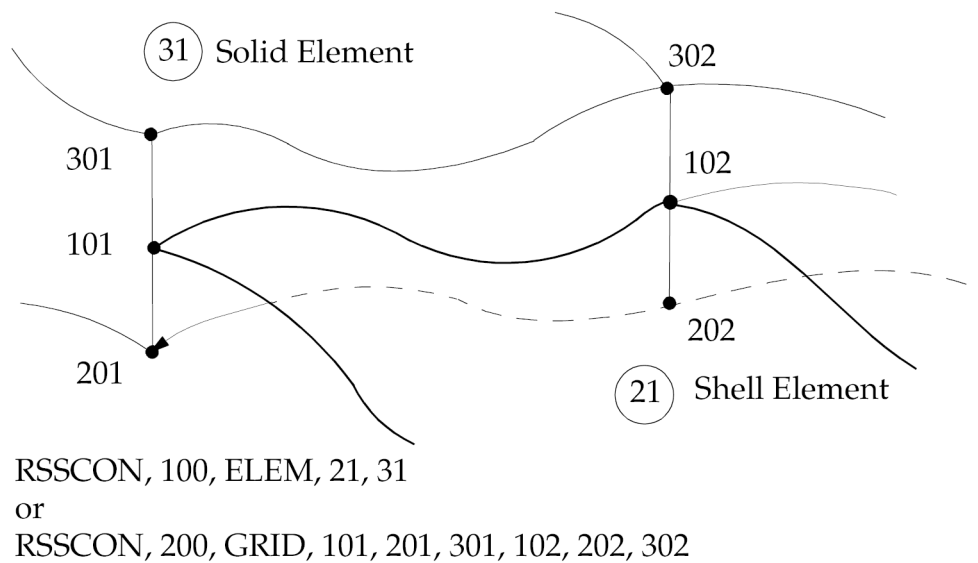


Figure 14-21. ELEM and GRID Option on the RSSCON Bulk Data Entry

The best modeling practice is illustrated in [Figure 14-22](#). The height of the connected solid element should be chosen equal to the thickness of the shell. If the height of the connected solid element is much larger than the thickness of the shell element, then the connection modeled with RSSCON will be stiffer than the continuum model. For example, in a mesh where shell grid points are identical or coincide with solid grid points, the RSSCON Bulk Data entry may model a connection that is too stiff (see [Figure 14-23\[a\]](#)). Note also that in the model of [Figure 14-23\[b\]](#), the GRID option must be used to connect a shell element to more than one solid element.

The RSSCON connector element can accommodate a variety of mesh topologies, as shown in [Figure 14-24](#).

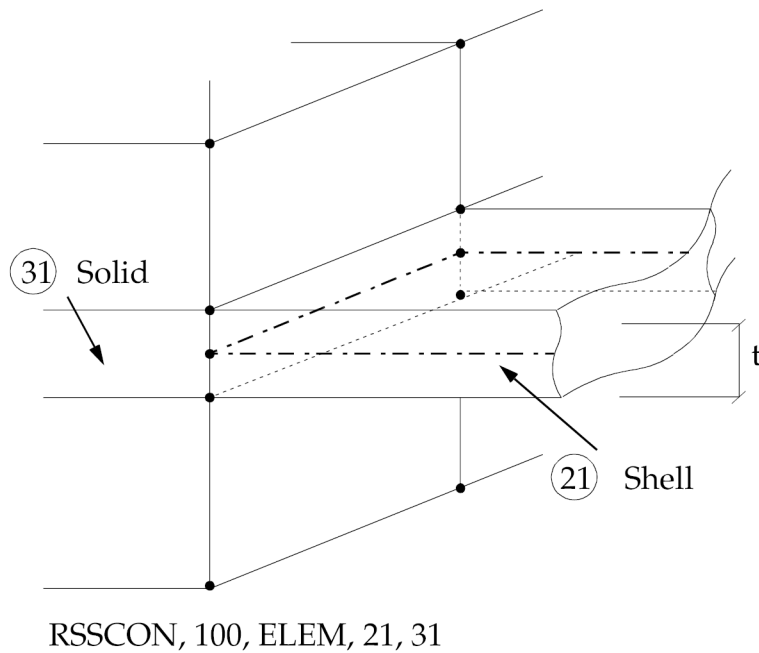


Figure 14-22. Best Modeling Practice for RSSCON

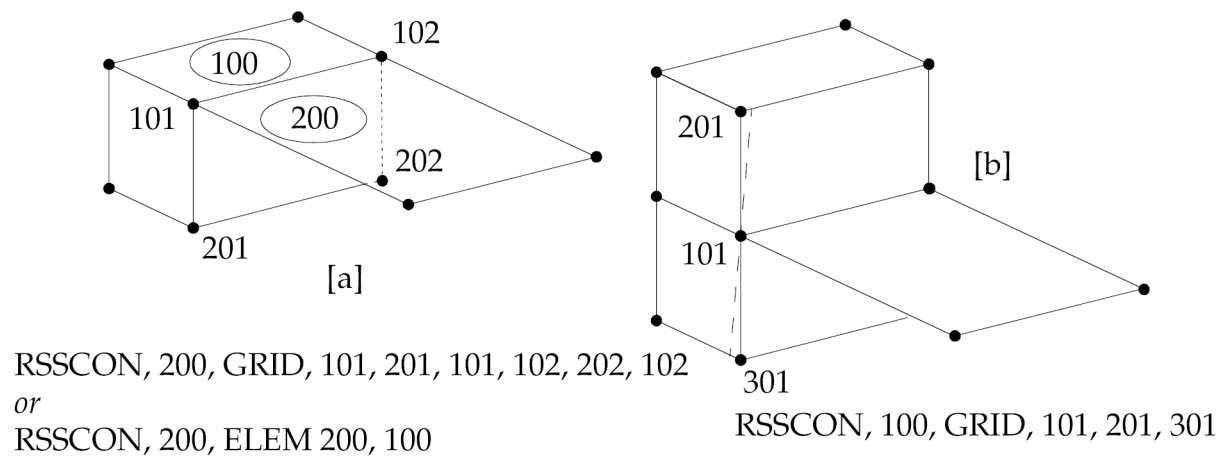
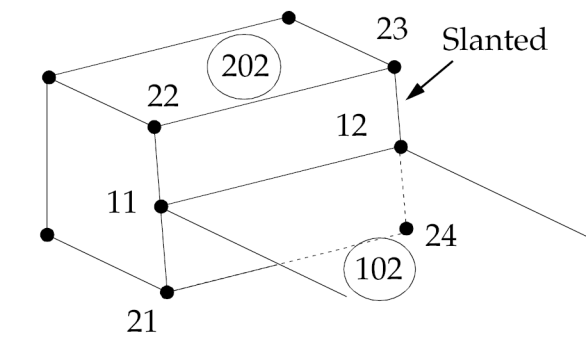


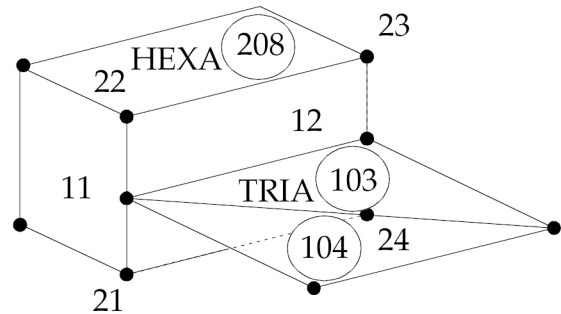
Figure 14-23. RSSCON in Meshes Where Solid and Shell Elements Share Grid Points



RSSCON, 100, ELEM, 102, 202

or

RSSCON, 100, GRID, 11, 21, 22, 12, 24, 23

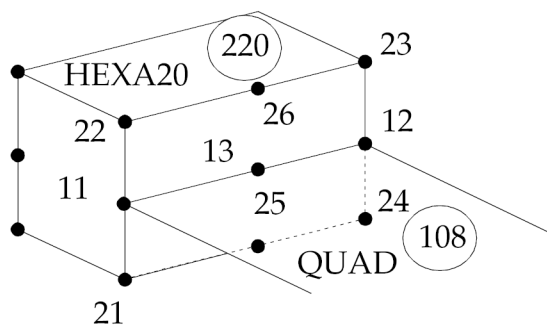


RSSCON, 200, ELEM, 103, 208

RSSCON, 201, ELEM, 104, 208 (not necessary)

or

RSSCON, 200, GRID, 11, 21, 22, 12, 24, 23

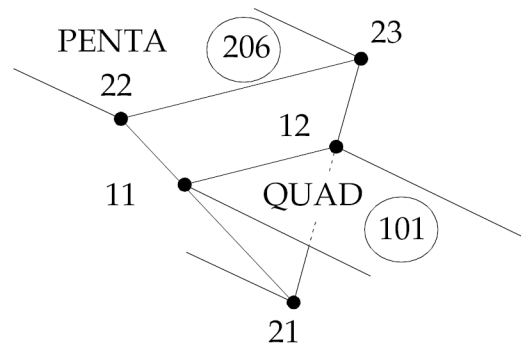


RSSCON, 200, ELEM, 108, 220

or

RSSCON, 200, GRID, 11, 21, 22, 12, 24, 23

RSSCON, 201, GRID, 13, 25, 26



RSSCON, 200, ELEM, 101, 206

or

RSSCON, 200, GRID, 11, 21, 22, 12, 21, 23

Figure 14-24. Modeling Options with RSSCON

The only modeling requirement is that the shell grid point must lie on the line connecting the lower and upper solid grid points. By default, the RSSCON allows the shell grid point to have an offset from this line of up to 5% of the distance between the two solid grid points. If a shell grid point does not lie on the line joining the two solid grid points, but within this 5% tolerance, the shell grid point will be moved to lie on the line. On the other hand, if a shell grid point lies outside this 5% tolerance, the run terminates with a fatal message. You can modify the default 5% tolerance by adding "PARAM,TOLRSC, " to the Bulk Data Section, where is the desired tolerance percentage. The default value for is 0.05. You can print the old and new location of the moved shell grid points with the parameter "PARAM,SEP1XOVR,16".

When you use the GRID option, NX Nastran does not verify that the grid points are valid shell or solid grid points. When the ELEM option is used, if a solid-shell connection has horizontal edges that are curved in the shell plane, NX Nastran assumes that the geometry of the shell element is compatible with that of the solid element. In the case of vertical edges, the shell element grid point must lie on the line between the upper and lower solid element grid point. A default offset tolerance of 5% (user modifiable via "param,tolrsc,") of the distance between the solid grid points is allowed. For limitations of the RSSCON related to the p-elements, see:

- **“Introduction to R-Type Elements”** in the *NX Nastran Element Library*

By default, the moved shell grid points aren't printed.

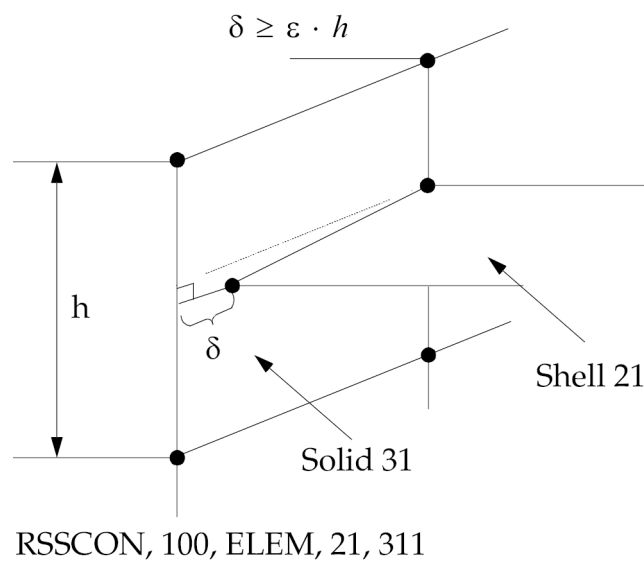


Figure 14-25. RSSCON Modeling Practice Parameters

RSSCON Modeling Recommendations

The RSSCON connector produces excellent results if you follow good modeling practices. The geometry of the RSSCON connector is checked, and fatal messages are issued for invalid connections. Additional recommendations are given below.

1. Midside-noded elements should *only* be connected to other midside-noded elements. For example, a CQUAD8 element connected to a CHEXA20 element is acceptable; however, a CQUAD4 element connected to a PENTA15 element is not allowed. Elements with missing midside grid points are not allowed.
2. For midside-noded elements, avoid using the RSSCON where shells connect to triangular solid faces.
3. Do not mix coordinate systems when using midside nodes. In other words, when using midside nodes, do not define a local coordinate system containing all the shell grid points and the basic coordinate system to define the solid grid points. Otherwise, incorrect answers will be generated.
4. Only one edge of a single shell element may connect to a solid surface.

5. Do *not* connect more than one shell element to one solid element when using the ELEM option.
6. Do not attempt solid-shell connections to severely warped (greater than 20%) midside-noded solid elements.
7. The plane of the shell element should not be parallel to the plane of the connecting face of the solid element.
8. The connecting edge of the shell element must lie in the face of the solid element. In addition, the vertex grid points of the shell edge connecting to a quadrilateral solid face must line up with its opposite edges; for triangular solid faces, the connections may be made between any of the edges. The tolerance in the geometry may be modified; see PARAM,TOLRSC, described above.
9. When using the ELEM option in the case of partitioned superelement Bulk Data, you must ensure that the solid element, RSSCON connector, and shell element which define a shell-solid connection are all contained within the same superelement.

Example - Cantilever Clamped to a Wall

A cantilever (4.1 in long, 0.1 in thick, 0.5 in wide) is clamped to a wall and loaded at the tip. This example demonstrates how accurate the local stress distribution is when a RSSCON entry is used to model the clamped condition. Four different finite element models are used, and their results are compared with classical beam theory. The deflection and bending stress from beam theory are 0.1103 in and 9,840 psi, respectively, at the clamped end of the cantilever.

1. The first model is a plate model consisting of all CQUAD4s. The stress contours of this model are shown in [Figure 14-26](#).
2. The second model is a plate model consisting of all CHEXAs. The stress contours of this model are shown in [Figure 14-28](#).
3. The third model is a plate-solid model using MPCs to transition between the plate and solid model. The input file and stress contours of this model are shown in [Listing 14-1](#) and [Figure 14-29](#), respectively.
4. The fourth model is a plate-solid model using RSSCONs to transition between the plate and solid model. The input file and stress contours of this model are shown in [Listing 14-2](#) and [Figure 14-31](#), respectively.

An exploded view of the transition used in the third and fourth models are shown in [Figure 14-26](#). Both the deflections and stress contours of the four models are in good agreement with the theoretical results, as shown in [Table 14-3](#).

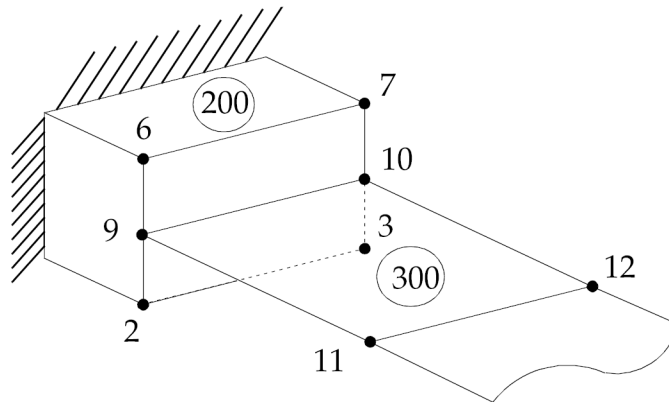


Figure 14-26. Stress Distribution for CQUAD4 Model

Table 14-3. Tip Displacements and Maximum Stresses at the Clamped End

Cantilever Clamped to Wall	Displacement at Tip	Stress at Clamped End (psi)
Beam Theory (Rigid Wall)	0.1103	9840.
All CQUAD4s	0.1103	9720.
All CHEXAs	0.1087	9720.
Plate-to-Solid Transition Using MPCs	0.1103	9720.
Plate-to-Solid Transition Using RSSCONS	0.1103	9720.

```

$
$   mpc1.dat
$
TIME 100
SOL 101
CEND
TITLE= cant cantilever clamped with HEXA
$
ECHO=BOTH
$SEALL=ALL
DISPLACEMENT =ALL
stress(corner) = all
OLOAD =ALL
spc= 100
mpc= 200
$
SUBCASE 10
LOAD = 100
BEGIN BULK
$
param,post,-1
$
grid,1,, 0., 0., 0.
grid,2,,0.1, 0.,0.
grid,3,,0.1,0.5,0.
grid,4,,0.,0.5,0.
grid,5,,0.,0.,0.1
grid,6,,0.1,0.,0.1
grid,7,,0.1,0.5,0.1
grid,8,,0.,0.5,0.1
grid,9,,0.1,0.,0.05
grid,10,,0.1,0.5,0.05
grid,11,,1.1,0.,0.05
grid,12,,1.1,0.5,0.05
grid,13,,2.1,0.,0.05
grid,14,,2.1,0.5,0.05
grid,15,,3.1,0.,0.05
grid,16,,3.1,0.5,0.05
grid,17,,4.1,0.,0.05
grid,18,,4.1,0.5,0.05
$
spc1, 100, 123, 1, 4, 5, 8
$
MPC, 200, 9,5, 1., 6, 1, -10.,,+MP01
+MP01, , 2,1, 10.
MPC, 200,10,5, 1., 7, 1, -10.,,+MP02
+MP02, , 3,1, 10.
MPC, 200, 9,4, 1., 2, 2, -10.,,+MP03
+MP03, , 6,2, 10.
MPC, 200,10,4, 1., 3, 2, -10.,,+MP04
+MP04, , 7,2, 10.
MPC, 200, 9,1, 1., 6, 1, -0.5.,+MP05
+MP05, , 2,1, -0.5
MPC, 200, 9,2, 1., 6, 2, -0.5.,+MP06
+MP06, , 2,2, -0.5
MPC, 200, 9,3, 1., 6, 3, -0.5.,+MP07
+MP07, , 2,3, -0.5
MPC, 200,10,1, 1., 7, 1, -0.5.,+MP08
+MP08, , 3,1, -0.5
MPC, 200,10,2, 1., 7, 2, -0.5.,+MP09
+MP09, , 3,2, -0.5
MPC, 200,10,3, 1., 7, 3, -0.5.,+MP10
+MP10, , 3,3, -0.5
$
chexa, 200, 210, 1, 2, 3, 4, 5, 6, +HX1
+HX1, 7, 8

```

```

CQUAD4, 300, 310, 9, 11, 12, 10
CQUAD4, 301, 310, 11, 13, 14, 12
CQUAD4, 302, 310, 13, 15, 16, 14
CQUAD4, 303, 310, 15, 17, 18, 16
$
PSOLID 210 100
PSHELL 310 100 .1 100 1. 100
MAT1 100 1.+7 0.
$
FORCE 100 17 1. 0. 0.0 -1.
FORCE 100 18 1. 0. 0.0 -1.
$
ENDDATA

```

Listing 14-1. Input File for Solid-Plate Model Using MPCs for Transition


```

$   rsscon1.dat
$
TIME      100
SOL 101
CEND
TITLE = CANTILEVER CLAMPED WITH HEXA
$
ECHO=BOTH
  DISPLACEMENT = ALL
  stress(corner) = all
  OLOAD = ALL
SPC = 100
$
SUBCASE 10
  LOAD = 100
$
BEGIN BULK
$
param,post,-1
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.1,0.0,0.0
GRID,3,,0.1,0.5,0.0
GRID,4,,0.0,0.5,0.0
GRID,5,,0.0,0.0,0.1
GRID,6,,0.1,0.0,0.1
GRID,7,,0.1,0.5,0.1
GRID,8,,0.0,0.5,0.1
GRID,9,,0.1,0.0,0.05
GRID,10,,0.1,0.5,0.05
GRID,11,,1.1,0.0,0.05
GRID,12,,1.1,0.5,0.05
GRID,13,,2.1,0.0,0.05
GRID,14,,2.1,0.5,0.05
GRID,15,,3.1,0.0,0.05
GRID,16,,3.1,0.5,0.05
GRID,17,,4.1,0.0,0.05
GRID,18,,4.1,0.5,0.05
$
SPC1, 100, 123, 1, 4, 5, 8
$
RSSCON, 200, elem, 300, 200
$
CHEXA, 200, 210, 2, 3, 7, 6, 1, 4, +HX1
+HX1, 8, 5
CQUAD4, 300, 310, 9, 11, 12, 10
CQUAD4, 301, 310,11, 13, 14, 12
CQUAD4, 302, 310,13, 15, 16, 14
CQUAD4, 303, 310,15, 17, 18, 16
$
PSOLID  210      100
$
PSHELL  310      100      .1      100      1.      100
MAT1    100      1.+7      0.
$
FORCE   100      17      1.      0.      0.0      -1.
FORCE   100      18      1.      0.      0.0      -1.
$
ENDDATA

```

Listing 14-2. Input File for Solid-Plate Model Using RSSCONS for Transition

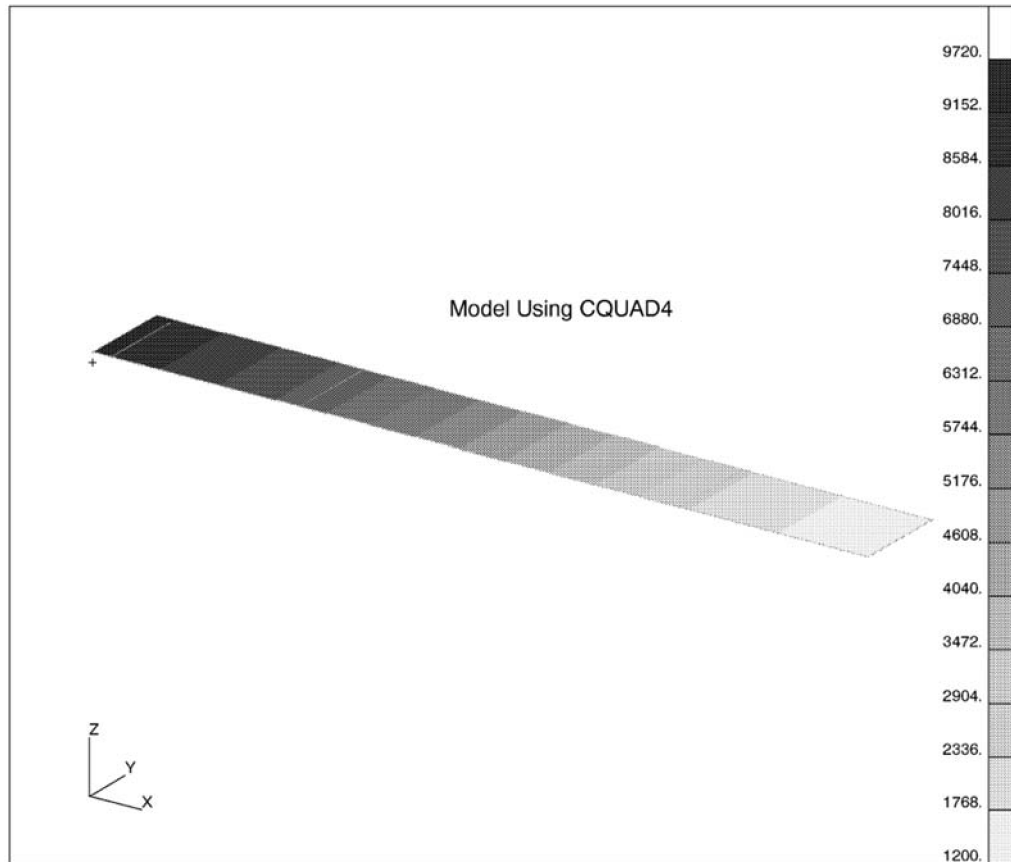


Figure 14-27. Stress Distribution for CQUAD4 Model

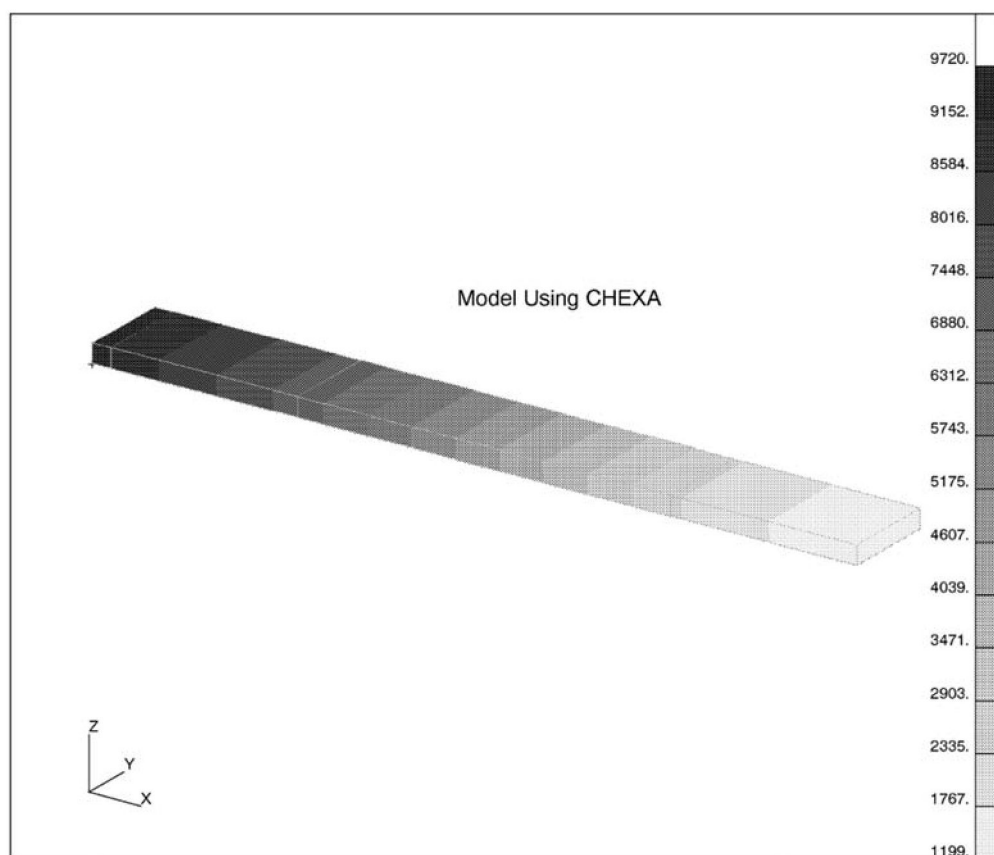


Figure 14-28. Stress Distribution for CHEXA Model

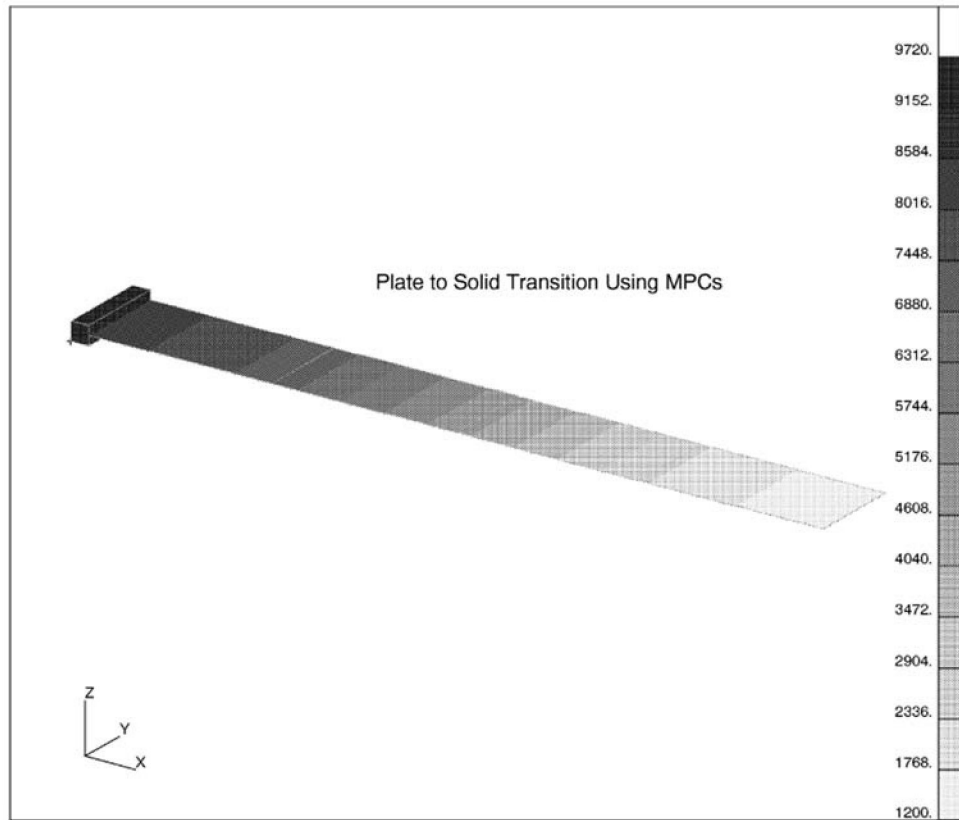


Figure 14-29. Stress Distribution for Solid-Plate Model Using MPCs for Transition

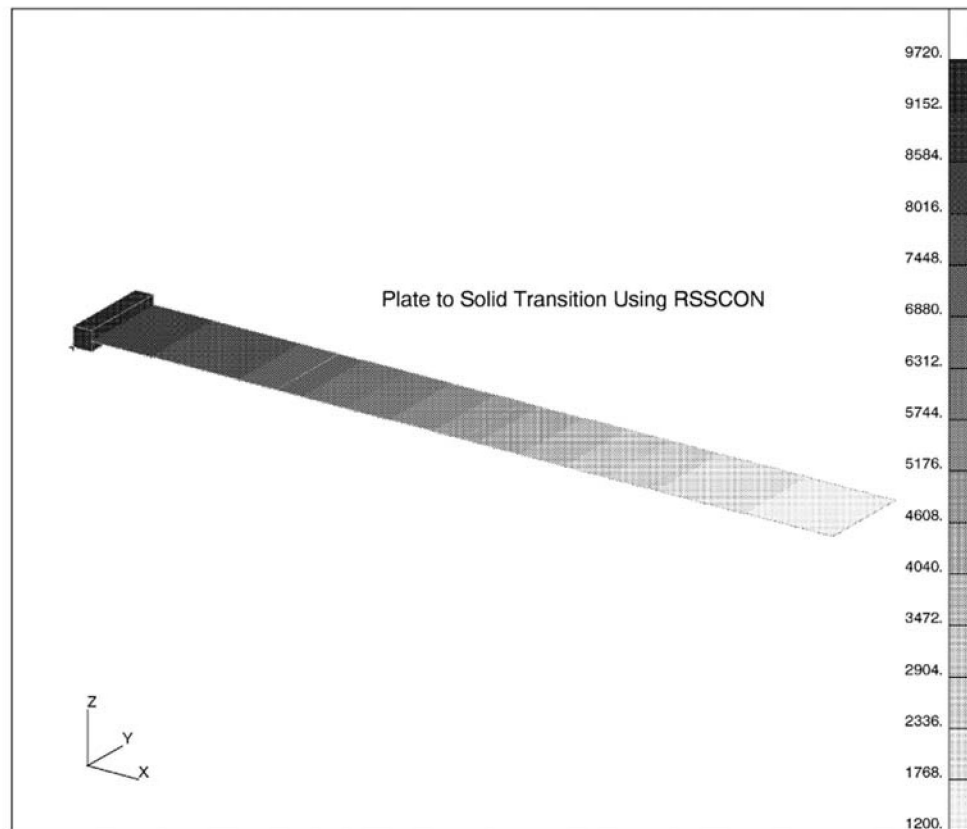


Figure 14-30. Stress Distribution for Solid-Plate Model Using RSSCONs for Transition

14.5 Reviewing Grid Point Stresses

When you're working with a structure that's modeled with either plate or solid elements, you will generally want to review the element component stresses. If this is the case, extra care must be taken on your part to ensure that the component stresses, which you are evaluating, are output in some consistent manner.

Consider a simple problem as shown in [Figure 14-31](#). This plate, which is 0.1 inch thick, is subjected to a uniaxial tensile load of 1 lb/in, and the boundary condition is as shown in [Figure 14-31](#). This model is represented with a 3 x 3 mesh of CQUAD4 elements. If the elements are connected in a manner as shown in [Figure 14-32](#), then all the element σ_x values are in the direction of the applied loads. In this case, $\sigma_x = 10$ psi for all nine elements. On the other hand, if you were to change, for instance, the connectivity order for element 5 from grid points 6-7-11-10 to 7-11-10-6, the component stresses and would be swapped. Note that this is the only difference between [Figure 14-32](#) and [Figure 14-33](#). However, if you are not aware of it and you are making a contour plot of σ_x , the results will be quite different for the two cases.

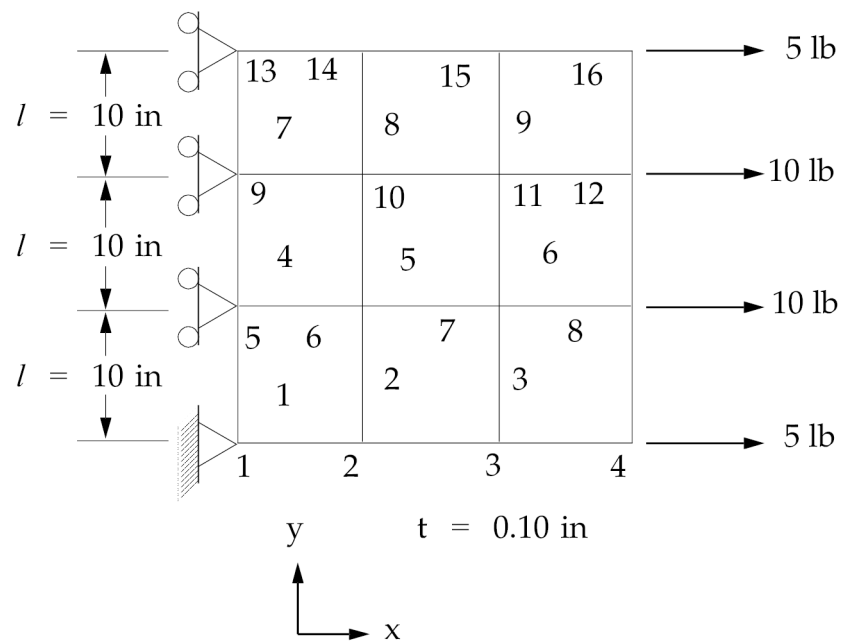


Figure 14-31. Uniaxial Load for a 3 x 3 Mesh

CQUAD4,7,1,9,10,14,13
 CQUAD4,8,1,10,11,15,14
 CQUAD4,9,1,11,12,16,15
 \$

CQUAD4,4,1,5,6,10,9
 CQUAD4,5,1,6,7,11,10
 CQUAD4,6,1,7,8,12,11
 \$

CQUAD4,1,1,1,2,6,5
 CQUAD4,2,1,2,3,7,6
 CQUAD4,3,1,3,4,8,7
 \$

$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$

Figure 14-32. Element Component Stresses

CQUAD4,7,1,9,10,14,13	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,8,1,10,11,15,14	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,9,1,11,12,16,15	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			
CQUAD4,4,1,5,6,10,9	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,5,1,7,11,10,6	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,6,1,7,8,12,11	$\sigma_v = 10$	$\sigma_y = 10$	$\sigma_v = 10$
\$			
CQUAD4,1,1,1,2,6,5	$\sigma_x = 10$	$\sigma_x = 10$	$\sigma_x = 10$
CQUAD4,2,1,2,3,7,6	$\sigma_y = 0$	$\sigma_y = 0$	$\sigma_y = 0$
CQUAD4,3,1,3,4,8,7	$\sigma_v = 10$	$\sigma_v = 10$	$\sigma_v = 10$
\$			

Figure 14-33. Element Component Stresses

To avoid this type of surprise, it may be easier and more meaningful to look at the invariant stress quantities, such as Hencky-von Mises stresses, rather than the component stresses. The grid point stress (GPSTRESS) option in NX Nastran offers you another alternative. The grid point stress option calculates the stresses at the grid points from the adjoining plate and solid elements in a coordinate system defined by you.

This option enables you to request the output of stresses at grid points in surfaces of two-dimensional plate elements—namely, the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, and CTRIA6 elements—and in volumes containing CHEXA, CPENTA, CPYRAM, and CTETRA solid elements.

See also

- “[Element Data Recovery Resolved at Grid Points](#)” in the *NX Nastran User's Guide*

14.6 Defining Consistent Loading

In finite element analysis, you often apply loads to the elements rather than to the grid points. Examples of such loadings include the edge load on a CQUAD4 element or a pressure load on one of the faces of a solid element.

When you convert these element loads to grid point loads, a common tendency is to spread them equally at the connected grid points. Let us look at an example of a typical plate element with uniform in-plane loading at one of the edges as shown in [Figure 14-34](#). The total load (P) is to be uniformly distributed along the edge of the element with an intensity of P/l , where l is the length of the edge. This same load is to be applied to six different elements—CQUAD4, CTRIA3, CQUAD8, CTRIA6, CQUADR, and CTRIAR. If you lump these loads equally to the grid points, the load at each node is $P/2$ for the CQUAD4, CTRIA3, CQUADR and CTRIAR, and $P/3$ for the CQUAD8 and CTRIA6. Since the geometry and boundary condition are identical for the six models, and the loads are “seemingly equivalent,” you will expect the results (displacements, stresses, etc.) to be identical. However, in general, this is not the case. The results for the cases using CQUAD8, CTRIA6, CQUADR, and CTRIAR are different from the results using CQUAD4 and CTRIA3. The reason for this is because these “seemingly equivalent” loads are actually not quite equivalent. They

are considered as lumped loads. In order to obtain the correct load distribution, these loads need to be converted to consistent loads.

The equivalent loads at the grid points computed from the element loads are known as consistent loads, and they are calculated by applying the principle of virtual work. The same shape function that is used in deriving the element stiffness is used for arriving at this load-and hence the word consistent load. They are a function of the element types and the applied loads.

Depending on the element, these lumped loads in general are not equal to the consistent loads. The consistent loads for each of these elements for a uniform in-plane edge load of P/l are shown next to each element. In this case, the lumped loads and consistent loads are the same for the CQUAD4 and CTRIA3. They are not the same for the CQUAD8, CTRIA6, CQUADR and CTRIAR. As you can see, they are substantially different from the lumped loads that were discussed in the previous paragraph. It is interesting to note that the loads are distributed as 1/6, 4/6, and 1/6 along the edge for the CQUAD8 and CTRIA6, which is quite different from the lumped load approach. For the CQUADR and CTRIAR, an additional moment P/l^2 is needed to arrive at the consistent loads. When the corresponding consistent loads shown in [Figure 14-34](#) are applied to each of the respective elements, the force distributions are then equivalent for all the elements.

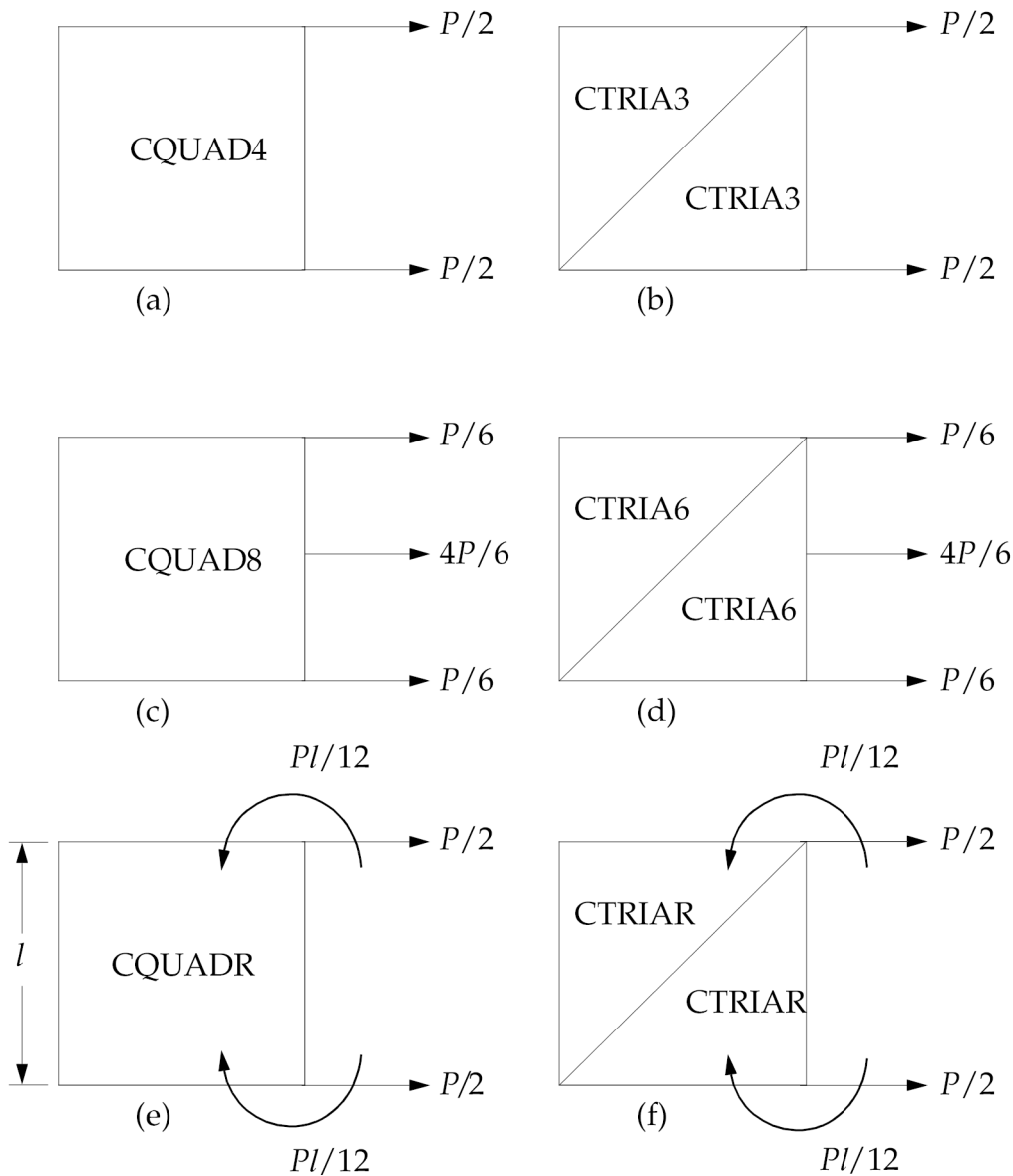


Figure 14-34. Consistent Loads Due to Uniform In-Plane Element Edge Load

Consider another example with solid elements. A load P is to be applied evenly as an outward pressure load (P/A) to a surface of each of the four solid elements shown in [Figure 14-35](#). Again, if you lump these loads equally to the grid points, the lumped load at each node is $P/4$ for the CHEXA (with eight nodes) and CPENTA (with six nodes), and $P/8$ for the CHEXA (with 20 nodes) and CPENTA (with 15 nodes). The consistent loads are shown in [Figure 14-35](#). The consistent and lumped loads are the same for the CHEXA (with eight nodes) and the CPENTA (with six nodes) for this case. The consistent loads for the CHEXA (with 20 nodes) and CPENTA (with 15 nodes) are shown in [Figure 14-35\(c\)](#) and [Figure 14-35\(d\)](#), respectively. The load distribution along each edge is $-.0833P$, $.3333P$ and $-.0833P$, or a $-1/4/-1$ ratio, which is quite counter intuitive, especially for the sign change. Fortunately, you do not have to calculate this consistent load for the pressure load. This consistent load is generated automatically inside NX Nastran by using the PLOAD4 entry.

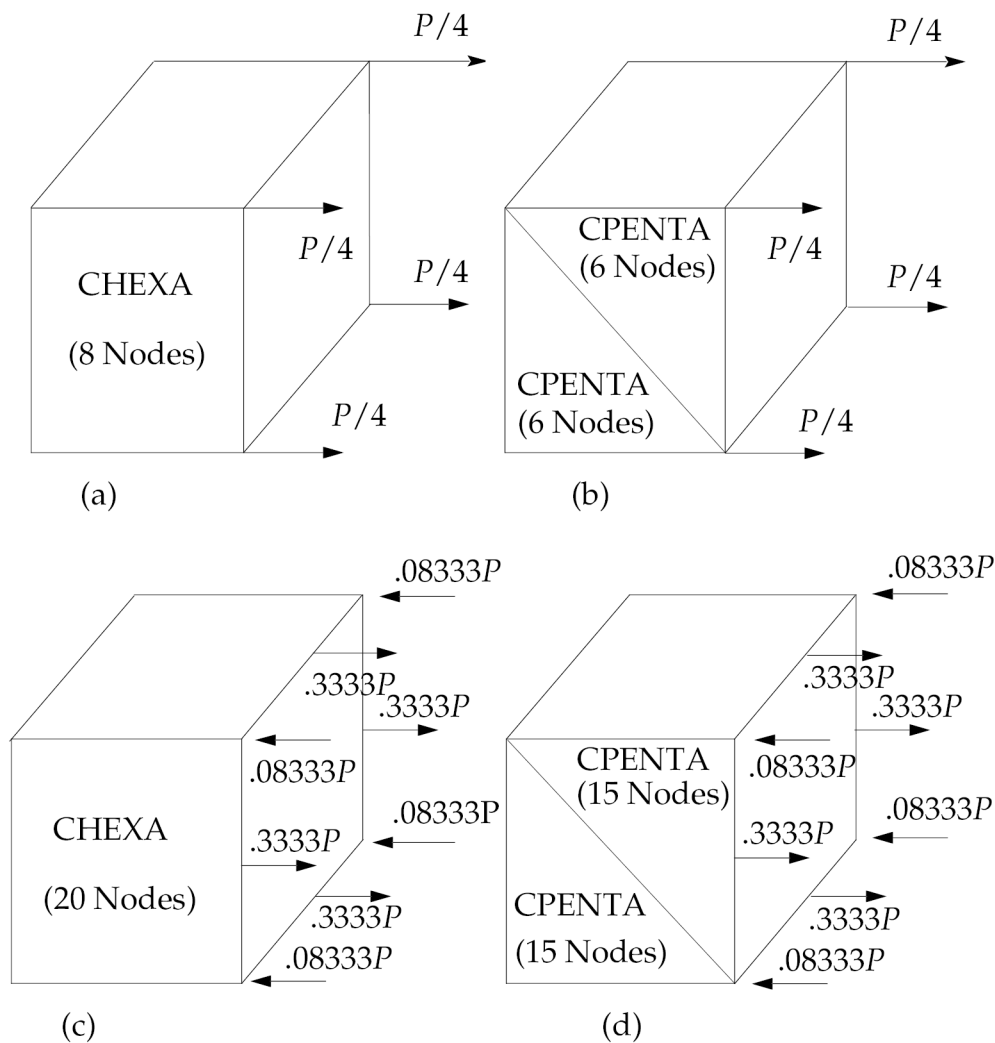


Figure 14-35. Consistent Loads Due to Uniform Loads on a Solid Element Face

As you can see, consistent loads are functions of the element types and applied loads.

14.7 Symmetry

When you think of a symmetrical structure, you most likely think of a structure that has one or more planes of reflective symmetry. Although there are other kinds of symmetry available in NX Nastran, reflective symmetry is the only type that is discussed in this section. NX Nastran also provides a series of special solution sequences that automates some of these other types of analyses. They are known as the cyclic symmetry features. If a structure is symmetric, then the size of your finite element model can be reduced, which, in turn, reduces the time and cost of your analysis. For each plane of symmetry that you have in your model, the model size can be reduced by a factor of approximately two. [Figure 14-36\(a\)](#) and [Figure 14-36\(b\)](#) illustrate structures that contain one and two planes of symmetry, respectively. In the first case, only half the model needs to be represented. In the second case, only a quarter of the model needs to be represented.

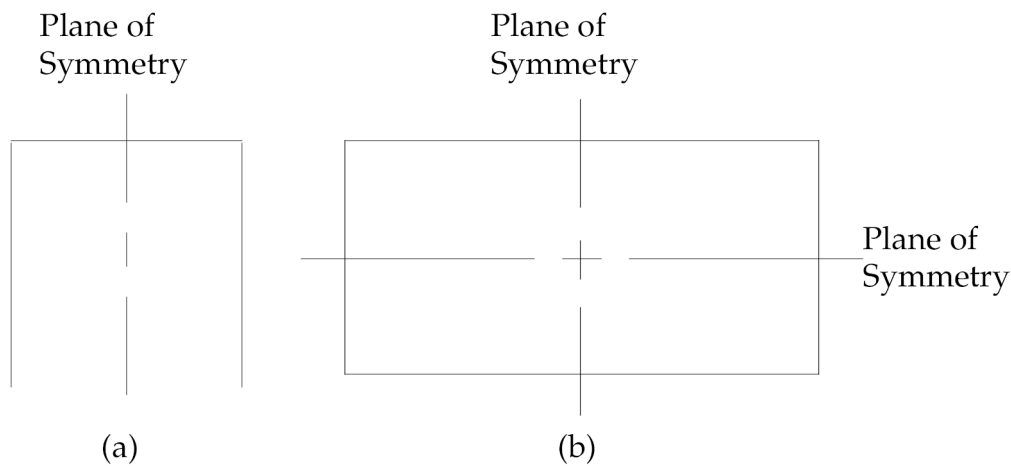


Figure 14-36. Structures with Planes of Symmetry

If the loads applied to the structure are symmetric relative to the plane of symmetry, then the full model can be replaced with half the model by applying a symmetric boundary condition. On the other hand, if the loads are antisymmetric, the same simplification can be achieved by applying the antisymmetric boundary condition.

A symmetric boundary condition implies that the displacements normal to the plane of symmetry and rotations about the axes in the plane of symmetry are zero at the plane of symmetry. An antisymmetric boundary condition implies that the displacements in the plane of symmetry and rotations normal to the plane of symmetry are zero at the plane of symmetry.

Any general loading condition can be broken into a combination of symmetric and antisymmetric loads relative to the plane of symmetry. You can work through an example of applying symmetry using the model in [Figure 14-37\(a\)](#). This load can be broken into two separate loads as shown in [Figure 14-37\(b\)](#) and [Figure 14-37\(c\)](#).

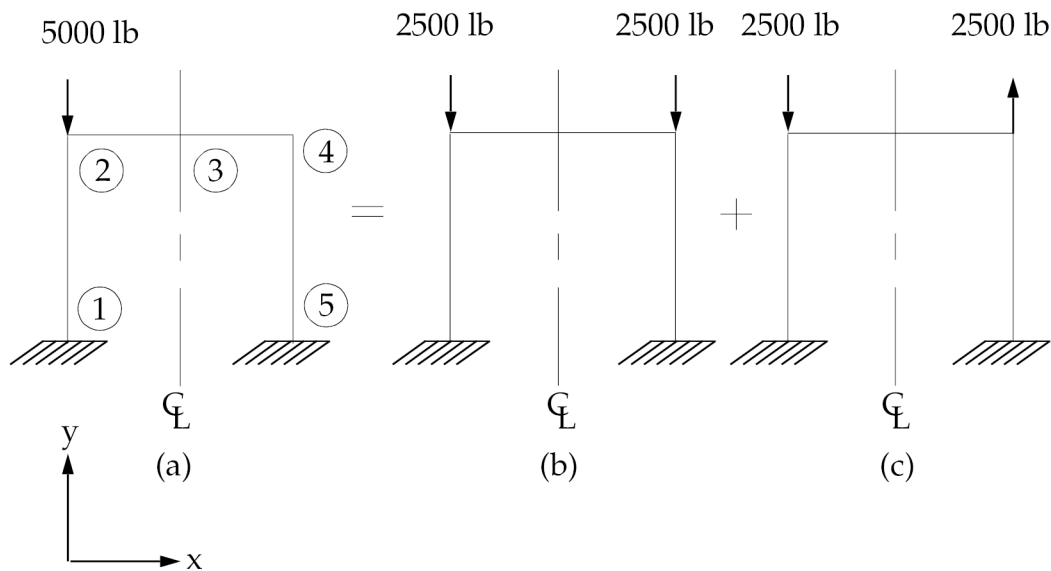


Figure 14-37. Decomposing a General Loading into Symmetric and Antisymmetric Loadings

Note that **Figure 14-37(a)** is a case of a general loading applied to a structure with one plane of symmetry. **Figure 14-37(b)** and **Figure 14-37(c)** represent a symmetric loading and an antisymmetric loading, respectively, applied to the same structure. By using symmetry, only one-half of the structure, as shown in **Figure 14-37(b)**, needs to be modeled by applying the symmetric boundary condition at the plane of symmetry (see **Figure 14-38**).

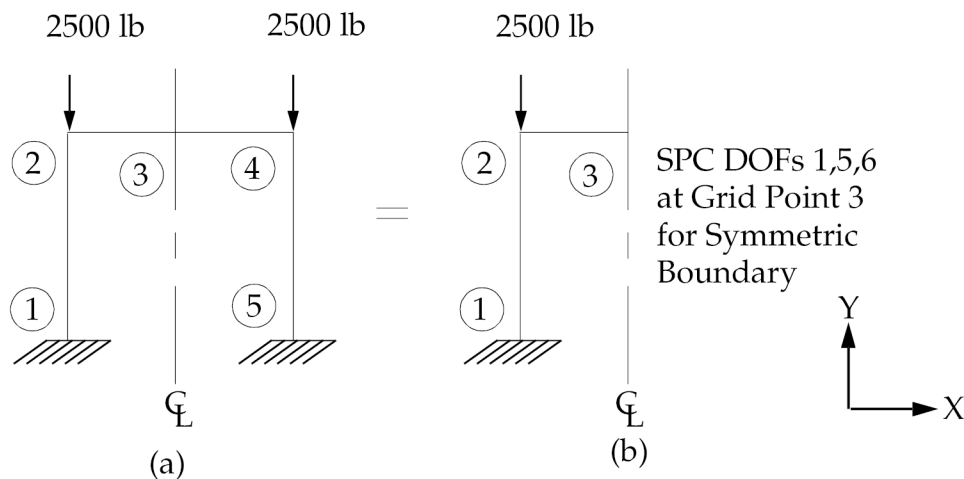


Figure 14-38. Symmetric Model

Similarly, only one-half of the structure shown in **Figure 14-37(c)** needs to be modeled by applying the antisymmetric boundary condition as shown in **Figure 14-39**.

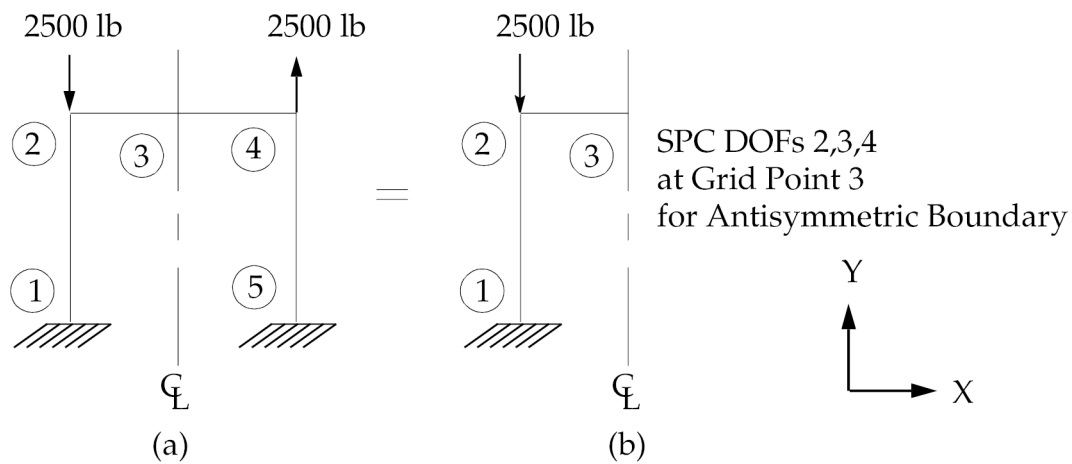


Figure 14-39. Antisymmetric Model

User Interface

No special user interaction is required in the Executive Control Section or Bulk Data Section. Depending on the number of planes of symmetry, approximately one-half or less of the structure needs to be modeled. For the above frame model, only one-half of the model needs to be analyzed.

This feature is basically activated by Case Control commands. The above frame example is used to illustrate the required Case Control commands. **Listing 14-3** contains the input file for this problem.

```

$
$ FILENAME - symbar.dat
$
ID SYM ANTI
TIME 5
SOL 101
CEND
$
TITLE = SYMMETRIC AND ANTISYMMETRIC
SUBCASE 1
  LABEL = SYMMETRIC CONSTRAINTS - Y LOAD
  SPC = 1
  LOAD = 2
$
SUBCASE 2
  LABEL = ANTISYMMETRIC CONSTRAINTS - Y LOAD
  SPC = 2
  LOAD = 2
$
SUBCOM 3
  LABEL = LEFT SIDE OF MODEL - Y LOAD
  SUBSEQ 1.0, 1.0
  DISP=ALL
$
SUBCOM 4
  LABEL = RIGHT SIDE OF MODEL - Y LOAD
  SUBSEQ 1.0, -1.0
  DISP=ALL
$
BEGIN BULK
CBAR    1      100    1      2      -1.0    0.0    0.0
CBAR    2      100    2      3      0.0    1.0    0.0
FORCE   2      2      2500.  0.0    -1.0    0.0
GRID    1      0.0    0.0    0.0      123456
GRID    2      0.0    10.0   0.0
GRID    3      5.0    10.0   0.0
MAT1    1      3.+7   0.3
PBAR    100    1      5.0    5.0    10.
SPC1    1      156    3
SPC1    2      234    3
ENDDATA

```

Listing 14-3. Input File for Using Symmetric and Antisymmetric Boundary Conditions

The first subcase calls out the symmetric boundary condition and load as shown in [Figure 14-38\(b\)](#). The second subcase calls out the antisymmetric boundary condition and load as shown in [Figure 14-39\(b\)](#).

The third subcase is a “results combination” subcase. No boundary or load conditions need to be applied for this loading condition (see “[Modeling Guidelines](#)”). This subcase produces results for the portion of the structure that you have modeled (the left-hand side of the frame in this case) by performing a linear combination of the first two subcases. This combination is achieved by adding 100% of the results for the first subcase to 100% of the second subcase using the SUBSEQ command. Note that in this case of results combination, the SUBCOM instead of the SUBCASE command is used.

The fourth subcase is also a “results combination” subcase. It produces results for the portion of the structure that you did not model (the right-hand side of the frame in this case). These results are achieved by subtracting 100% of the results of the second subcase from 100% of the results from the first subcase. As in Subcase 3, the SUBCOM command instead of the SUBCASE command is used in this case. Once again, no load or boundary condition is needed for this subcase; this is

an optional subcase. It is not required if you do not want to obtain results for the other half of the structure that you did not model.

The displacements for SUBCOM 3 and SUBCOM 4 are shown in [Figure 14-40](#) along with the results obtained from a full model run. The results for SUBCOM 3 (left-hand side of the frame) correlate with the full model results. The results for SUBCOM 4 (right-hand side of the frame) correlate with the full model results except for the sign changes for components x , θ_y , and θ_z . The sign changes occur because the results of the right half (reflected half) are produced in terms of its left half using the left-hand coordinate system. Note that these are the same degrees of freedom that are constrained for the symmetric boundary condition.

LEFT SIDE OF MODEL - Y LOAD								SUBCOM 3
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0	.0	.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0	.0	.0	.0	3.453039E-05
RIGHT SIDE OF MODEL - Y LOAD								SUBCOM 4
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	1.381215E-04	-5.524853E-06	.0	.0	.0	.0	-2.762431E-05
3	G	1.381215E-04	-1.666667E-04	.0	.0	.0	.0	-3.453039E-05

FULL MODEL - NEGATIVE Y LOAD								SUBCASE 1
POINT ID.	TYPE	T1	D I S P L A C E M E N T			V E C T O R		
			T2	T3		R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0	.0
2	G	-1.381215E-04	-3.278085E-04	.0	.0	.0	.0	2.762431E-05
3	G	-1.381215E-04	-1.666667E-04	.0	.0	.0	.0	3.453039E-05
4	G	-1.381215E-04	-5.524862E-06	.0	.0	.0	.0	2.762431E-05
5	G	.0	.0	.0	.0	.0	.0	.0

Figure 14-40. Results Comparison Using Symmetry Versus a Full Model

In the past, when computers were not as fast as they are now, the use of symmetry to reduce the finite element model size was more popular than it is today. The advantage of using symmetry is obviously the reduction in model size. The disadvantage is that it requires more effort on your part and it is more prone to errors since you have to provide equivalent symmetric/antisymmetric boundaries and loading conditions to your model. In NX Nastran, the use of symmetry is built into some of the solution sequences—they are known as the cyclic symmetry solution sequences. Similar approaches can be used with superelement analysis.

The SYM/SYMCOM Case Control command combination can be used instead of the SUBCASE/SUBCOM combination. If you use this combination, the only changes you have to make is to replace the SUBCASE and SUBCOM Case Control commands with the SYM and SYMCOM Case Control commands, respectively. When the SUBCASE/SUBCOM combination is used, output is available in both the SUBCASE and SUBCOM subcases. On the other hand, when the SYM/SYMCOM combination is used, output requests are only available in the SYMCOM subcases. There is no real advantage to using the SYM/SYMCOM command as compared to the SUBCASE/SUBCOM command.

Chapter 15: Model Verification

- *Introduction to Model Verification*
- *Preprocessor Checks*
- *Strain Energy Output*
- *Diagnostic Tools*
- *Stress Error Estimators*
- *Postprocessor Checks*

15.1 Introduction to Model Verification

This chapter is dedicated to various tools that are available for verifying and improving the quality of your models. They are broken into five different categories:

- Preprocessor checks.
- Strain energy output.
- Diagnostic tools.
- Stress error estimators.
- Postprocessor checks.

Some of these categories overlap each other. Most of these checks provide you with a tremendous amount of information, but yet their usages are quite straightforward and require very little effort on your part.

15.2 Preprocessor Checks

Preprocessors are used extensively for the generation of finite element models. They also contain features that can help you to improve the quality of your model and detect errors in your model prior to analysis. These features can result in substantial savings in terms of time and money. Depending on the graphics package that you use, some of these features may or may not be available to you.

Creating a Shrink Plot of the Elements

The shrink option allows you to shrink your elements by a specified percentage. This feature is an excellent tool for identifying missing elements. As an example, let us look at a plate model with 12 CQUAD4 elements surrounded by 31 CBAR elements as shown in [Figure 15-1](#). (A) shows the regular plot, while (B) shows the plot with the shrink option turned on. The CBAR elements aren't visible since they lie on top of the edges of the CQUAD4 elements.

If you turn on the shrink option, as shown in [Figure 15-1\(b\)](#), you can easily see that you're missing a CBAR element at the top and a CQUAD4 element close to the center. You may argue that had the element labels been turned on, the missing elements would probably have been noticed. This is certainly a valid argument in this case since the model is a simple one. However, for complex models, the picture would have become too crowded to be useful had all the labels been turned on. This same shrink feature can also be applied to solid elements in a similar manner.

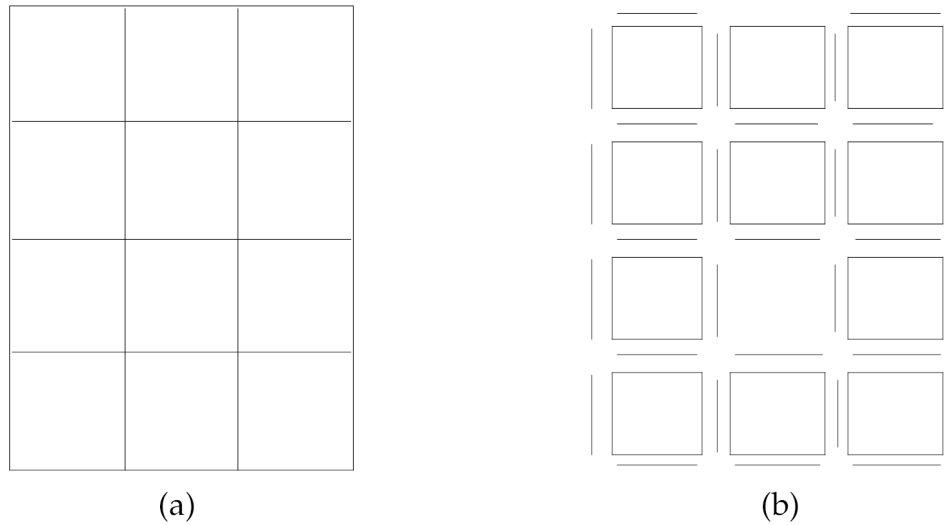


Figure 15-1. Shrink Plot

Hidden Line/Shaded Plots

For a complicated model, looking at a wireframe model by itself can be quite confusing since you lose the depth of field. However, hidden line and shaded plots can serve as the perfect complement to the wireframe plots. The box model in [Figure 15-2](#) is a perfect example. This model is made of CHEXA elements and has a hole in the front. By looking at the wireframe plot on the top left-hand corner of [Figure 15-2](#), you may not notice that there is a hole in the front. However, the hidden line (top right), shaded (bottom left), and shaded with shrink option (bottom right) plots all reveal the hole in front of the box. Furthermore, when using these features, you can see the structure being plotted from back to front, which gives you a better perspective of what the structure looks like. As the structure becomes more complicated, you will appreciate these features even more.

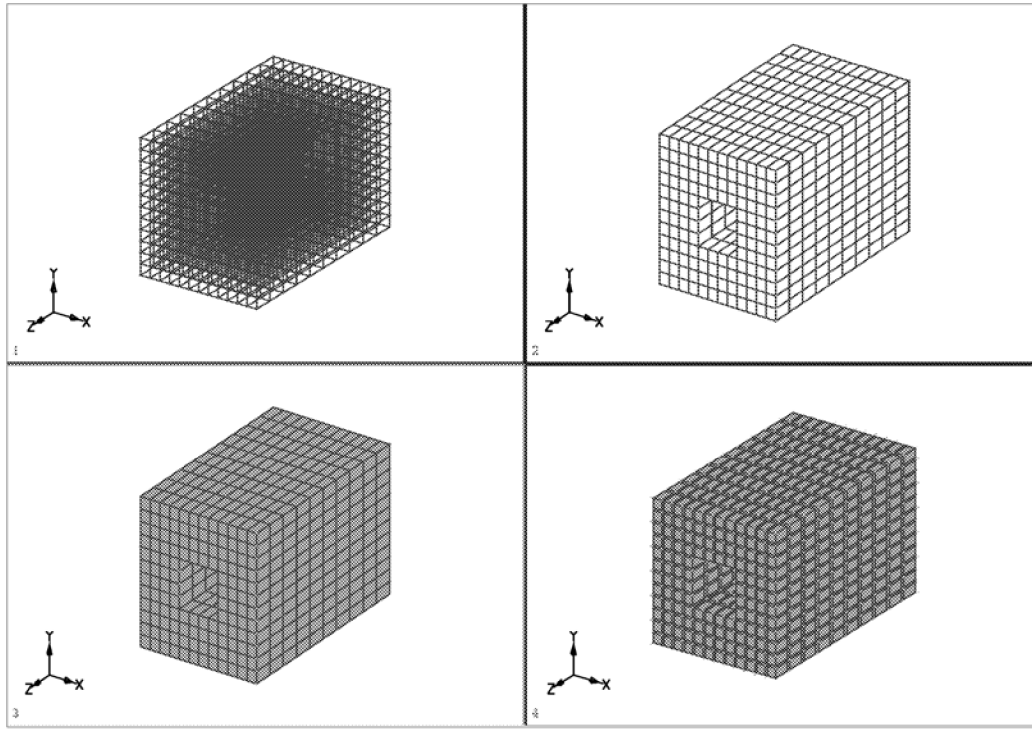


Figure 15-2. Wireframe, Hiddenline, and Shaded Plots

Free Edge/Face

A free edge is an edge that is connected by one single two-dimensional element (e.g., CQUAD4). The existence of free edges does not necessarily indicate a modeling problem. Figure 15-3 represents a crude wing model. The bold lines around the outside are examples of legitimate free edges. On the other hand, whether the internal vertical bold line is a legitimate free edge or not depends on your design intent. This free edge indicates that elements 12 and 14, 11, and 13 are not connected to each other. If this is not your intention, then this free edge indicates a potential modeling error.

This can happen, for example, if you create this wing with four separate surfaces as shown in Figure 15-4. Most preprocessors typically create geometric surfaces and then generate the finite element mesh for each of these surfaces. Since elements 12 and 14 are created from two separate geometric surfaces, their connecting grid points have distinct IDs. After these elements are generated, if you want them to share the same edge, then you must perform some sort of equivalencing operation. The exact operation depends on the graphics package itself. This free edge indicates that you have either forgotten to perform this equivalencing operation or the equivalencing tolerance level is not tight enough for the program to perform this task.

16	14	12	10	8	6	4	2
15	13	11	9	7	5	3	1

Figure 15-3. Wing Free Edge Check

Surface 1	Surface 2	Surface 3	Surface 4
--------------	--------------	--------------	--------------

Figure 15-4. Wing Geometric Surfaces

The free face concept is similar to the free edge concept except that it applies to three-dimensional elements instead of two-dimensional elements. A free face is a face that is occupied by one single three-dimensional element (e.g., CHEXA). The example in [Figure 15-5](#) contains two volumes. After these two volumes are meshed with solid elements (e.g., CHEXAs), a free face check is performed. The outside surfaces of these two volumes indicate that they are free faces that are legitimate. In addition, it indicates that the adjoining surface (hashed area) is also a free surface. This indicates that the two volumes are not connected, which may or may not be your intention. If it is not your intention, there is probably a modeling error.

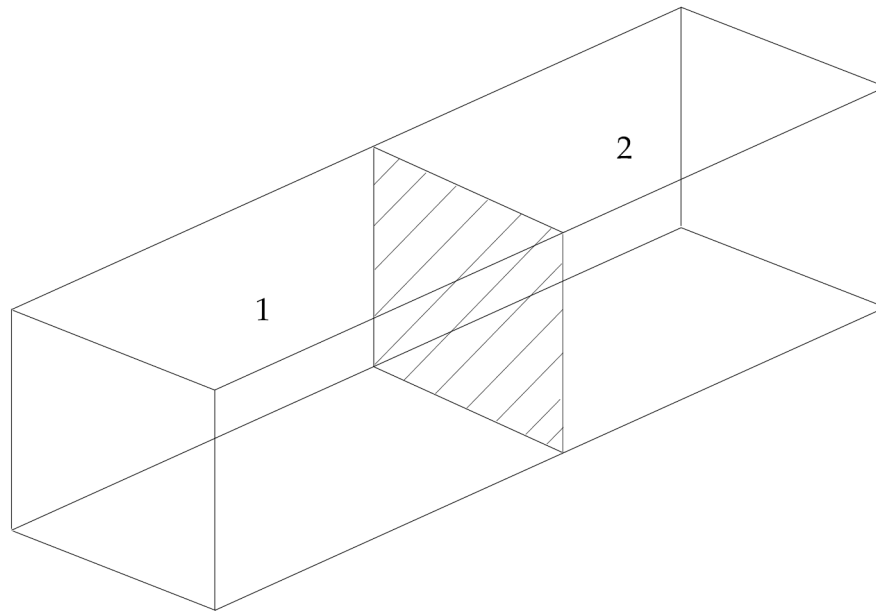


Figure 15-5. Possible Modeling Error as Indicated by Free Face

Zipper Effect

When adjoining surfaces are connected at very few points, the interface has a tendency to open up when it is loaded. This looks similar to a zipper and hence is classified as the “zipper effect.” Use the surfaces in [Figure 15-4](#) and create new meshes. Surfaces 1, 2, 3, and 4 contain meshes of 4X2, 4X2, 3X2, and 3X2, respectively. As you can see in [Figure 15-6](#) the interface between surfaces 2 and 3 is only connected at two grid points—grid points A and B. The loads can only be transferred between these two portions of the structure through these two locations. In other words, the load path may be quite different from what the actual structure does.

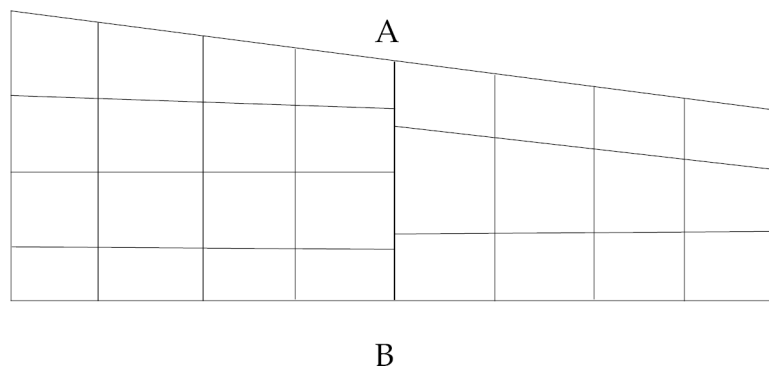


Figure 15-6. Model with Zipper Effect

The “zipper effect” can occur in both plate and solid element models. Therefore, when generating meshes for a structure with multiple surfaces and/or volumes, you should keep this in mind so that

the interfaces are connected properly. You should always perform a quick check by zooming in at all the interface locations.

CBAR/CBEAM Orientation and Offset Check

One common error that many users make is the incorrect orientation of the CBAR/CBEAM inertia properties. If the graphics package that you are using plots these orientation vectors, you should take advantage of this feature to ensure that you have oriented these elements properly. You should also check the CBAR/CBEAM element offset plots, if they are available.

See also

- “CBAR Element” in the *NX Nastran Element Library*
- “CBEAM Element” in the *NX Nastran Element Library*

Duplicate Grid Points/Elements

The “duplicate grid point check” feature highlights grid points that occupy the same location. Having grid points occupy the same space may or may not be your intention. An example of unintentional grid points occupying the same location can be a result of not performing an equivalencing operation after meshing more than one curve, surface, or volume. An application of intentional duplicate grid points can be a model used to simulate a bolted joint. This effect can be achieved by connecting a stiff spring between these two points that occupy the same location but are connected to different portions of the structure.

The “duplicate element check” feature is similar to the “duplicate grid check” except the check is done on the elements rather than the grid points. These errors are often due to unintentionally meshing the same line, surface, or volume more than once.

Properties/Material Plots

You can also assign different colors to elements based on their property IDs (e.g., PSHELL IDs). If you have assigned an incorrect property, graphically this becomes quite obvious. Differentiating by color can also be used to highlight material properties (e.g., MATi IDs).

Using Consistent Normals

When you create a model, you should always generate the elements in a consistent manner. In some cases, if you don't generate your model in a consistent manner, you might inadvertently apply the loads in the wrong directions.

An example of such a case is when the applied load is a pressure load. The plate model, consisting of four CQUAD4s as shown in [Figure 15-7](#), is used to illustrate this point. As mentioned in Chapter 4, each CQUAD4 has a positive normal direction associated with it. This direction is defined by the way you connect the element using the right-hand rule. The symbols used to denote the directions of the normals are shown in [Figure 15-8](#).

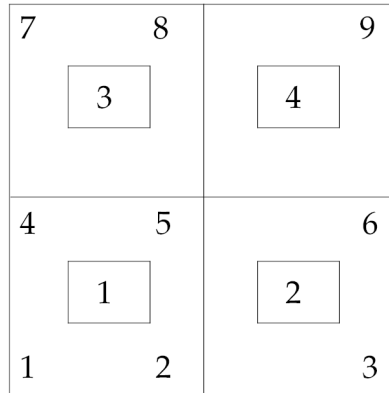


Figure 15-7. CQUAD4 Model

Now apply a 100 psi pressure load to these four elements. The direction of the pressure load is pointing into the paper. Listing 15-1 contains the relevant partial input file for this job. In this case, the grid points for each element are defined in a consistent manner. They are connected in a clockwise direction. Therefore, the positive normals are all pointing into the paper as indicated by Figure 15-8(a).

```
ID RUN1 PLOAD4
SOL 101
CEND
TITLE = CONSISTENT ELEMENT NORMAL DIRECTIONS
LOAD = 100
.
.
BEGIN BULK
$
CQUAD4,1,10,1,4,5,2
CQUAD4,2,10,2,5,6,3
CQUAD4,3,10,4,7,8,5
CQUAD4,4,10,5,8,9,6
$
PLOAD4,100,1,100.0,,,THRU,2
PLOAD4,100,3,100.0
PLOAD4,100,4,100.0
.
.
ENDDATA
```

Listing 15-1. Consistent CQUAD4 Connectivity

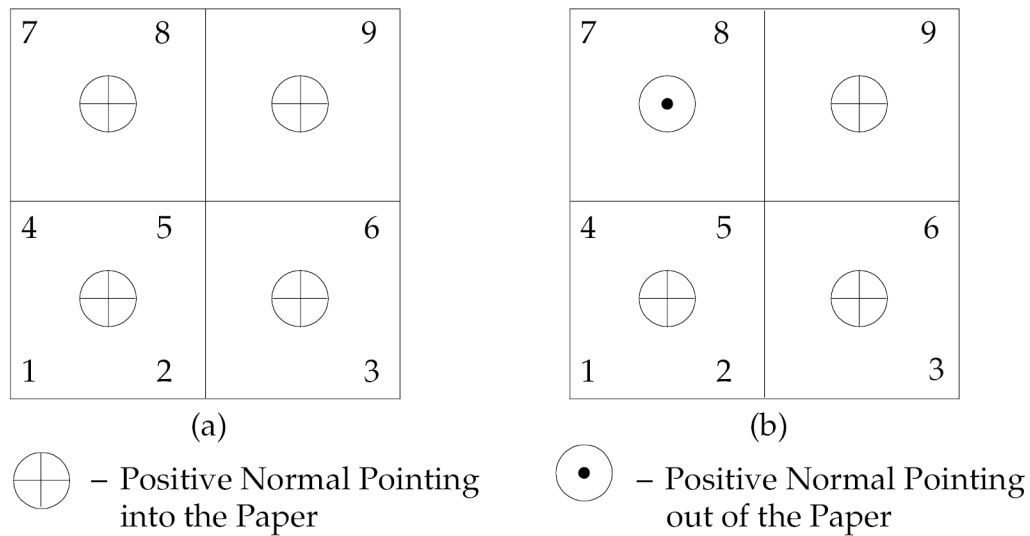


Figure 15-8. CQUAD4s with Pressure Loads

On the other hand, perhaps for some reason, you connect element 3 in a counterclockwise direction (Figure 15-8(b)) instead of a clockwise direction like the other three elements. If this is the case, you then need to change the sign of the pressure load for element number 3 in order to have identical loads for both cases (see Listing 15-2). Determining the positive direction of the pressure load for the CQUAD4 element can be achieved by checking the connectivity order using the right-hand rule. If the sign for the pressure load applied to element number 3 is not switched, then you have three elements with the pressure load pointing into the paper and the fourth element with the pressure load pointing out of the paper.

```
ID RUN1 PLOAD4
SOL 101
CEND
TITLE = INCONSISTENT ELEMENT NORMAL DIRECTIONS
LOAD = 100
.
.
BEGIN BULK
$
CQUAD4,1,10,1,4,5,2
CQUAD4,2,10,2,5,6,3
CQUAD4,3,10,4,5,8,7
CQUAD4,4,10,5,8,9,6
$
PLOAD4,100,1,100.0,,,THRU,2
PLOAD4,100,3,-100.0
PLOAD4,100,4,100.0
.
.
ENDDATA
```

Listing 15-2. Inconsistent CQUAD4 Connectivity

This type of inconsistency can occur if the whole model is generated with a preprocessor using multiple surfaces and the consistency of the connectivity direction is not retained. This problem is also likely to occur if you edit some of the elements manually and forget to account for the sign change in the pressure load. Many graphics packages offer this element normal check feature. This type

of check requires very little effort, and in many cases the graphics processor may also allow you to reverse the direction of the normals.

The CQUAD4 element outputs are in the element coordinate system. Different connectivity orders affect the way the results are printed. If you are not aware of this fact, you may interpret the results improperly (see the example in “[Stress Error Estimators](#)”).

15.3 Strain Energy Output

Frequently you are faced with the decision of modifying your structure in order to meet stringent design criteria. In most instances, your objective is to maximize the benefits with minimal changes. The element strain energy output is an excellent tool for identifying areas of modification that will reap the most benefits for design changes.

The element strain energy is basically the elastic energy stored in the structural element. As an example, if you hold onto one end of the spring and push slowly on the other end starting from rest, the load deflection curve looks something like [Figure 15-9](#) for small deflections.

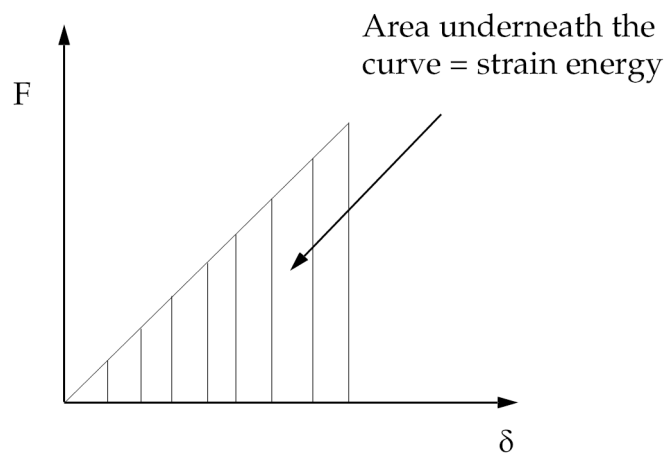


Figure 15-9. Load Deflection Curve

This energy is defined as

$$U = 0.5 \cdot F \cdot \delta$$

Since $F = k \cdot \delta$,

$$U = 0.5 \cdot k \delta^2$$

Equation 15-1.

By working through a couple of examples, you can see how the element strain energy output is used to identify areas to be modified in order to reduce deflections.

The problem of interest is shown in [Figure 15-10](#). It consists of two springs in a series with a tip load applied at grid point 2. The stiffnesses of the springs and the load applied are as shown in [Figure](#)

15-10. The goal is to reduce the tip deflection at grid point 2. Without performing any calculation, it is quite obvious that stiffening K_2 is more efficient than stiffening K_1 . The next step is to calculate the strain energy and see if it also guides you in the same direction.

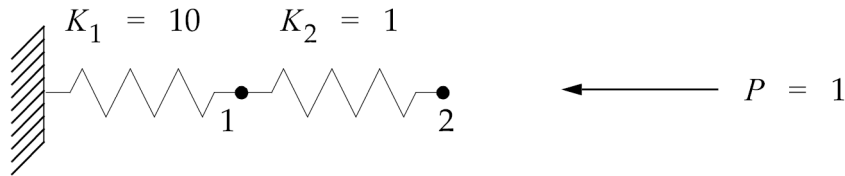


Figure 15-10. Spring Sample Problem

The deflections at grid points 1 and 2 can be calculated as follows:

$$\delta_1 = \frac{P}{K_1}; \delta_2 = \frac{P}{K_1} + \frac{P}{K_2}$$

Equation 15-2.

From [Eq. 15-1](#) and [Eq. 15-2](#),

$$U_1 = 0.5 (K_1)(\delta_1)^2 = 0.5 (10) \left(\frac{1}{10} \right)^2 = 0.05$$

$$U_2 = 0.5(K_2)(\delta_2 - \delta_1)^2 = 0.5(1) \left[\left(\frac{1}{10} \right) + \left(\frac{1}{1} \right) - \left(\frac{1}{10} \right) \right]^2 = 0.5$$

As you can see, U_2 is an order of magnitude larger than U_1 . Therefore, it concurs with our intuition that stiffening K_2 is more effective than stiffening K_1 for reducing the deflection at the tip.

The second example is a classic cantilever beam with a vertical tip load applied at the end as shown in [Figure 15-11](#). This finite element model is made up of five bar elements of equal length with square cross-sectional properties (0.05m x 0.05m) as shown in [Table 15-1](#).

Table 15-1. Cross-Sectional Properties of a Cantilever Beam							
Elem. No.	L(m)	W(m)	D(m)	A(m²)	I₁(m⁴)	I₂(m⁴)	J(m⁴)
1	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
2	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
3	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
4	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7
5	1.0	0.05	0.05	2.5E-3	5.208E-7	5.208E-7	8.789E-7

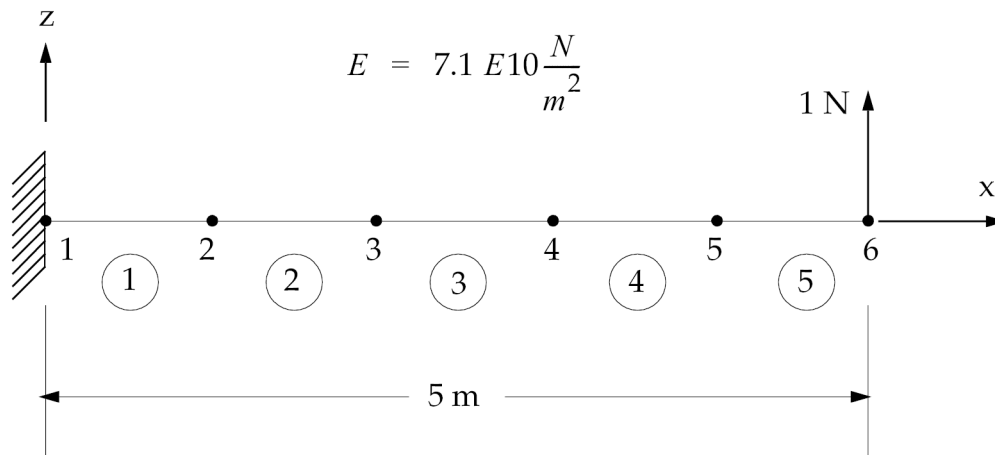


Figure 15-11. Cantilever Beam

As is commonly known, the largest deflection occurs at the tip of the cantilever beam. The vertical deflection at grid point 6 is equal to $1.126833 \cdot 10^{-3}$ meters. The goal is to minimize this deflection at grid point 6. To make the problem more interesting, hypothetically impose the following constraint such that all that is available is one structural member that is $0.06m \cdot 0.06m$ and 1m long. In other words, you can only replace one of the existing five CBAR elements with this new CBAR element. Which one should you replace to minimize the deflection at grid point 6? Since grid point 6 has the largest deflection, combined with our experience with the spring problem, you may be tempted to replace CBAR element number 5 with this new element. If you do this, the vertical displacement at grid point 6 is then reduced from $1.126833 \cdot 10^{-3}$ meters down to $1.122165 \cdot 10^{-3}$ meters, a reduction of only 0.414%. Now run this problem using NX Nastran and see what the element strain energy output suggests. **Listing 15-3** contains a listing of the NX Nastran input file used for this purpose. Note that the only additional request required is the following command in the Case Control Section:

```
ESE = ALL
```

or

```
SET a = x,y,.....ESE = a
```

or

```
ESE(PLOT) = ALL
```

For large models, the element strain energy request can potentially generate a large amount of printout. If this is not desired, you can use the “plot” option shown above to create a postprocessing file containing element strain energy data without generating printed output (see Section). The rest of the input file is standard NX Nastran input.

```

$
$  FILENAME - secant.dat
$
ID CANT ESE
SOL 101
TIME 10
CEND
$
TITLE = BASELINE MODEL
SUBTITLE =
$
SPC = 10
LOAD = 10
$
SUBCASE 1
SET 1 = 6
DISP = 1
ESE = ALL
$
BEGIN BULK
$
CBAR    1      10      1      2      10
CBAR    2     100      2      3      10
CBAR    3     100      3      4      10
CBAR    4     100      4      5      10
CBAR    5      20      5      6      10
FORCE  10      6      1.      0.      0.      0.      1.
GRID    1      0.      0.      0.
GRID    2      1.      0.      0.
GRID    3      2.      0.      0.
GRID    4      3.      0.      0.
GRID    5      4.      0.      0.
GRID    6      5.      0.      0.
GRID   10      0.      0.     10.
MAT1    1      7.1+10    .33    2700.
PBAR   10      1      2.5-3    5.208-7  5.208-7  8.789-7
PBAR   20      1      2.5-3    5.208-7  5.208-7  8.789-7
$
$  REPLACE PBAR,10,...
$  WITH THE FOLLOWING FOR .06 X .06 CROSS SECTION
$
$PBAR,10,1,3.6-3,1.08-6,1.08-6,1.823-6
$
PBAR   100      1      2.5-3    5.208-7  5.208-7  8.789-7
SPC1   10     123456    1
$
ENDDATA

```

Listing 15-3. Input File for the Strain Energy of a Cantilever Beam

The NX Nastran element strain energy output for this run is summarized in [Figure 15-12](#). As it turns out, element number 1 has the highest element strain energy among the five elements. In fact, it contains 48.8% of the total strain energy of the whole structure. On the other hand, element number 5 contains the lowest element strain energy among the five elements—only 0.8%. In other words, element number 5 is the least effective element to modify if you wish to increase the stiffness of the structure for the same amount of weight increase.

This job is now rerun by replacing element number 1 with the new 0.06×0.06 element. The deflection at grid point 6 is now reduced from 1.126833×10^{-3} meters down to 8.421097×10^{-4} meters, or a reduction of 25.27% as compared to a mere 0.414% reduction if you had replaced element number 5 instead.

E L E M E N T S T R A I N E N E R G I E S				
ELEMENT-TYPE = BAR		* TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM	=	5.634164E-04
SUBCASE 1		TOTAL ENERGY OF ALL ELEMENTS IN SET	-1 =	5.634164E-04
			*	
ELEMENT-ID	STRAIN-ENERGY	PERCENT OF TOTAL	STRAIN-ENERGY-DENSITY	
1	2.749472E-04	48.8000	1.099789E-01	
2	1.667712E-04	29.6000	6.670850E-02	
3	8.563929E-05	15.2000	3.425572E-02	
4	3.155132E-05	5.6000	1.262053E-02	
5	4.507331E-06	.8000	1.802932E-03	
TYPE = BAR	SUBTOTAL	5.634164E-04	100.0000	

Figure 15-12. Element Strain Energy Output for a Cantilever Beam Model

As you can see, the element strain energy output is an extremely useful tool in helping you to identify the most efficient locations for modification. The same concept used for the previous two simple examples can be applied to complex models in the same manner. Again, the only additional input needed is the “ESE” request in the Case Control Section. The output format is identical to that of [Table 15-2](#). Furthermore, most modern plot packages also support element strain energy output. For plate and solid elements, these strain energy plots look similar to stress contour plots.

A car model (courtesy of Lapcad Engineering), containing CQUAD4 and CTRIA3 elements, is used to illustrate the element strain energy contour plot. The Executive Control and Case Control Sections are shown in [Listing 15-4](#). As you can see, the amount of user input required is quite small. The Bulk Data for this model is inserted into the input file using the INCLUDE statement. A 1g loading is applied to each of the three orthogonal axes. The element strain energy contour plot for a 1g loading in the z-direction is shown on a deformed plot in [Figure 15-13](#).

```

$ FILENAME - secar.dat
$
$   MODEL COURTESY OF LAPCAD ENGINEERING
$
ID    LSUG, CAR
SOL 101
TIME 60
CEND
TITLE = CAR MODEL WITHOUT SUSPENSION SPRINGS
ECHO = UNSORT
SPC = 10
DISP(PLOT) = all
ESE(PLOT) = all
$
SUBCASE 10
LABEL = 1G LOAD IN X-DIRECTION
LOAD = 1
$
SUBCASE 20
LABEL = 1G LOAD IN Y-DIRECTION
LOAD = 2
$
SUBCASE 30
LABEL = 1G LOAD IN Z-DIRECTION
LOAD = 3
$
BEGIN BULK
$
INCLUDE 'carbulk.dat'
$
ENDDATA

```

Listing 15-4. Input File for the Strain Energy of a Car Model

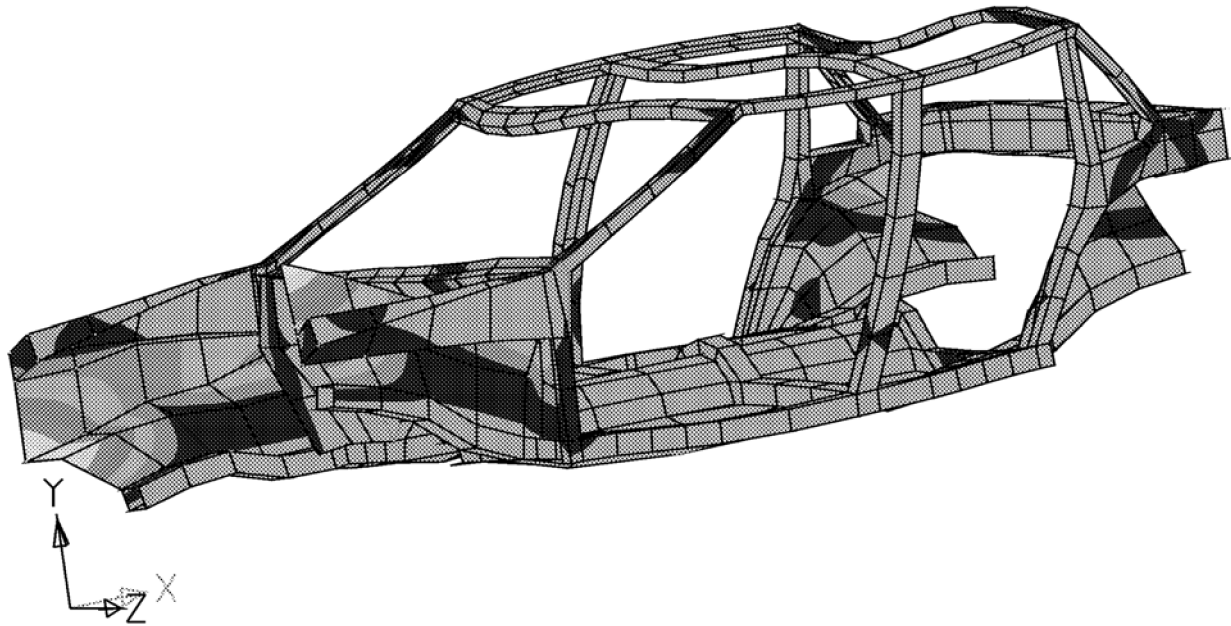


Figure 15-13. Element Strain Energy Contour Plot of an Automobile Model

(Geometry model courtesy of LAP CAD Engineering)

15.4 Diagnostic Tools

There are numerous diagnostic tools available in NX Nastran for debugging and understanding the solution process. Some of them are standard output; others are available with requests in the Executive Control Section, Case Control Section, and/or Bulk Data Section.

This section covers many of these tools. Whenever possible, the problem in [Listing 15-5](#) is used to demonstrate these features. The problem in [Listing 15-5](#) is similar to the cantilever beam shown in [Figure 15-5](#) ([Listing 15-3](#)) with the exception of an additional load case (gravity load) and output request. The modifications and additions are indicated by the shaded regions.

```

$ FILENAME - diag.dat
SOL 101
TIME 10
CEND
$
TITLE = DIAGNOSTIC TOOLS
$
SPC = 10
DISP = ALL
SPCF = ALL
OLOAD = ALL
$
SUBCASE 1
LABEL = VERTICAL TIP LOAD
LOAD = 10
$
SUBCASE 2
LABEL = GRAVITY LOADS
LOAD = 20
$
BEGIN BULK
$
$PARAM,GRDPNT,0
$
CBAR      1      10      1      2      10
CBAR      2      10      2      3      10
CBAR      3      10      3      4      10
CBAR      4      10      4      5      10
CBAR      5      10      5      6      10
FORCE    10      6      100.    0.    0.    1.
GRAV     20      9.8    0.    0.    1.
GRID      1      0.    0.    0.
GRID      2      1.    0.    0.
GRID      3      2.    0.    0.
GRID      4      3.    0.    0.
GRID      5      4.    0.    0.
GRID      6      5.    0.    0.
GRID     10      0.    0.    10.
MAT1      1      7.1+10 .33    2700.
PBAR     10      1      2.5-3  5.208-7 5.208-7 8.789-7
SPC1     10      123456  1
$
ENDDATA

```

Listing 15-5. Input File for Illustrating Diagnostic Checks

Element Summary Output

You can use the ELSUM case control command to request that an element summary be output. The element summary includes element-id, material-id, length (or thickness), area, volume, structural mass, non-structural mass, total mass, and weight (weight mass * total mass). You can specify that the element summary be grouped according to either element type or property type. You can also specify whether the software should:

- Include only mass totals for the element type or property type grouping.
- Include non-structural mass in the mass totals for the element type or property type grouping.

Prior to the ELSUM case control command, the EST parameter was used to obtain length, area, and volume output for each element in the model.

The following example uses the ELSUM case control command to request summary information for the elements included in SET 1.

```

0
0
COMMAND
COUNT
1      $
2      SET 1 = 401 THRU 403
3      DISP (PRINT, PUNCH) = ALL
4      ELST (PRINT, PUNCH) = ALL
5      ELFO (PRINT, PUNCH) = ALL
6      SPCF (PRINT, PUNCH) = ALL
7      ESE (PRINT, PUNCH) = ALL
8      SPC = 50
9      $
10     SUBCASE 100
11     ELSUM = 1
12     LOAD = 100
13     ESE (PRINT) = ALL
14     $
15     BEGIN BULK

```

The corresponding element summary is as follows:

```

0
ELEMENT TYPE = ROD
ELEMENT PROPERTY SUMMARY
ID      MID      LENGTH      AREA      VOLUME      SM      NSM      TM      WEIGHT
401     202     1.000000E+01  6.000000E+00  6.000000E+01  4.800000E+02  5.000000E+00  4.850000E+02  4.850000E+02
402     201     1.000000E+01  4.000000E+00  4.000000E+01  1.600000E+02  8.000000E+00  1.680000E+02  1.680000E+02
403     202     1.000000E+01  6.000000E+00  6.000000E+01  4.800000E+02  5.000000E+00  4.850000E+02  4.850000E+02
SUBTOTAL MASS -      1.120000E+03  1.800000E+01  1.138000E+03  1.138000E+03
TOTAL MASS -      1.120000E+03  1.800000E+01  1.138000E+03  1.138000E+03

```

Figure 15-14.

See also

- **ELSUM** in the *NX Nastran Quick Reference Guide*

Element Geometry Checks

Thorough evaluations of the various elements occurs prior to and during element matrix generation operations. There are two categories of these element geometry checks.

- The **system element checks** determine if the element geometry is adequate for finite element matrix generation. As the name implies, there is no user control to these checks. The system controlled checks will always produce a fatal error if a condition is found which prevents the analysis to proceed.
- The **user controlled element checks** are controlled with the GEOMCHECK executive statement. These checks report on the element quality even when the system checks consider all elements adequate for matrix generation. They can be turned off, or alternately, you can request that they generate a fatal error when an element fails your defined criteria. By default, they do not produce fatal errors, and occur using default values even when the GEOMCHECK statement is not included.

Taking a beam element for example, the system checks will report a fatal geometry condition if the two end points are in the same location resulting in a zero length element. The user controlled checks on the other hand check the beam for an excessive ratio of the length with offset vectors to the length

of the beam without them. In the first case, the system check is fatal because the element cannot be formulated from the given geometry. In the second case, the geometry check is informational and the simulation analysis proceeds when the GEOMCHECK defaults are used. It is your responsibility to inspect these informational messages and determine if the results are acceptable.

System Element Checks

The system element checks described below are not part of the GEOMCHECK functionality. These checks always occur, you cannot control their thresholds, and they always produce a fatal error in the .f06 file when an element fails.

Shell Element System Check	
<p>Jacobian check.</p> <p>Occurs for all shell elements: CQUAD4, CQUADR, CQUAD8, and CTRIA6.</p>	<p>For the CQUAD4, CQUADR, CQUAD8, and CTRIA6 shell elements, the determinant of the Jacobian at the Gauss points is calculated.</p> <p>A failure occurs if any $DET J \leq 0.0$.</p> <p>In addition, the average DET J is calculated on the CQUADR and CQUAD4 elements and multiplied by $1E-10$.</p> <p>A failure occurs if any $DET J < (DET J)_{AVG} \times 1E-10$.</p>
3-D Solid Element System Checks	
<p>Jacobian check.</p> <p>Occurs for all 3-D solid elements: CHEXA, CPENTA, CPYRAM, and CTETRA.</p>	<p>a. For the CHEXA, CPENTA, and CPYRAM elements, the determinant of the Jacobian at the Gauss points is calculated. For the CTETRA element, the calculation occurs at both the Gauss points and at the corners.</p> <p>b. The sign of DET J at the first calculated Gauss or corner point is determined. If it is negative, then the value of DET J for all points is multiplied by -1. This step is not a requirement for the Jacobian check, but is done for elements having reversed connectivity. 3D-Solid elements which have a consistent Jacobian at the Gauss points but with reversed connectivity will have a negative DET J at all Gauss points. To allow for this type of connectivity, the software simply reverses the sign of DET J by multiplying by -1.</p> <p>c. The sign of all DET J is checked for consistency (all positive or all negative). A failure occurs if the sign of any DET J is inconsistent.</p> <p>d. Out of all the elements failing the Jacobian check, the element which has the most negative DET J is reported.</p>
<p>Volume Check.</p> <p>Occurs for the CTETRA and CPYRAM elements.</p>	<p>A failure occurs if $Volume \leq 0.0$.</p>
Axisymmetric Element System Checks	

Jacobian check. Occurs for the CTRAX3, CTRAX6, CQUADX4 and CQUADX8 elements.	The determinant of the Jacobian at the Gauss points is calculated. A failure occurs if any $DET J \leq 0.0$.
Consistent Y check when elements are modeled in XZ plane. Consistent Z check when elements are modeled in XY plane. Occurs for the CTRAX3, CTRAX6, CQUADX4 and CQUADX8 elements.	When axisymmetric elements are modeled in the XZ plane, the grids used must all lie in the same Y-plane within the tolerance $\pm 1.E-6$, or a failure occurs. When axisymmetric elements are modeled in the XY plane, the grids used must all lie in the same Z-plane within the tolerance $\pm 1.E-6$, or a failure occurs.
$X \geq 0.0$ check when elements are modeled in XZ plane. $Y \geq 0.0$ check when elements are modeled in XY plane. Occurs for the CTRAX3, CTRAX6, CQUADX4 and CQUADX8 elements.	When axisymmetric elements are modeled in the XZ plane, the X location for all grids must be ≥ 0.0 or a failure occurs. When axisymmetric elements are modeled in the XY plane, the Y location for all grids must be ≥ 0.0 or a failure occurs.
Plane Stress and Plane Strain Element System Checks	
Jacobian check. Occurs for the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8 and the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.	The determinant of the Jacobian at the Gauss points is calculated. A failure occurs if any $DET J \leq 0.0$.
Consistent Y check when elements are modeled in XZ plane. Consistent Z check when elements are modeled in XY plane. Occurs for the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8 and the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.	When plane stress or plane strain elements are modeled in the XZ plane, the grids used must all lie in the same Y-plane within the tolerance $\pm 1.E-6$, or a failure occurs. When plane stress or plane strain elements are modeled in the XY plane, the grids used must all lie in the same Z-plane within the tolerance $\pm 1.E-6$, or a failure occurs.

User Controlled Element Checks

The GEOMCHECK executive statement is used to control the optional, user controlled element checks. By default, these checks do not produce fatal errors and occur using default values even when the GEOMCHECK statement is not included. GEOMCHECK controls are available for the shell elements CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, the solid elements CHEXA, CPENTA, CPYRAM, CTETRA, the axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6, the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, the CBAR element, and the CBEAM element.

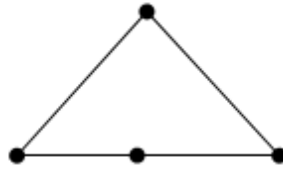
The GEOMCHECK executive control statement format is as follows:

GEOMCHECK test_keyword [= tol_value], [MSGLIMIT=n], [MSGTYPE = FATAL
INFORM
WARN], [SUMMARY], [NONE]

Also see **GEOMCHECK** in the *NX Nastran Quick Reference Guide*.

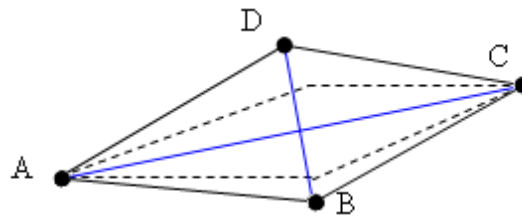
The following table describes the user controlled element checks in detail.

Shell Element GEOMCHECK Descriptions
<p>Quad Skew for CQUAD4, CQUAD8, and CQUADR elements.</p> <p>Test_keywords: Q4_SKEW, Q8_SKEW</p> <div data-bbox="574 669 974 892" data-label="Image"> <p>The diagram shows a quadrilateral with vertices P, Q, R, and S. Vectors are drawn from P to Q and from R to S. The angle between these two vectors is labeled as θ.</p> </div> <p>The smallest angle θ between vectors joining side mid-points PQ and RS.</p> <p>Default: Fails if $\theta < 30.0$ degrees</p>
<p>Quad Taper for CQUAD4, CQUAD8, and CQUADR elements.</p> <p>Test_keywords: Q4_TAPER, Q8_TAPER</p> <div data-bbox="597 1155 1024 1421" data-label="Image"> <p>The diagram shows a quadrilateral with vertices A, B, C, and D. Diagonals AC and BD are drawn, intersecting at a point.</p> </div> <p>The element is transformed to a mean plane such that grids A, B, C, and D all lie in one plane. Taper is given by</p> $(A_{\max} - Q) / Q \text{ (non-dimensional)}$ <p>where A_{\max} is the largest of the four corner triangular areas ABD, BCA, CDB, DAC, and $Q = 0.5 * \text{total quad area}$.</p> <p>Default: Fails if Q4_TAPER or Q8_TAPER > 0.5</p> <p>Note: When a quad element is in the shape of a triangular as shown below, the taper value reaches 1.0.</p>



Quad Warp for CQUAD4 and CQUADR elements.

Test_keyword: Q4_WARP



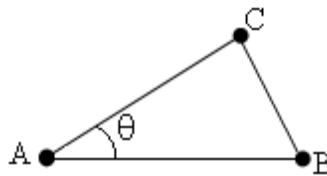
Quad warp is defined as the ratio of HH to the sum of diagonal lengths.

$W = HH / (D_{AC} + D_{BD})$, where HH is the dot product $AB \cdot PK$, $PK = (AC \times BD) / |AC \times BD|$ and D_{AC} and D_{BD} are distances from A to C and from B to D. In more detail, the normalized PK is an out-of-plane unit vector. HH is the amount vector AB is projected onto vector PK. If an element is perfectly flat, HH will be zero (no projection) and there is no warp. Dividing HH by the sum of diagonal lengths D_{AC} and D_{BD} makes warp independent of the element size.

Default: Fails if $W > 0.05$

Tria Skew for CTRIA3 and CTRIAR elements.

Test_keyword: T3_SKEW

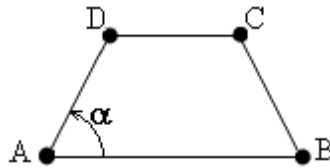


The smallest included (interior) angle θ from all three corners.

Default: Fails if $\theta < 10.0$ degrees

Interior Angle Checks for CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, and CTRIAR elements.

Test_keywords: Q4_IAMAX, Q4_IAMIN, Q8_IAMIN, Q8_IAMAX, T3_IAMAX, TA6_IAMN, TA6_IAMX



The extreme interior angles are designated by α_{\min} & α_{\max} .

Defaults:

For CQUAD4, CQUAD8, and CQUADR: Fails if $\alpha_{\max} > 150.0$ degrees or $\alpha_{\min} < 30.0$ degrees

For CTRIA3 and CTRIAR: Fails if $\alpha_{\max} > 160.0$ degrees

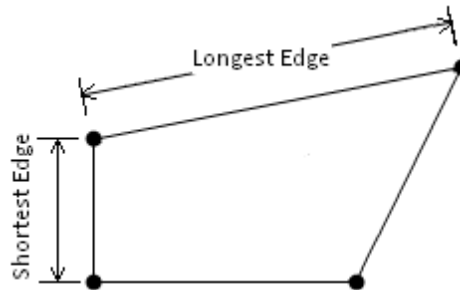
For CTRIA6: Fails if $\alpha_{\max} > 160.0$ degrees or $\alpha_{\min} < 10.0$ degrees

Note: α_{\min} for CTRIA3 and CTRIAR is checked by T3_SKEW.

Aspect Ratio for CQUAD4, CQUAD8, and CTRIA6 elements.

Test_keywords: QAD_AR, Q8_AR, TA6_AR

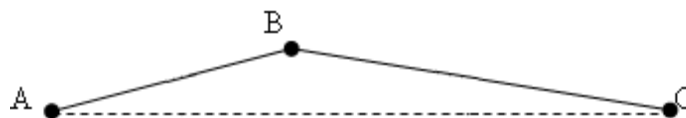
Aspect Ratio = longest element edge / shortest element edge



Default: Fails if Aspect Ratio > 100.0

Edge-Point-Length-Ratio for CQUAD8 and CTRIA6 elements.

Test_keywords: Q8_EPLR, TA6_EPLR



Default: Fails if $\text{Min}(AB/BC, BC/AB) < 0.5$

Note: A, B, C are grid locations on edge ABC.

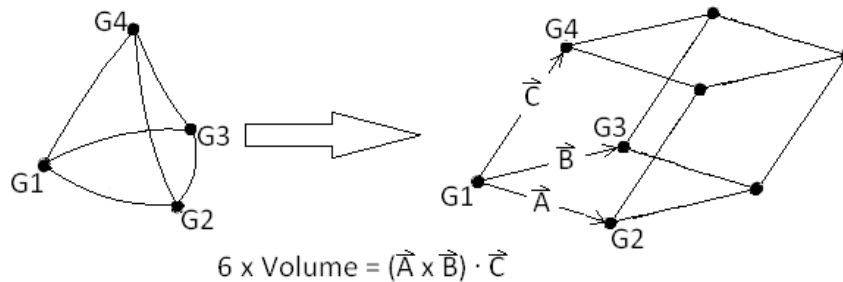
3D-Solid GEOMCHECK Descriptions

Aspect Ratio for CTETRA, CHEXA, CPENTA, and CPYRAM elements.

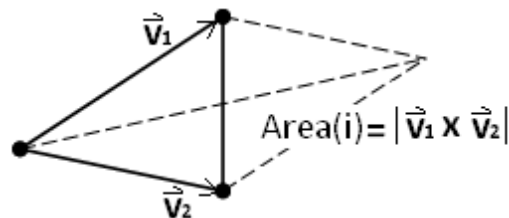
Test_keywords: TET_AR, HEX_AR, PEN_AR, PYR_AR

Default: Fails if AR > 100.0 for all 3-D Solid elements

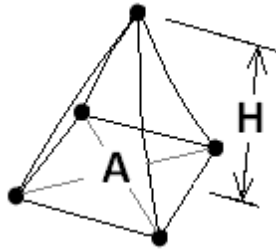
- For HEX_AR and PEN_AR, it is defined by
AR = longest element edge / shortest element edge
- For TET_AR, it is defined by
AR = longest element edge / shortest height
The shortest height is the minimum of H1, H2, H3, and H4
where $H_i = (6 * \text{Element Volume}) / \text{Area}(i)$,
and $(6 * \text{Element Volume})$ is a parallelepiped volume calculated by



Area(i) is calculated for each of the four element faces and is the magnitude of the cross product of two of the edge vectors on a face. The Area(i) calculation result is equivalent to the area of a parallelogram.



- For PYR_AR, it is defined by
AR = longest element edge / shortest element edge or pyramid height
The shortest element edge or height is the smallest of one of the eight element edges or the element height.



The element height is calculated by $H = (3 * \text{Element Volume}) / A$

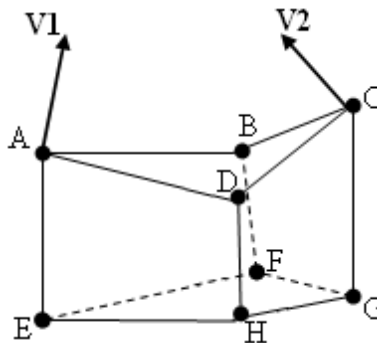
where A is the area of the quad face.

The element volume is calculated by dividing the pyramid element into two tetrahedral elements. The tetrahedral volumes are calculated as described under the TET_AR description above, each divided by 6, then added together to obtain the pyramid total volume.

Note: Unlike the other solid element aspect ratio checks, the PYR_AR check calculates the element volume explicitly. A failure occurs if Volume = 0.0. A CPYRAM element volume check also occurs in the **system element checks**.

Face-Planarity for CHEXA, CPENTA, and CPYRAM elements.

Test_keywords: HEX_WARP, PEN_WARP, PYR_WARP

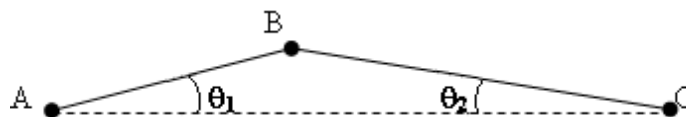


In the example above, the hex element face ABCD is heavily warped. The face-planarity criterion is defined as the cosine of the angle θ made by opposite corner face vectors V1 and V2.

Default: Fails if $\text{COS}(\theta) < 0.707$

Edge-Point-Length-Ratio for CTETRA, CHEXA, CPENTA, and CPYRAM elements with midside grids.

Test_keywords: TET_EPLR, HEX_EPLR, PEN_EPLR, PYR_EPLR



Default: Fails if:

a) $\text{Min}(AB/BC, BC/AB) < 0.5$ or

b) $\theta_1 + \theta_2 > 30.0$ degrees

Note: Only the threshold for a) can be modified using the test_keyword descriptors. The threshold for b) is always 30 degrees. Although, the other GEOMCHECK descriptors such as NONE and MSGTYPE apply to both a) and b).

Note: A, B, C are grid locations on edge ABC.

Ratio of longest edge to shortest edge in stacking direction for CHEXA and CPENTA elements that reference PCOMPS entries only.

Test_keywords: HEX_TP, PEN_TP

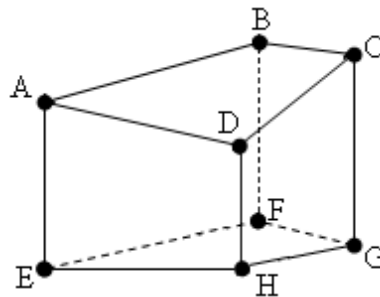
- For HEX_TP

Assume edges AE, BF, CG, and DH coincide with the stack direction

CG is the longest

DH is the shortest

Default: Fails if $CG/DH > 1.2$



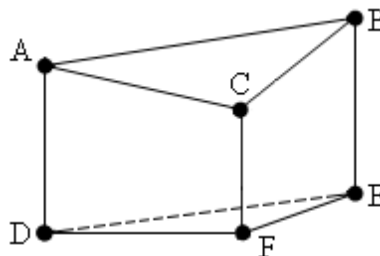
- For PEN_TP

Assume edges AD, BE, and CF coincide with the stack direction

BE is the longest

CF is the shortest

Default: Fails if $BE/CF > 1.2$



Ratio of the difference between thickness in stacking direction as defined by grids and as defined by ply thickness specification to the thickness in the stacking direction as defined by grids for CHEXA and CPENTA elements that reference PCOMPS entries only.

Test_keywords: HEX_TK, PEN_TK

- For HEX_TK

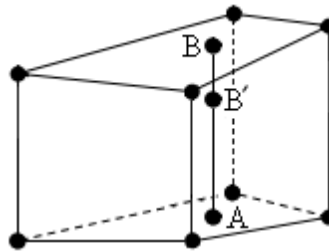
Assume AB coincides with the stack direction

A and B are located at the center of the bottom and top faces of the element, respectively

AB is thickness as defined by grids.

AB' is thickness as defined by ply thickness specification.

Default: Fails if $|AB - AB'|/AB > 0.05$



- For PEN_TK

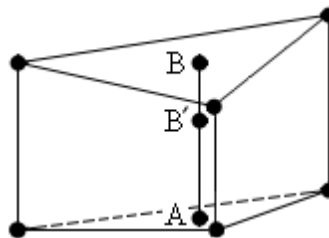
Assume AB coincides with the stack direction

A and B are located at the center of the bottom and top faces of the element, respectively

AB is thickness as defined by grids.

AB' is thickness as defined by ply thickness specification.

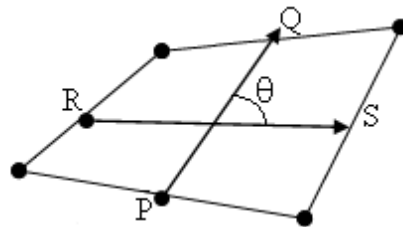
Default: Fails if $|AB - AB'|/AB > 0.05$



Axisymmetric, Plane Stress, and Plane Strain GEOMCHECK Descriptions

Skew for the axisymmetric elements CQUADX4 and CQUADX8, the plane stress elements CPLSTS4 and CPLSTS8, the plane strain elements CPLSTN4 and CPLSTN8.

Test_keyword: QDX_SKEW

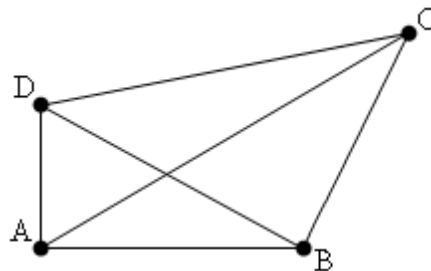


The smallest angle θ between vectors joining side mid-points PQ and RS.

Default: Fails if $\theta < 30.0$ degrees

Taper for the axisymmetric elements CQUADX4 and CQUADX8, the plane stress elements CPLSTS4 and CPLSTS8, the plane strain elements CPLSTN4 and CPLSTN8.

Test_keyword: QDX_TAPER



The element is transformed to a mean plane such that grids A, B, C, and D all lie in one plane. Taper is defined as

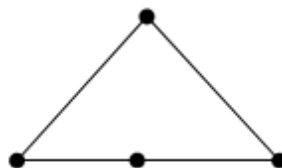
$$Q4_TAPER = (A_{\max} - Q) / Q \text{ (non-dimensional)}$$

where A_{\max} is the largest of the four corner triangular areas ABD, BCA, CDB, DAC

and $Q = .5 * \text{total quad area}$.

Default: Fails if $Q4_TAPER > 0.5$

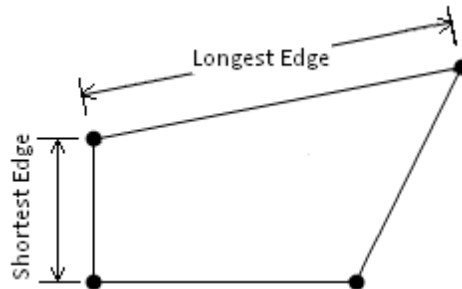
Note: When a quad element is in the shape of a triangular as shown below, the taper value reaches 1.0.



Aspect Ratio for the axisymmetric elements CTRAX3, CTRAX6, CQUADX4, CQUADX8, the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, and the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.

Test_keyword: TRX_AR, QDX_AR

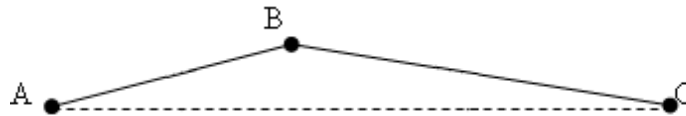
Aspect Ratio = longest element edge / shortest element edge



Default: Fails if Aspect Ratio > 100.0

Edge-Point-Length-Ratio for the axisymmetric elements CTRAX6 and CQUADX8, the plane stress elements CPLSTS6 and CPLSTS8, the plane strain elements CPLSTN6 and CPLSTN8.

Test_keywords: TRX_EPLR, QDX_EPLR

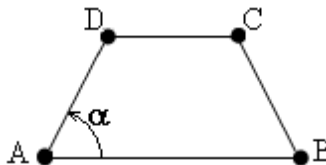


Default: Fails if $\text{Min}(AB/BC, BC/AB) < 0.5$

Note: A, B, C are grid locations on edge ABC.

Interior Angle Checks for axisymmetric elements CTRAX3, CTRAX6, CQUADX4, CQUADX8, the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.

Test_keywords: QDX_IAMX, QDX_IAMN, TRX_IAMN, TRX_IAMX



The extreme interior angles are designated by α_{\min} & α_{\max} .

Defaults:

For the axisymmetric elements CQUADX4 and CQUADX8, the plane stress elements CPLSTS4 and CPLSTS8, the plane strain elements CPLSTN4 and CPLSTN8:

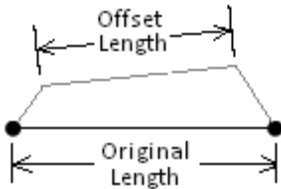
Fails if $\alpha_{\max} > 150.0$ degrees or $\alpha_{\min} < 30.0$ degrees

For the axisymmetric elements CTRAX3 and CTRAX6, the plane stress elements CPLSTS3 and CPLSTS6, the plane strain elements CPLSTN3 and CPLSTN6:
Fails if $\alpha_{\max} > 160.0$ degrees or $\alpha_{\min} < 10.0$ degrees

CBAR and CBEAM GEOMCHECK Descriptions

BAR and BEAM OFFSET Ratio for CBAR and CBEAM elements with offsets defined.

Test_keywords: BAR_OFF, BEAM_OFF



BAR and BEAM OFFSET Ratio = $\text{ABS}(\text{Offset Length} - \text{Original Length}) / \text{Original Length}$

Default: Fails if Ratio > 0.15

Sample Output

The following output from a small example provides a sample of the message formats that are produced for geometry tests that can be controlled using the GEOMCHECK statement. For this output, most of the default test tolerances were modified so that the effect could be observed in the tolerance information lines of the messages that were produced.

Note that a small table is generated at the end of all messages that summarizes the number of tests that actually exceeded the tolerance value for each element type as well as a list of the elements that produced the worst violations. The summary table itself can be produced by using the keyword SUMMARY on the GEOMCHECK statement.

```

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
  FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
  User Action: Use the GEOMCHECK (Executive Control Statement) keyword-value to change tolerance values if desired.
  A MINIMUM OF      100 OFFSET LENGTH RATIO  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.

TOLERANCE LIMIT IS:  BAR_/BEAN_OFF(SET) =      .15 (XXXX = LIMIT VIOLATED)
ELEMENT TYPE  ID    LENGTH W/O OFFSET  LENGTH W/ OFFSET  OFFSET LENGTH RATIO
BAR           7101    1.00000E+00      3.66367E+00      2.66 XXXX

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
  FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
  A MINIMUM OF      4 OFFSET LENGTH RATIO  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

TOLERANCE LIMIT IS:  BAR_/BEAN_OFF(SET) =     -0.02 (WARN = LIMIT VIOLATED)
ELEMENT TYPE  ID    LENGTH W/O OFFSET  LENGTH W/ OFFSET  OFFSET LENGTH RATIO
BEAM          8101    1.00000E+00      1.10372E+01      10.04 WARN
BEAM          8102    1.00000E+00      1.00000E+00       .00 WARN

*** USER INFORMATION MESSAGE 7555 (ENEXGD)
  FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
  A MINIMUM OF      100 EDGE LENGTH ASPECT RATIO (AR)  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 EDGE NODE POINT LENGTH RATIO (ER) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  EDGE ANGLE VALUES ARE INDICATED BY AN * AFTER THE VALUE.  THE TOLERANCE LIMIT IS 30 DEGREES.
  ALL NEG/ZERO DET(JACOBIAN) VALUE (DJ) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY FAIL.
  A MINIMUM OF      100 WARPED FACE COEFFICIENT  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.

TOLERANCE LIMITS ARE:  HEX_AR =    300.00, HEX_EPLR =    .40, HEX_DETJ =    .01, HEX_WARD =    .65 (XXXX = LIMIT VIOLATED)
ELEMENT TYPE  ID    LONGEST  SHORTEST  ASPECT  EDGE POINT  MIN. JACOBIAN  FACE WARP
HEXA          8602    4.00      1.00      4.00      N/A- ONLY 8 NODE    .92      .52 XXXX
HEXA          8620    4.00      1.00      4.00      .60XXXXX          .50      1.00

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
  FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
  A MINIMUM OF      100 SKEW ANGLE (SA)  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 MIN INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 MAX INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 WARPING FACTOR (WF) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 TAPER RATIO (TR)  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.

TOLERANCE LIMITS ARE:  SA =    30.00, IA(MIN) =    30.00, IA(MAX) =   150.00, WF =    .05, TR =    .50 (XXXX = LIMIT VIOLATED)
ELEMENT TYPE  ID    SKEW ANGLE  MIN INT. ANGLE  MAX INT. ANGLE  WARPING FACTOR  TAPER RATIO
QUAD4         105    29.74 XXXXX    29.05 XXXXX    150.95 XXXXX    .00      .05
QUAD4         106    28.39 XXXXX    27.76 XXXXX    152.24 XXXXX    .00      .05
QUAD4         107    27.15 XXXXX    26.57 XXXXX    153.43 XXXXX    .00      .05
QUAD4         108    26.00 XXXXX    25.46 XXXXX    154.54 XXXXX    .00      .05
QUAD4         109    24.94 XXXXX    24.44 XXXXX    155.56 XXXXX    .00      .05
QUAD4        2105    29.74 XXXXX    29.05 XXXXX    150.95 XXXXX    .00      .05
QUAD4        2106    28.39 XXXXX    27.76 XXXXX    152.24 XXXXX    .00      .05
QUAD4        2107    27.15 XXXXX    26.57 XXXXX    153.43 XXXXX    .00      .05
QUAD4        2108    26.00 XXXXX    25.46 XXXXX    154.54 XXXXX    .00      .05
QUAD4        2109    24.94 XXXXX    24.44 XXXXX    155.56 XXXXX    .00      .05

*** USER INFORMATION MESSAGE 7555 (GMTSTD)
  FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.
  A MINIMUM OF      100 SKEW ANGLE (SA)  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 MIN INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 MAX INT. ANGLE (IA) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 WARPING FACTOR (WF) TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.
  A MINIMUM OF      100 TAPER RATIO (TR)  TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY XXXX.

```

TOLERANCE LIMITS ARE: SA = 30.00, IA(MIN) = 30.00, IA(MAX) = 150.00, WF = .05, TR = .50 (XXXX = LIMIT VIOLATED)

ELEMENT TYPE	ID	SA	SKEN ANGLE	MIN INT. ANGLE	MAX INT. ANGLE	WARPING FACTOR	TAPER RATIO
QUADR	1105	29.74	XXXX	29.05	XXXX	.00	.05
QUADR	1106	28.39	XXXX	27.76	XXXX	.00	.05
QUADR	1107	27.15	XXXX	26.57	XXXX	.00	.05
QUADR	1108	26.00	XXXX	25.46	XXXX	.00	.05
QUADR	1109	24.94	XXXX	24.44	XXXX	.00	.05

*** USER INFORMATION MESSAGE 7555 (ETETOD)

FINITE ELEMENT GEOMETRY CHECK RESULTS EXCEED TOLERANCE LEVELS FOR THE FOLLOWING ELEMENTS.

A MINIMUM OF	4	EDGE LENGTH TO HEIGHT RATIO (AR)/TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.
A MINIMUM OF	4	EDGE NODE POINT LENGTH RATIO (ER)/TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.
A MINIMUM OF	4	EDGE ANGLE VALUES ARE INDICATED BY AN * AFTER THE VALUE. THE TOLERANCE LIMIT IS 30 DEGREES.
A MINIMUM OF	4	NEG/ZERO DET(JACOBIAN) VALUE (DJ)/TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.
A MINIMUM OF	4	NEG DET(JAC) AT VERTEX NODE (DG)/TOLERANCE LIMIT VIOLATIONS WILL BE IDENTIFIED AND INDICATED BY WARN.

TOLERANCE LIMITS ARE: TET_AR = 351.00, TET_EPLR = .45, TET_DETJ = -.01, TET_DETJ = -.02 (XXXX = LIMIT VIOLATED)

ELEMENT TYPE	ID	LONGEST EDGE	SHORTEST HEIGHT	ASPECT RATIO	EDGE POINT LENGTH RATIO	MIN. JACOBIAN DETERMINANT	MIN. DET(JAC) AT VERTEX
TETRA	6601	1.41	.58	2.45	.25 WARN	.14	-.20 WARN

ELEMENT GEOMETRY TEST RESULTS SUMMARY

TOTAL NUMBER OF TIMES TOLERANCES WERE EXCEEDED

ELEMENT TYPE	SKEN ANGLE	ASPECT/ MINIMUM	INTER. ANGLE	INTER. ANGLE	WARP FACTOR	OFFSET RATIO	EDGE POINT LENGTH RATIO	JACOBIAN DETERMINANT
BAR	N/A	N/A	N/A	N/A	N/A	0	N/A	N/A
BEAM	N/A	N/A	N/A	N/A	N/A	0	N/A	N/A
HEXA	N/A	0	N/A	N/A	1	N/A	1	0
PENTA	N/A	0	N/A	N/A	0	N/A	1	0
QUAD4	10	0	10	10	0	N/A	N/A	N/A
QUADR	5	0	5	5	0	N/A	N/A	N/A
TETRA	N/A	0	N/A	N/A	N/A	N/A	1	1
TRIA3	0	N/A	N/A	0	N/A	N/A	N/A	N/A
TRIA4	0	N/A	N/A	0	N/A	N/A	N/A	N/A

N/A IN THE ABOVE TABLE INDICATES TESTS THAT ARE NOT APPLICABLE TO THE ELEMENT TYPE AND WERE NOT PERFORMED.

FOR ALL ELEMENTS WHERE GEOMETRY TEST RESULTS HAVE EXCEEDED TOLERANCES,

HEXA	ELEMENT ID	8620	PRODUCED	SMALLEST EDGE COS(ANGLE)	OF	.60 (TOLERANCE = .97)
HEXA	ELEMENT ID	8602	PRODUCED	SMALLEST FACE WARP FACTOR	OF	.52 (TOLERANCE = .65)
PENTA	ELEMENT ID	3915	PRODUCED	SMALLEST EDGE COS(ANGLE)	OF	.78 (TOLERANCE = .87)
QUAD4	ELEMENT ID	109	PRODUCED	SMALLEST SKEW ANGLE	OF	24.94 (TOLERANCE = 30.00)
QUAD4	ELEMENT ID	109	PRODUCED	SMALLEST INTERIOR ANGLE	OF	24.44 (TOLERANCE = 30.00)
QUAD4	ELEMENT ID	109	PRODUCED	LARGEST INTERIOR ANGLE	OF	155.56 (TOLERANCE = 150.00)
QUADR	ELEMENT ID	1109	PRODUCED	SMALLEST SKEW ANGLE	OF	24.94 (TOLERANCE = 30.00)
QUADR	ELEMENT ID	1109	PRODUCED	SMALLEST INTERIOR ANGLE	OF	24.44 (TOLERANCE = 30.00)
QUADR	ELEMENT ID	1109	PRODUCED	LARGEST INTERIOR ANGLE	OF	155.56 (TOLERANCE = 150.00)
TETRA	ELEMENT ID	6601	PRODUCED	LARGEST EDGE POINT LR	OF	.25 (TOLERANCE = .45)
TETRA	ELEMENT ID	6601	PRODUCED	SMALLEST VERTEX DET(JAC)	-UWN 6828- OF	-.20 (TOLERANCE = -.02)

Weight Center of Gravity and Moment of Inertia Check

You can use the Grid Point Weight Generator (GPWG) to check whether the weight of your model is correct. To turn on the Grid Point Weight Generator, include the parameter GRDPNT in the Bulk Data or Case Control Section of your input file.

The data entry format is

```
PARAM,GRDPNT,i
```

where "i" is an integer value defining a reference point. The value of "i" can be any grid point in the model, or if it is set to zero, the reference point is the origin of the basic coordinate system.

The output from the GPWG includes a rigid body mass matrix, various coordinate transformations, and the location of the center of mass. The output from the GPWG is generally more than what you need. The mass and center of gravity (CG) location is typically all that is used. A partial output for the problem in Listing 15-5 is shown in Figure 15-15.

DIRECTION					
MASS AXIS SYSTEM (S)	MASS	X-C.G.	Y-C.G.	Z-C.G.	
X	3.375000E+01	0.000000E+00	0.000000E+00	0.000000E+00	
Y	3.375000E+01	2.500000E+00	0.000000E+00	0.000000E+00	
Z	3.375000E+01	2.500000E+00	0.000000E+00	0.000000E+00	

Figure 15-15. Partial Output from the Grid Point Weight Generator

NX Nastran does not keep track of your units. Therefore, you must input all of your properties using a consistent set of units. For example, if you are using meters (m) for defining locations on your grid entries, then your properties, such as areas (A), should be in terms of m². NX Nastran expects your mass input (MATi, CONMi, etc.) to be in terms of mass units. However, if you would rather input your mass in terms of weight units, then you must add the following entry to your Bulk Data Section:

```
PARAM,WTMASS,x
```

In the parameter statement above, x is the appropriate conversion factor based on the units that you are using with a default value of 1.0. In other words, the weight density is related to the mass density by the relationship

$$\rho_m = \left(\frac{1}{g}\right) \cdot \rho_w = \text{WTMASS} \cdot \rho_w$$

where:

ρ_w = weight density

ρ_m = mass density

g = gravitational acceleration constant

In the example below, the mass came from the density entered on the MAT1 entry associated with the CBAR elements. In this case, the density is in terms of mass units. Since the grid point locations are defined in terms of meters, the density should then be in units of kg/m³. Using a consistent set of units, the Young's modulus (E) and density (ρ) are 7.1E10 N/m² and 2700 kg/m³ as indicated by Fields 3 and 6, respectively, on the following MAT1 entry.

```
$  MASS DENSITY (kg/m3)
$
MAT1      1      7.1+10      .33      2700.
```

On the other hand, if you want to input your density in weight units, then you should replace the above MAT1 entry with the following two entries.

```
$  WEIGHT DENSITY (N/m3)
$  ACCELERATION (m/sec2)
$
MAT1      1      7.1+10      .33      2.65+4
PARAM    WTMASS  0.1019
```

This density and all the other masses (excluding M2PP or M2GG input) are multiplied by this scale factor. In this case, (0.1019 · 2.65E4) = 2700. The results (e.g., displacements, stresses, etc.) are the same using either of the above mass or weight units.

The only difference is in the grid point weight generator output. The GPWG output is in the same units of your density or point masses. In other words, if your input is in mass units, then the GPWG output is also in mass units. If your input is in weight units, then the GPWG output is also in weight units.

If you are using English units, then the following two sets of entries yield the same results. Of course, other units in your model must also be consistent, e.g., the grid point locations and cross-sectional properties.

```
$  MASS DENSITY (lb-sec2/in4)
$
MAT1      1      1.0 +7      .33      2.51-4
```



```

$  WEIGHT DENSITY (lb/in3)
$  ACCELERATION (in/sec2)
$
MAT1      1      1.0 +7      .33      0.097
PARAM     WTMASS  0.002588

```

The output from GPWG is for informational purposes only and is not used in any subsequent steps in the solution process.

See also

- “Grid Point Weight Generator” in the *NX Nastran Basic Dynamic Analysis User's Guide*

Mechanisms and Singularities

When performing the solution to a system of linear equations, singularities lead to conditions in which a unique solution is not possible. NX Nastran considers two types of singularities:

- Grid point singularity that is identified by considering the stiffness terms of only one grid point.
- A mechanism type of singularity that requires the consideration of the stiffness terms of more than one grid point.

Singularities cause ill-conditioned matrices that can be detected in several phases of the NX Nastran execution.

After matrix assembly, the grid point singularities are detected. At each grid point, a 3X3 partition of the stiffness matrix for each of the three translational and three rotational DOFs is solved as an eigenvalue problem to determine the principal stiffnesses. Each stiffness term is compared to the principal stiffness using the formula

$$\varepsilon = \frac{K_{ii}}{K_{max}}$$

where K_{ij} is the term in the i -th row and j -th column of the matrix and K_{max} is the principal stiffness. If ε is less than the value of PARAM,EPZERO, the global direction nearest i is considered singular. The default value for EPZERO is 10^{-8} . A list of potential singularities is printed in the grid point singularity table (see “[Strain Energy Output](#)” for further details).

PARAM,AUTOSPC. If PARAM,AUTOSPC,YES is specified (this is the default in the Structured Solution Sequences, except SOLs 106 and 129), the potential singularities are automatically constrained if possible.

During decomposition, mechanisms can be detected based on the maximum ratio of the matrix diagonal to the factor diagonal

$$\text{MAXRATIO} = \frac{K_{ii}}{D_{ii}}$$

where K_{ii} is the i -th diagonal term of the original stiffness matrix and D_{ii} is the i -th diagonal term of the factor diagonal matrix. For a symmetric matrix K it can be represented as

$$[K] = [L][D][L^T]$$

where:

$[L]$ = lower triangular factor

$[D]$ = factor diagonal matrix

PARAM,MAXRATIO. Refer to the *NX Nastran Numerical Methods User's Guide* for further details on the subject of system of linear equations. All terms whose ratio exceed the value of PARAM,MAXRATIO are printed. The default for MAXRATIO is 10^7 . UIM 4158 prints the statistics for the decomposition that include the number of negative terms on the factor diagonal and the maximum ratio of matrix diagonal to factor diagonal at a specified row number and its corresponding grid point. User Warning Message (UWM) 4698 prints the degrees of freedom that have a factor diagonal ratio greater than the MAXRATIO value or have negative terms on the factor diagonal. Both of these messages are issued by the DECOMP module and are shown below:

```
*** USER INFORMATION MESSAGE 4158---STATISTICS FOR SYMMETRIC DECOMPOSITION OF DATA BLOCK KLL      FOLLOW
      NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL =      1
      MAXIMUM RATIO OF MATRIX DIAGONAL TO FACTOR DIAGONAL = 7.2E+15 AT ROW NUMBER      16
*** USER WARNING MESSAGE 4698.  STATISTICS FOR DECOMPOSITION OF MATRIX KLL

THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN 1.00000E+05 OR HAVE
NEGATIVE TERMS ON THE FACTOR DIAGONAL.
      GRID POINT ID      DEGREE OF FREEDOM      MATRIX/FACTOR DIAGONAL RATIO      MATRIX DIAGONAL
      6714      T1      -7.19297E+15      6.02908E+07
^^^ DMAP FATAL MESSAGE 9050 (SEKRRS) - RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS
    IN MATRIX KLL. USER PARAMETER BAILOUT MAY BE USED      TO CONTINUE THE RUN.
```

If the MAXRATIO value is exceeded, your job terminates with DMAP Fatal Message 9050. You can override this fatal message by inserting "PARAM,BAILOUT,-1" in your input file. You should, however, be aware that a large value of the MATRIX/FACTOR DIAGONAL may be an indication of a potential modeling problem. Taking the \log_{10} of MAXRATIO indicates how many significant digits may have been lost during the decomposition. The MAXRATIO may change slightly if a different sequencer is used. This change is due to the fact that the order of operation may change with a different sequencer that yields a different numerical roundoff.

After decomposition, a singularity may lead to an incorrect solution. In static analysis, NX Nastran solves

$$Ku = P$$

to obtain u (displacements). Using these displacements, NX Nastran then calculates a “residual” loading vector as follows:

$$Ku - P = \delta P$$

This residual vector should theoretically be null but may not be null due to numeric roundoff. To obtain a normalized value of the residual loading, an error measure ε is calculated by

$$\varepsilon = \frac{u^T \cdot \delta P}{u^T \cdot P}$$

δP can be printed by including PARAM,IRES,1 in your Bulk Data Section. The value ε (epsilon) is printed in User Information Message (UIM) 5293 as shown below. Epsilon values that are greater than 0.001 are flagged for a possible loss of accuracy due to numeric conditioning as shown below:

*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL			
LOAD SEQ. NO.	EPSILON	EXTERNAL WORK	EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
1	-4.5961966E-14	5.6341639E+00	
2	-9.2748066E-15	9.4523830E+00	

One ε is generated for each loading condition. An acceptable value of ε depends on the model complexity and the machine that it runs on. An epsilon value of $|\varepsilon|$ in the neighborhood of less than 10^{-9} is generally considered acceptable.

Some general causes for singularity can include:

- Degrees of freedom without stiffness because of missing elements.
- A 2-dimensional plate problem with the normal rotation unconstrained.
- A solid model with rotational DOFs at the corners unconstrained.
- Incorrect modeling of offset beams.
- Incorrect multipoint constraints.
- Mechanisms and free bodies, such as sloped plates, beam to plate connections, beam to solid connections, and plate-to-solid connections.
- Low stiffness in rotation.
- A stiff element adjacent to a very flexible element.

Applied Loads Check

There are two checks in NX Nastran to let you verify that loads are being applied correctly to the structure. The first one is the summation of the total loads applied to the structure about the reference point x as specified on the PARAM,GRDPNT, x entry. If PARAM,GRDPNT, x is not specified, this

summation is performed about the origin of the basic coordinate system. This quantity is known as the OLOAD RESULTANT output and will output by default. It can be deactivated by adding PARAM,PRTRESLT,NO in the bulk data. It consists of seven lines of output per load case. The OLOAD RESULTANT output for the problem in Listing 15-5 is shown in Figure 15-15. Note that loads from scalar points are not included in the OLOAD RESULTANT output.

0		OLOAD		RESULTANT			
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
0 1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
0 2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	3.307500E+02	0.000000E+00	-8.268750E+02	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-8.268750E+02	0.000000E+00

Figure 15-16. OLOAD Resultant

T1, T2, T3, R1, R2, R3 are the basic coordinate directions of the summed loads in the table. The row headings Fx, Fy, Fz, Mx, My, Mz are relative to the applied loads, also in basic coordinates.

Look at Subcase 1 from the problem in Listing 15-5 which has a vertical tip load applied to grid point 6 in the +z direction. Since there are no loads applied to the structure in the x- or y-direction, the loads in those two directions are equal to zero. The total z-load is equal to 100 Newtons contributed solely by the applied load at grid point 6. The total moment about the origin of the basic coordinate system is $-5 \cdot (100 \text{ N}) = -500 \text{ NM}$ about the y-axis. This R2 moment is contributed by the Z load as indicated by the -500.0 in row FZ, column R2. There is no moment about the other two axes. The next seven lines in the OLOAD RESULTANT output is due to the gravity load in subcase 2.

So far, only the resultant applied load was discussed. You can also request more detailed information regarding the exact location on the structure where loads are applied. This can be done with the OLOAD request in the Case Control Section. The format is as follows:

```
OLOAD = All
```

or

```
SET 10 = 1,2,3,6
OLOAD = 10
```

In the case of OLOAD = All, a printout of all the applied loads is requested. As shown in Figure 15-17, there is one load applied to grid point 6 in the +z direction for Subcase 1. However, for the gravity load in Subcase 2, each grid point that has mass associated with it has a 1g load applied to it. Each CBAR element contributes $\{(2700)(1.0)(2.5\text{E-}3)(9.8)/2\} = 33.075 \text{ N}$ to each end of its connecting grid points. Therefore, a 33.075 N load is applied to the end points and a 66.15 N load is applied to each of the intermediate points since there are two CBAR elements connected to each of these intermediate grid points. As you can see, for a large model, OLOAD output for gravity loads can potentially generate many lines of output.

VERTICAL TIP LOAD								SUBCASE 1
LOAD VECTOR								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
6	G	.0	.0	1.000000E+02	.0	.0	.0	
GRAVITY LOADS								SUBCASE 2
LOAD VECTOR								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	3.307500E+01	.0	.0	.0	
2	G	.0	.0	6.615000E+01	.0	.0	.0	
3	G	.0	.0	6.615000E+01	.0	.0	.0	
4	G	.0	.0	6.615000E+01	.0	.0	.0	
5	G	.0	.0	6.615000E+01	.0	.0	.0	
6	G	.0	.0	3.307500E+01	.0	.0	.0	

Figure 15-17. OLOAD Output

Verifying Reaction Loads

There are two checks available in NX Nastran to let you check whether the reaction loads are correct. The first one is the summation of the total reaction loads about the reference point specified on the PARAM,GRDPNTx entry. If PARAM,GRDPNT,x is not specified, this summation is performed about the origin of the basic coordinate system. This operation is known as the SPCFORCE RESULTANT output and this output is obtained if the SPCFORCE output is requested. It consists of seven lines of output per load case. The SPCFORCE RESULTANT output for the above problem ([Listing 15-5](#)) is shown below:

=0		SPCFORCE RESULTANT						
SUBCASE/	LOAD							
DAREA ID	TYPE	T1	T2	T3	R1	R2	R3	
0 1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00	
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00	
	FZ	----	----	-1.000000E+02	0.000000E+00	0.000000E+00	----	
	MX	----	----	----	0.000000E+00	----	----	
	MY	----	----	----	----	5.000000E+02	----	
	MZ	----	----	----	----	----	0.000000E+00	
TOTALS		0.000000E+00	0.000000E+00	-1.000000E+02	0.000000E+00	5.000000E+02	0.000000E+00	
0 2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00	
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00	
	FZ	----	----	-3.307500E+02	0.000000E+00	0.000000E+00	----	
	MX	----	----	----	0.000000E+00	----	----	
	MY	----	----	----	----	8.268750E+02	----	
	MZ	----	----	----	----	----	0.000000E+00	
TOTALS		0.000000E+00	0.000000E+00	-3.307500E+02	0.000000E+00	8.268750E+02	0.000000E+00	

Figure 15-18. SPCFORCE RESULTANT

Since the SPCFORCE RESULTANT and OLOAD RESULTANT are summed about the same reference point, you would expect the total applied loads and total reaction loads to be equal and opposite of each other. As indicated by [Figure 15-16](#) and [Figure 15-18](#), this is indeed the case.

Similar to the applied loads, you can also request more detailed information regarding where exactly on the structure is reacting to these applied loads (grid point locations and directions). These are also known as boundary loads. This operation can be done with the SPCFORCE request in the Case Control Section. The format is as follows:

```
SPCFORCE = All
```

or

```
SET 10 = 15, 16, 18, 25
SPCFORCE = 10
```

VERTICAL TIP LOAD					SUBCASE 1			
FORCES OF SINGLE - POINT CONSTRAINT								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	-1.000000E+02	.0	5.000000E+02	.0	
GRAVITY LOADS					SUBCASE 2			
FORCES OF SINGLE - POINT CONSTRAINT								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	-3.307500E+02	.0	8.268750E+02	.0	

Figure 15-19. Single-Point Constraint Forces

In this case, since there is only one reaction point (fixed at grid point 1) in the model, the SPCFORCE output only contains one line of output per subcase (see [Figure 15-19](#)). Some guidelines to consider when examining the reaction loads include:

1. Are they equal to the applied loads? Is the structure in equilibrium?
2. Are the boundary loads coming out in the directions and magnitudes that you are expecting? As an example, if one of your reaction loads is to simulate a shear bolt and you are getting a tension reaction load at that location, the chances are that you have modeled the boundary condition incorrectly.
3. Is the load distribution over the structure what you expect? If not, you may want to go back and check your modeling of your reaction points to make sure that they are what you intended.
4. Grid points with reaction force equal to 0.0 on all six components are suppressed from the printed output.

1g Load

A 1g gravitation load applied separately in each of the three orthogonal directions is suggested for model checkout. This check is accomplished with three separate subcases. This is a basic check to determine grid points on the structure that are either loosely connected or have very small stiffnesses. A displacement output indicates that these points are moving excessively as compared to other parts of the model, especially if you are using a postprocessor. To use this feature, you must ensure that your structure contains masses—either by direct masses (e.g., CONM2) or indirect masses (e.g., from the density on the MATi entries), or nonstructural mass (e.g., a nonzero value on the NSM field on the PBAR entry).

Output of Maximum, Minimum Grid Point-based Responses

Documenting the validation of a mathematical simulation model can be a very tedious and time consuming process. A portion of this process usually requires the generation of reports to completely describe the results of the performed analysis. Although a substantial amount of the reporting requirements can be satisfied using graphical results plots, tabular data that summarizes the information depicted in the graphics are often required. Also, during the validation process itself, many tools are employed that assist users in evaluating the results produced by the simulation. Maximum/minimum surveys of output quantities are one such tool provided by the MAXMIN Case Control command.

Historically, NX Nastran users have been able to obtain simple surveys of grid point related results output such as displacements, applied loads, and forces of single point constraint. The output consists of a summary that identifies the grid point at which the maximum and minimum values occurred in each of the six output directions. This information is provided for each subcase.

The max/min information appears as typical grid point related output. You can request a number of the highest and lowest values for a particular component. All components of the grid related quantities are output together with the one component that is being surveyed. For example, if 10 maximum and 10 minimum values of T3 displacements are requested, the output will also contain the T1, T2, R1, R2, and R3 displacement values associated with the grid points identified in the T3 component survey. The output heading clearly identifies the component being surveyed. Output can be surveyed for SUBCASEs and SUBCOMs. You can also specify sets of points to be surveyed and coordinate system to be used for the survey output. Only SORT1 output is available.

If output coordinate systems are specified for any of the grid points being surveyed, the user has the option of specifying one of several coordinate system views of the max/min output. The data can be displayed in the basic system, a local system, or even the global system. Any grid point that is output in the max/min survey will have its output presented in both the survey coordinate frame of reference and its original frame of reference if they are different. In this way, the user can see both the relative position of the grid point component in the max/min output as well as the original output quantities without having to research other portions of the displacement output produced by the analysis.

See also

- “MAXMIN” in the *NX Nastran Quick Reference Guide*

Example: Using the MAXMIN Command

The following sample output demonstrates some features available with the MAXMIN Case Control command.

Inspection of the Case Control Echo reveals that the MAXMIN command at line 7 requests print and punch of max/min survey results for all available output quantities generated (via keywords PRINT, PUNCH and ALL). It also specifies that survey results be limited to component directions T1 and T2 for grid points defined in case control set 100. Because the command has been placed above the subcase level, it applies to all subcases and subcoms.

The MAXMIN command at line 11 requests a survey of the forces of single point constraint output (keyword SPCF) for component direction T1 only (keyword T1) in the basic coordinate system frame of reference (keyword CID selects BASIC). It also requests that the highest and lowest 15 values be output considering ALL grid points. Because it has been placed within subcase 1000, it only pertains to that subcase.

The MAXMIN command at line 14 requests print and punch of max/min survey results for single point forces of constraint. A vector magnitude survey is requested (keyword VMAG) for the four highest and lowest values. The results for these types of surveys appear in the columns of output normally containing component directions T1 and R1. Note that the specification of component directions to be surveyed (keyword COMP) has no effect in this case. All grid points will be included in the survey.

```

C A S E      C O N T R O L      E C H O
COMMAND
COUNT
1  TITLE = EXAMPLE OUTPUT FROM THE NEW MAXMIN CASE CONTROL COMMAND
2  SUBTITLE = ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
3  LABEL = COMPONENT DOF SPECIFICATIONS ARE POSSIBLE
4  SPC=100
5  SET 100 = 2,51,59
6  DISP=100
7  MAXMIN (PUNCH,PRINT,ALL ,COMP=T1/T2) = 100 $
8  SPCF=ALL
9  OLOAD=ALL
10 SUBCASE 1000
11 MAXMIN (SPCF,CID=BASIC,COMP=T1,BOTH=15) = ALL
12 LOAD=1000
13 SUBCASE 2000
14 MAXMIN (PUNCH,PRINT,SPCF,VMAG=4,COMP=T1/T3) = ALL
15 LOAD=1000
16 SUBCOM 3000
17 SUBSEQ = 0.5,0.5
18 BEGIN BULK

```

Examination of the following actual MAXMIN survey output results reveals that the three heading lines contain important information that describe the options used to produce the results. The first line identifies the component direction that was searched, the type of output being searched, and the SUBCASE/SUBCOM identification number. The second line summarizes the options specified on the MAXMIN Case Control command used to perform the search. The third line contains column headings for the data values that follow in the output listing. Note that the field, TYPE, identifies whether it is a grid point (G) or scalar point (S) and the field, POINT ID, identifies the grid point or scalar ID number. In addition, the coordinate system reference frame for the output values is identified. If data values for a point were transformed to a reference frame different from the one originally defined in the output, then the data values for both frames of reference are output. This is shown in the output for the first grid point (ID = 2) in the sample output. Here it is seen that values for the forces of single point constraint are provided for the point in the basic and CID 100 systems. This occurs because the output for the point has been specified in CID=100, but the max/min survey was requested in the basic coordinate system. See the MAXMIN Case Control command for further details.


```

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 NX NASTRAN 8/21/04 PAGE 19
0  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T1 *** APPLIED LOAD MAX / MIN VALUE SUMMARY                RESULTS FOR SUBCASE 1000
MAININ OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1
POINT ID. TYPE  CID      *** T1 ***      T2      T3      R1      R2      R3
2  G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
2  G  100        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S1 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S5 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 NX NASTRAN 8/21/04 PAGE 20
0  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T2 *** APPLIED LOAD MAX / MIN VALUE SUMMARY                RESULTS FOR SUBCASE 1000
MAININ OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T2
POINT ID. TYPE  CID      *** T2 ***      T3      R1      R2      R3
2  G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
2  G  100        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S1 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S5 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 NX NASTRAN 8/21/04 PAGE 21
0  ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T1 *** APPLIED LOAD MAX / MIN VALUE SUMMARY                RESULTS FOR SUBCASE 2000
MAININ OPTIONS: SET=100, CID=BASIC, MAX=1, MIN=2, COMP=T1
POINT ID. TYPE  CID      *** T1 ***      T2      T3      R1      R2      R3
2  G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
2  G  100        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S1 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
S5 G  BASIC      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

```

1	EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND										AUGUST 22, 2004 NI NASTRAN 8/21/04 PAGE 22	23
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED											
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE											
0	*** T2 *** APPLIED LOAD MAX / MIN VALUE SUMMARY RESULTS FOR SUBCASE 2000											
MAININ OPTIONS: SET=100, CID=BASIC, MAI=1, MIN=2, COMP=T2												
POINT ID.	TYPE	CID	T1	T2	T3	R1	R2	R3				
2	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
2	G	100	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
51	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
59	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
1	EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND										AUGUST 22, 2004 NI NASTRAN 8/21/04 PAGE 22	23
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED											
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE											
0	*** T1 *** APPLIED LOAD MAX / MIN VALUE SUMMARY RESULTS FOR SUBCOM 3000											
MAININ OPTIONS: SET=100, CID=BASIC, MAI=1, MIN=2, COMP=T1												
POINT ID.	TYPE	CID	T1	T2	T3	R1	R2	R3				
2	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
2	G	100	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
51	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
59	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
1	EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND										AUGUST 22, 2004 NI NASTRAN 8/21/04 PAGE 22	24
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED											
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE											
0	*** T2 *** APPLIED LOAD MAX / MIN VALUE SUMMARY RESULTS FOR SUBCOM 3000											
MAININ OPTIONS: SET=100, CID=BASIC, MAI=1, MIN=2, COMP=T2												
POINT ID.	TYPE	CID	T1	T2	T3	R1	R2	R3				
2	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
2	G	100	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
51	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
59	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
1	EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND										AUGUST 22, 2004 NI NASTRAN 8/21/04 PAGE 22	25
0	ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED											
0	COMPONENT DOF SPECIFICATIONS ARE POSSIBLE											
0	*** T1 *** SPC FORCE MAX / MIN VALUE SUMMARY RESULTS FOR SUBCASE 1000											
MAININ OPTIONS: SET=ALL, CID=BASIC, MAI=15, MIN=15, COMP=T1												
POINT ID.	TYPE	CID	T1	T2	T3	R1	R2	R3				
3	G	BASIC	-1.118073E+03	0.000000E+00	0.000000E+00	5.236134E+01	2.360384E-16	1.280828E+02				
3	G	100	0.000000E+00	0.000000E+00	-1.118073E+03	1.280828E+02	-2.360384E-16	5.236134E+01				
101	G	BASIC	-1.000002E+03	-1.733791E-11	-1.208690E-09	0.000000E+00	0.000000E+00	8.325034E-02				
251	G	BASIC	-1.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
201	G	BASIC	-1.000000E+03	-1.731096E-11	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
151	G	BASIC	-9.999980E+02	0.000000E+00	-6.541988E-14	0.000000E+00	0.000000E+00	-2.454053E+02				
1	G	BASIC	-4.012542E+02	1.588410E-11	1.125718E-15	-5.499307E+00	-2.439500E-01	7.725024E+01				
51	G	BASIC	-3.474817E+02	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-4.013108E+01				
2	G	BASIC	-1.331908E+02	0.000000E+00	0.000000E+00	-4.686203E+01	2.439500E-01	2.724495E+02				
2	G	100	0.000000E+00	0.000000E+00	-1.331908E+02	2.724495E+02	-2.439500E-01	-4.686203E+01				
209	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
208	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
207	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
206	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
205	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
204	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
203	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				
55	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-2.368870E-02				
56	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-7.088747E-04				
57	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-4.525901E-04				
58	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-1.690151E-04				
59	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	1.056287E-04				
60	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	8.333347E+01				
6	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	1.338936E-03				
102	G	BASIC	0.000000E+00	0.000000E+00	1.209136E-09	0.000000E+00	0.000000E+00	2.414764E+02				
103	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-3.667573E+00				
104	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-1.456001E+00				
105	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-8.364819E+01				
5	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	1.390992E-01				
152	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	6.736528E+00				
153	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	1.944592E+00				
700	G	BASIC	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00				

```

78      S      0.000000E+00
      SS      0.000000E+00
1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 HI NASTRAN 8/21/04 PAGE 26
      ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T1 ***      S P C F O R C E      M A I / M I N      V A L U E      S U M M A R Y      R E S U L T S F O R S U B C A S E      3000
      N A S T R A N O P T I O N S : S E T - A L L , C I D - B A S I C , V M A G = 4 , C M P = T 1
      P O I N T I D . T Y P E      C I D      *** T M A G ***      T 2      T 3      R 1      R 2      R 3
700  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
257  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
202  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
203  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
251  G      B A S I C      1.000000E+03      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
201  G      B A S I C      1.000000E+03      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
101  G      B A S I C      1.000000E+03      0.000000E+00      0.000000E+00      8.325034E-02      0.000000E+00      0.000000E+00
3  G      B A S I C      1.118073E+03      0.000000E+00      0.000000E+00      1.383724E-02      0.000000E+00      0.000000E+00
3  G      100      0.000000E+00      0.000000E+00      -1.118073E+03      1.280828E+02      -2.350384E-16      5.236134E+01
78  S      0.000000E+00
88  S      0.000000E+00

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 HI NASTRAN 8/21/04 PAGE 27
      ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** R1 ***      S P C F O R C E      M A I / M I N      V A L U E      S U M M A R Y      R E S U L T S F O R S U B C A S E      3000
      N A S T R A N O P T I O N S : S E T - A L L , C I D - B A S I C , V M A G = 4 , C M P = R 1
      P O I N T I D . T Y P E      C I D      T 1      T 2      T 3      *** F M A G ***      R 2      R 3
700  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
254  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
210  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
209  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
52  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      2.110957E-02      0.000000E+00      0.000000E+00
102  G      B A S I C      1.209136E-09      0.000000E+00      0.000000E+00      2.414754E-02      0.000000E+00      0.000000E+00
151  G      B A S I C      9.999980E-02      0.000000E+00      0.000000E+00      2.454053E-02      0.000000E+00      0.000000E+00
2  G      B A S I C      1.331908E+02      0.000000E+00      0.000000E+00      2.764504E-02      0.000000E+00      0.000000E+00
2  G      100      0.000000E+00      0.000000E+00      -1.331908E+02      2.724495E+02      -2.439500E-01      -4.686203E+01

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 HI NASTRAN 8/21/04 PAGE 28
      ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T1 ***      S P C F O R C E      M A I / M I N      V A L U E      S U M M A R Y      R E S U L T S F O R S U B C O M      3000
      N A S T R A N O P T I O N S : S E T - 1 0 0 , C I D - B A S I C , M A I = 1 , M I N = 2 , C M P = T 1
      P O I N T I D . T Y P E      C I D      *** T 1 ***      T 2      T 3      R 1      R 2      R 3
51  G      B A S I C      -3.474817E+02      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      -4.013108E+01
2  G      B A S I C      -1.331908E+02      0.000000E+00      0.000000E+00      -4.686203E-01      2.439500E-01      2.724495E+02
2  G      100      0.000000E+00      0.000000E+00      -1.331908E+02      2.724495E+02      -2.439500E-01      -4.686203E+01
59  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      1.056287E-04

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 HI NASTRAN 8/21/04 PAGE 29
      ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T2 ***      S P C F O R C E      M A I / M I N      V A L U E      S U M M A R Y      R E S U L T S F O R S U B C O M      3000
      N A S T R A N O P T I O N S : S E T - 1 0 0 , C I D - B A S I C , M A I = 1 , M I N = 2 , C M P = T 2
      P O I N T I D . T Y P E      C I D      T 1      *** T 2 ***      T 3      R 1      R 2      R 3
51  G      B A S I C      -3.474817E+02      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      -4.013108E+01
2  G      B A S I C      -1.331908E+02      0.000000E+00      0.000000E+00      -4.686203E-01      2.439500E-01      2.724495E+02
2  G      100      0.000000E+00      0.000000E+00      -1.331908E+02      2.724495E+02      -2.439500E-01      -4.686203E+01
59  G      B A S I C      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      1.056287E-04

1  EXAMPLE OUTPUT FROM THE NEW MAININ CASE CONTROL COMMAND          AUGUST 22, 2004 HI NASTRAN 8/21/04 PAGE 30
      ONLY GRID POINT RELATED OUTPUT QUANTITIES CAN BE PROCESSED
0  COMPONENT DOF SPECIFICATIONS ARE POSSIBLE

0  *** T1 ***      D I S P L A C E M E N T      M A I / M I N      V A L U E      S U M M A R Y      R E S U L T S F O R S U B C A S E      1000
      N A S T R A N O P T I O N S : S E T - 1 0 0 , C I D - B A S I C , M A I = 1 , M I N = 2 , C M P = T 1
      P O I N T I D . T Y P E      C I D      *** T 1 ***      T 2      T 3      R 1      R 2      R 3
2  G      B A S I C      0.000000E+00      -3.568517E-05      -2.158988E-07      0.000000E+00      0.000000E+00      0.000000E+00
2  G      100      -2.158988E-07      3.568517E-05      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
51  G      B A S I C      0.000000E+00      1.852034E-04      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00
59  G      B A S I C      1.939060E-02      -2.595800E-02      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00

```

Performing an Unconstrained Equilibrium Check

If a structure is truly unconstrained, then movement at one point of this structure should cause the whole structure to move as a rigid body. The following procedure can be used to perform this check.

1. Remove all the constraints.
2. Apply a unit enforced displacement in the x-direction at one selected grid point while constraining the other five components at this selected grid point to zero. This grid point should be close to the center of gravity (C.G.) of the structure, although this is not a requirement. The x-displacements at all the other grid points should also be equal to unity if the structure is truly unconstrained. If

the displacement at a certain grid point is not equal to unity in the x-direction, it is very likely that it is overstrained. The likely causes for this can include incorrect modeling of rigid elements, offset beams, non-colinear CELASi, etc. See “Constraints” for details regarding enforced displacement.

- Repeat the same procedure for the other two orthogonal directions. You can perform similar checks for rotations; however, it is more difficult to interpret the results.

At the conclusion of this check, you should remember to put your original constraints back into your model.

Automatic Identification of Unintentional Grounding

In addition to the unconstrained equilibrium check mentioned above, you may also request a more robust and more automatic grounding check of the stiffness matrix. This grounding check is also referred to as rigid body check because the stiffness matrix is multiplied by the rigid body transformation matrix. The grounding check identifies unintentional constraints and ill-conditioning in the stiffness matrix.

The grounding check is requested with the GROUNDCHECK Case Control command.

$$\text{GROUNDCHECK} \left[\left(\begin{bmatrix} \text{PRINT} \\ \text{NOPRINT} \end{bmatrix}, \text{PUNCH, SET} = \left(\begin{bmatrix} \text{G, N, N + AUTOSPC, F, A} \\ \text{ALL} \end{bmatrix} \right) \right) \right] = \begin{bmatrix} \text{YES} \\ \text{NO} \end{bmatrix}$$

$$\left[\text{GRID} = \text{gid, THRESH} = e, \text{DATA REC} = \begin{bmatrix} \text{YES} \\ \text{NO} \end{bmatrix}, (\text{RTHRESH} = r) \right]$$

For example, the following command

```
GROUNDCHECK=YES
```

will request a grounding check of the g-set. The following command

```
GROUNDCHECK (SET=ALL) =YES
```

will request a grounding check of all DOF sets.

The check may be performed at any or all stages of the stiffness reduction based on the specification of the SET keyword.

SET Keyword	DOF Set	Description
G	g-set	before single point, multipoint constraints, and rigid elements are applied
N	n-set	after multipoint constraints and rigid elements are applied
N+AUTO	n-set with AUTOSPC	same as the n-set with the rows/columns in the stiffness matrix corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out
F	f-set	after single point, multipoint constraints, and rigid elements are applied
A	a-set	after static condensation

The THRESH keyword specifies the maximum strain energy that passes the grounding check. DATA REC=YES requests data recovery of grounding forces. The RTHRESH=r keyword prints the grounding forces larger than r percent of the largest grounding force if DATA REC is set to YES.

User Information Message 7570 is issued by GROUNDHECK for each DOF set requested. The strain energy is computed in each direction of the rigid body motion and by default, if the strain energy exceeds the tolerance, then "FAIL" is printed out for that directory. The tolerance is set by dividing the largest stiffness term by 1.E10. The THRESH keyword can be used to further control the amount of output (see Appendix A). Possible reasons for failure are also printed after the strain energies. Here is an example for the g-set that indicates the stiffness matrix passes the grounding check in all six rigid body directions:

RESULTS OF RIGID BODY CHECKS OF MATRIX KGG (G-SET) FOLLOW:		
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 2.725275E-04		
DIRECTION	STRAIN ENERGY	PASS/FAIL
-----	-----	-----
1	1.484295E-09	PASS
2	2.182787E-10	PASS
3	1.637090E-11	PASS
4	1.619810E-10	PASS
5	2.727802E-10	PASS
6	1.054841E-07	PASS
SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:		
1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;		
2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;		
3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;		
4. IMPROPERLY DEFINED DMIG MATRICES;		

Here is an example for the f-set that indicates the stiffness matrix fails the grounding check in all six rigid body directions:

*** USER INFORMATION MESSAGE 7570 (GPWG1D)		
RESULTS OF RIGID BODY CHECKS OF MATRIX KFF (F-SET) FOLLOW:		
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 1.464858E-04		
DIRECTION	STRAIN ENERGY	PASS/FAIL
-----	-----	-----
1	2.564102E+05	FAIL
2	7.326008E+05	FAIL
3	2.237437E+03	FAIL
4	6.057062E+02	FAIL
5	4.015165E+03	FAIL
6	8.747863E+04	FAIL
SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:		
1. CONSTRAINTS WHICH PREVENT RIGID-BODY MOTION.		

If the DATAREC keyword is specified and the rigid body check fails then data recovery will be performed to compute and print the grounding forces to make it easy to locate the source of the failure. By default, only those grounding forces larger than 10 percent of the largest grounding force will be printed (see RTHRESH keyword). Here is an example of grounding forces created by moving the model in rigid body direction 1 associated with the f-set failure shown above:

DIRECTION		1	GROUND CHECK FORCES (F - S E T)						
POINT	ID.	TYPE	T1	T2	T3	R1	R2	R3	
	3	G	2.564102E+05	-2.884615E+05	.0	.0	.0	.0	
	4	G	2.564102E+05	2.884615E+05	.0	.0	.0	.0	

Thermal Equilibrium Check

The following procedure can also be used to check for unconstrained thermal expansion if the structure is going to be subjected to thermal loads.

1. Remove the actual boundary conditions and apply a set of statically determinate constraints. Typically this procedure is done by constraining all six DOFs at a single grid point. If a single grid point is used, make sure this single grid point contains six degrees of freedom. A single grid point

of a model consisting of all solid elements, for example, cannot satisfy this requirement since each grid point of a solid element contains only three degrees of freedom.

2. Change all the thermal coefficients of expansion to a single value.
3. Apply a uniform ΔT to the structure.

If the model is “clean,” then the structure should be strain free; in other words, there should be no reaction loads, element forces, or stresses. If this is not the case, then you may want to investigate around the vicinity where the element forces or stresses are nonzero. Incorrect modeling of rigid elements or offsets is a common cause of these types of errors. Once you are satisfied with your model, remember to change the boundary condition, thermal coefficients of expansion, and ΔT back to their original values.

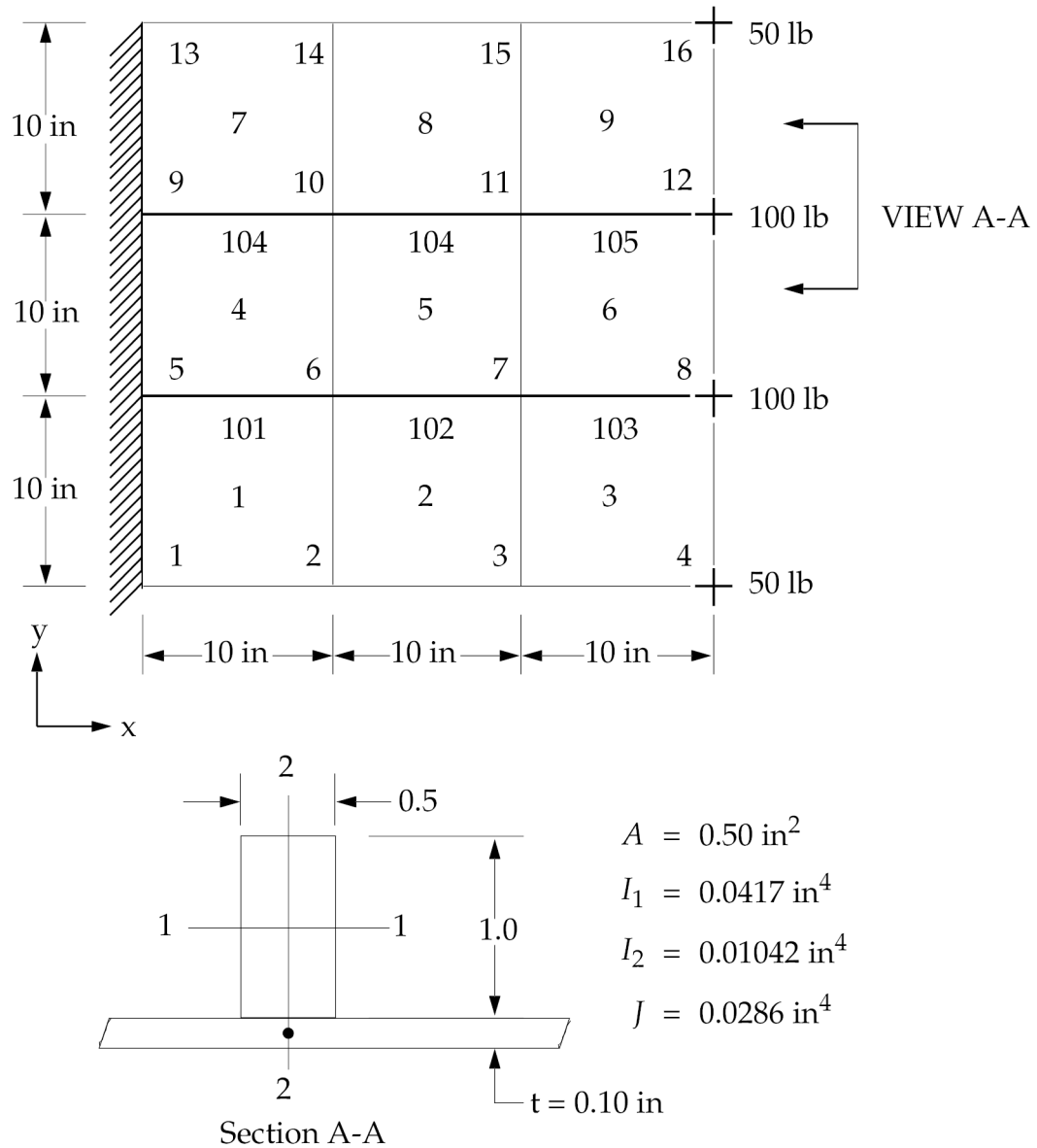
Grid Point Force Balance

Frequently, there are several elements connected to a single grid point. When this happens, you may want to know how the loads are distributed among these elements at the common grid point. In other words, you want to understand the load path. Is the load path correct? These questions can be answered by using the GPFORCE (grid point force) option in NX Nastran.

Grid point force balance is computed only from linear stiffness elements, the sum of applied loads and thermal loads, and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), rigid elements and MPCs, general elements, DMIG entries, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. **Table 15-2** summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system.

Table 15-2. Contributions Used for the Grid Point Force Balance	
Contributions Included	Contributions Ignored
Applied Loads	Boundary Loads from Upstream Superelements
SPC Forces	GENEL Forces
Element Elastic Forces	DMIG and DMI Forces
Thermal Loads	

The model in **Figure 15-20** illustrates this feature. This model consists of nine CQUAD4 and six CBAR elements and is subjected to vertical loads applied at the free end. The input file is shown in **Listing 15-6**.

EXAMPLE**Figure 15-20. Clamped Plate with Stiffeners**

```

$  FILENAME - q4bargpf.dat
$
SOL      101
TIME     5
CEND
TITLE = ILLUSTRATE USE OF GPFORCE
SUBTITLE =  INCORRECT PROPERTIES
DISP = ALL
GPFORCE = ALL
STRESS(CORNER) = ALL
SPC = 1
LOAD = 1
PARAM    AUTOSPC YES
PARAM    POST    0
$
$  THIS SECTION CONTAINS BULK DATA
$
GRID      1          0.0      0.0      0.0
GRID      2          10.      0.0      0.0
GRID      3          20.      0.0      0.0
GRID      4          30.      0.0      0.0
GRID      5          0.0      10.      0.0
GRID      6          10.      10.      0.0
GRID      7          20.      10.      0.0
GRID      8          30.      10.      0.0
GRID      9          0.0      20.      0.0
GRID     10          10.      20.      0.0
GRID     11          20.      20.      0.0
GRID     12          30.      20.      0.0
GRID     13          0.0      30.      0.0
GRID     14          10.      30.      0.0
GRID     15          20.      30.      0.0
GRID     16          30.      30.      0.0
$
CBAR      101      100      5          6          0.0      0.0      1.
+C101          0.      0.      .55          0.      0.      .55
CBAR      102      100      6          7          0.0      0.0      1.
+C102          0.      0.      .55          0.      0.      .55
CBAR      103      100      7          8          0.0      0.0
+C103          0.      0.      .55          0.      0.      .55
CBAR      104      200      9         10          0.0      0.0
+C104          0.      0.      .55          0.      0.      .55
CBAR      105      200     10         11          0.0      0.0
+C105          0.      0.      .55          0.      0.      .55
CBAR      106      200     11         12          0.0      0.0
+C106          0.      0.      .55          0.      0.      .55
$

```

Listing 15-6. Input File for the Demonstration of Grid Point Force Balance (Continued)


```

CQUAD4 1      1      1      2      6      5
CQUAD4 2      1      2      3      7      6
CQUAD4 3      1      3      4      8      7
CQUAD4 4      1      5      6     10      9
CQUAD4 5      1      6      7     11     10
CQUAD4 6      1      7      8     12     11
CQUAD4 7      1      9     10     14     13
CQUAD4 8      1     10     11     15     14
CQUAD4 9      1     11     12     16     15
$
$ THIS SECTION CONTAINS THE LOADS and CONSTRAINTS
$
$
FORCE 1      4      0      50.      0.      0.      -1.
FORCE 1      8      0     100.      0.      0.      -1.
FORCE 1     12      0     100.      0.      0.      -1.
FORCE 1     16      0      50.      0.      0.      -1.
$
SPC1 1      123456 1      5      9      13
$
$ THIS SECTION CONTAINS THE PROPERTY AND MATERIAL BULK DATA ENTRIES
$
PSHELL 1      1      .1      1
$
PBAR 100      1      .50      .0417      .01042      .0286
+P100 .5      .25     -.5      .25      -.5      -.25      .5      -.25
PBAR 200      1      .50      .0417      .01042      .0286
+P200 .5      .25     -.5      .25      -.5      -.25      .5      -.25
$
$ CORRECT PBAR ENTRY WITH A AND I1
$
$PBAR 200      1      .50      .0417      .01042      .0286
$
$
MAT1 1      1.+7      .3
ENDDATA

```

Listing 15-6. Input File for the Demonstration of Grid Point Force Balance

If everything is done correctly, expect the stress contour plot to look something like [Figure 15-21\(a\)](#). However, say that you made a modeling mistake somewhere, and the stress contour plot is coming out as shown in [Figure 15-21\(b\)](#). You know that something is wrong because the stress contour plot is not symmetrical. Since the structure, boundary conditions, and applied loads are all symmetrical, you also expect the stress contour plot to be symmetrical. By inspecting the grid point force balance output ([Figure 15-22](#)), you notice that the stiffeners (CBAR 104, 105, and 106) are not picking up any axial, vertical, or out-of-plane bending loads at grid points 9, 10, 11, and 12.

A review of the input file explains the reason for this result. The area (A) and the out-of-plane bending moment of inertia (I1) were "accidentally" left out for CBAR elements 104, 105, and 106 (PBAR 200). Hence, all the axial and out-of-plane bending loads at grid points 9, 10, 11, and 12 were picked up completely by the surrounding CQUAD4 elements. By correcting the PBAR element properties, note that CBAR elements 104, 105, and 106 now pick up the correct loads as shown in [Figure 15-23](#).

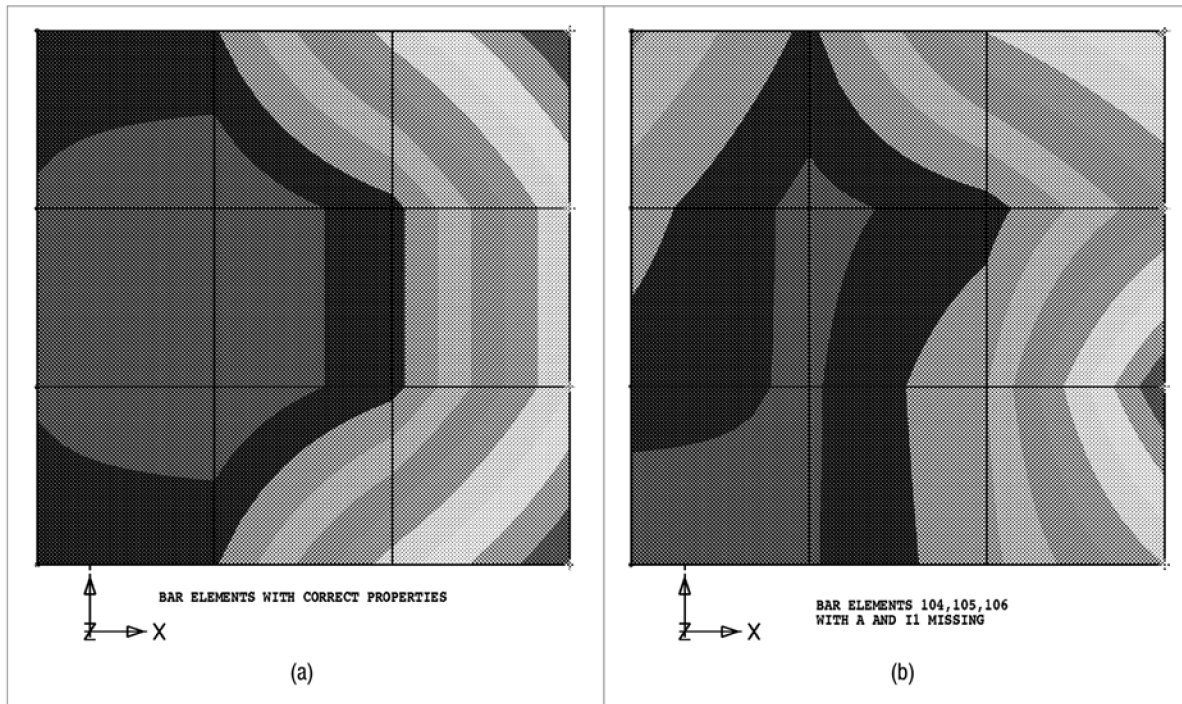


Figure 15-21. Stress Contour Plots of a Cantilever Plate

POINT-ID	ELEMENT-ID	SOURCE	GRID POINT FORCE			BALANCE			R1	R2	R3
			T1	T2	T3	R1	R2	R3			
1		F-OF-SPC	2.974426E+03	9.828222E+02	1.589665E+01	1.144652E+02	1.100169E+01	.0			
1	1	QUAD4	-2.974426E+03	-9.828222E+02	-1.589665E+01	-1.144652E+02	-1.100169E+01	.0			
1		*TOTALS*	4.547474E-13	.0	-8.881784E-15	-4.263256E-14	-2.486900E-14	.0			
9		F-OF-SPC	2.149695E+03	-7.652444E+02	4.273880E+01	5.070187E+02	-2.392951E+02	-1.157442E+02			
9	104	BAR	.0	1.266284E+01	.0	-7.117958E+02	.0	1.157442E+02			
9	4	QUAD4	-1.788234E+03	6.868671E+02	-5.524031E+01	1.670983E+02	1.526720E+02	.0			
9	7	QUAD4	-3.614603E+02	6.571449E+01	1.250151E+01	3.767887E+01	8.662309E+01	.0			
9		*TOTALS*	1.136868E-13	1.705303E-13	-1.136868E-13	-1.350031E-13	4.263256E-13	.0			
10	104	BAR	.0	-1.266284E+01	.0	7.117958E+02	.0	-1.088424E+01			
10	105	BAR	.0	2.289362E-01	.0	-9.452994E+02	.0	-1.088424E+01			
10	4	QUAD4	1.194813E+03	-1.018952E+02	6.047494E+01	-1.471362E+02	4.924568E+01	.0			
10	5	QUAD4	-1.501499E+03	3.827450E+02	-6.910983E+01	3.978395E+02	-4.706585E+01	.0			
10	7	QUAD4	7.029081E+02	-2.641614E+02	-1.464156E+01	-7.215542E+01	-5.501759E+01	.0			
10	8	QUAD4	-3.962218E+02	-4.254534E+00	2.327645E+01	5.495581E+01	5.283775E+01	.0			
10		*TOTALS*	8.526513E-13	-8.970602E-14	4.476419E-13	-1.627143E-12	-1.946887E-12	-5.329071E-15			
11	105	BAR	.0	-2.289362E-01	.0	9.452994E+02	.0	1.317360E+01			
11	106	BAR	.0	-1.317360E+00	.0	-7.749208E+02	.0	-1.317360E+01			
11	5	QUAD4	5.103351E+02	-5.226174E+02	9.304677E+01	-4.448763E+02	-8.715293E+01	.0			
11	6	QUAD4	-8.479946E+02	6.063411E+02	-8.722735E+01	4.217841E+02	3.954052E+01	.0			
11	8	QUAD4	4.692538E+02	-8.115437E+01	-3.112581E+01	-1.103274E+02	2.823395E-01	.0			
11	9	QUAD4	-1.315942E+02	-1.023029E+00	2.530639E+01	-3.695915E+01	4.733007E+01	.0			
11		*TOTALS*	-3.439027E-12	2.285061E-12	-2.501110E-12	5.400125E-12	-7.815970E-13	6.572520E-14			
12		APP-LOAD	.0	.0	-1.000000E+02	.0	.0	.0			
12	106	BAR	.0	1.317360E+00	.0	7.749208E+02	.0	-4.178994E-14			
12	6	QUAD4	-1.224487E+02	-1.046283E+01	1.404051E+02	-5.812056E+02	3.421728E+00	.0			
12	9	QUAD4	1.224487E+02	9.145473E+00	-4.040514E+01	-1.937151E+02	-3.421728E+00	.0			
12		*TOTALS*	7.389644E-13	-6.270540E-13	-1.371347E-12	1.023182E-12	-4.396483E-12	-4.178994E-14			
16		APP-LOAD	.0	.0	-5.000000E+01	.0	.0	.0			
16	9	QUAD4	-4.145872E-14	-8.016339E-13	5.000000E+01	-3.638509E-12	2.948251E-12	.0			
16		*TOTALS*	-4.145872E-14	-8.016339E-13	1.691092E-12	-3.638509E-12	2.948251E-12	.0			

Figure 15-22. Grid Point Force Balance Output for a Model with Incorrect CBAR Properties

GRID POINT FORCE BALANCE								
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	1.695155E+03	6.360602E+02	1.951741E+01	8.563783E+01	-2.406043E+01	.0
1	1	QUAD4	-1.695155E+03	-6.360602E+02	-1.951741E+01	-8.563783E+01	2.406043E+01	.0
1		*TOTALS*	.0	.0	3.552714E-15	1.421085E-14	-3.552714E-15	.0
9		F-OF-SPC	-1.695155E+03	-4.214514E+02	1.304826E+02	1.775948E+02	-4.475939E+03	-4.778485E+01
9	104	BAR	4.887901E+03	5.637726E+00	-1.451854E+02	-2.707557E+02	4.427706E+03	4.778485E+01
9	4	QUAD4	-1.885757E+03	3.980623E+02	-1.594168E+00	1.076575E+01	2.244092E+01	.0
9	7	QUAD4	-1.306990E+03	1.775138E+01	1.629696E+01	8.239518E+01	2.579244E+01	.0
9		*TOTALS*	-6.821210E-13	7.815970E-14	-1.101341E-13	5.684342E-14	-2.984279E-13	.0
10	104	BAR	-4.887901E+03	-5.637726E+00	1.451854E+02	2.707557E+02	-2.975852E+03	8.592416E+00
10	105	BAR	2.834930E+03	-2.358537E-02	-1.454878E+02	-2.992984E+02	2.890022E+03	-8.592416E+00
10	4	QUAD4	1.885757E+03	2.042302E+02	1.594168E+00	1.819319E+01	-6.499233E+00	.0
10	5	QUAD4	-1.123152E+03	-1.147349E+02	-1.597489E+00	3.257575E+01	2.503494E+01	.0
10	7	QUAD4	1.960801E+03	-4.728610E+02	-1.916611E+01	-9.600525E+01	-2.932959E+00	.0
10	8	QUAD4	-6.704347E+02	3.890270E+02	1.947184E+01	7.377895E+01	7.022765E+01	.0
10		*TOTALS*	-1.023182E-12	3.410605E-13	-3.623768E-13	-2.842171E-14	-4.263256E-13	1.598721E-14
11	105	BAR	-2.834930E+03	2.358537E-02	1.454878E+02	2.992984E+02	-1.435144E+03	8.356563E+00
11	106	BAR	8.836555E-02	-8.356562E-01	-1.431583E+02	-1.784329E+02	1.377845E+03	-8.356563E+00
11	5	QUAD4	1.123152E+03	-2.516147E+01	1.597489E+00	4.088165E+01	-9.060055E+00	.0
11	6	QUAD4	-3.665653E+02	1.245955E+02	-1.571911E+00	5.274417E+01	2.618646E+01	.0
11	8	QUAD4	1.240412E+03	-2.789030E+02	-2.937232E+01	-1.661472E+02	1.650724E+01	.0
11	9	QUAD4	-4.572457E+01	1.802911E+02	2.701724E+01	-4.834404E+01	2.366565E+01	.0
11		*TOTALS*	-6.849632E-12	-3.126388E-13	-1.712408E-12	2.771117E-13	3.197442E-12	5.151435E-14
12		APP-LOAD	.0	.0	-1.000000E+02	.0	.0	.0
12	106	BAR	-8.836555E+02	8.356562E-01	1.431583E+02	1.784329E+02	5.373812E+01	-3.213094E-14
12	6	QUAD4	3.665653E+02	4.705300E+02	1.571911E+00	5.770844E+01	-1.046736E+01	.0
12	9	QUAD4	5.170903E+02	-4.713657E+02	-4.473020E+01	-2.361413E+02	-4.327076E+01	.0
12		*TOTALS*	3.069545E-12	-8.526513E-13	-4.902745E-13	-8.810730E-13	1.975309E-12	-3.213094E-14
16		APP-LOAD	.0	.0	-5.000000E+01	.0	.0	.0
16	9	QUAD4	2.966625E-13	3.832166E-14	5.000000E+01	-1.397169E-12	1.610404E-12	.0
16		*TOTALS*	2.966625E-13	3.832166E-14	8.171241E-13	-1.397169E-12	1.610404E-12	.0

Figure 15-23. Grid Point Force Balance Output for a Model with Correct CBAR Properties

Both [Figure 15-22](#) and [Figure 15-23](#) are reduced versions of the full output. Only the grid point force balance output as related to grid points 1, 9, 10, 11, 12, and 16 are shown in these two figures for clarity. Notice that for grid points 1 and 9, in addition to the elastic element forces, the SPC forces are also printed. Similarly, for grid points 12 and 16, the applied loads are also printed. As you can see, the grid point force balance feature can provide you with a tremendous insight regarding the load path of your structure.

15.5 Stress Error Estimators

“[Reviewing Grid Point Stresses](#)” is devoted to a discussion of the averaging procedures utilized to provide meaningful stresses at the grid points of finite element models in NX Nastran. However, when you look at a stress distribution, frequently you may not be sure of whether further model refinement is necessary. This section focuses upon the description of error estimators that help you identify regions of your model that may require refinement.

The essence of the averaging procedures used to obtain grid point stress data are to:

1. Convert these local element stress components into a common coordinate system defined by you.
2. Average several values of each stress component to obtain a unique value of the stress component that is to be associated with the grid point in question.
3. Compute the stress invariants at the grid points from the stress components at the grid points.

In the general case, the stress components are σ_x , σ_y , σ_z , τ_{xy} , τ_{xz} , and τ_{yz} .

For discussion purposes, the averaging process used to compute the stress components at the grid points can be represented in the form

$$\sigma_g = \sum_{i=1}^{N_e} (W_i \sigma_{ei})$$

Equation 15-3.

where:

- σ_g = the weighted mean value of the stress component computed at the grid point.
- σ_{ei} = the value of the stress component in the i -th element $i = 1, 2, \dots, N_e$ connected to the grid point. σ_{ei} is in the same coordinate system as σ_g .
- W_i = the weighting factor assigned to the i -th element. The sum of the values of must equal 1. This requirement assures that all computed statistics are unbiased. The attribute of being unbiased implies that the variance is equal to the mean square error. Equal weighting, i.e., $W_i = 1/N_e$, is assumed in NX Nastran.

An estimate of the error in a particular component of stress at a grid point can then be computed by assuming that the values of the corresponding stress components computed by NX Nastran for the elements in the neighborhood of the grid point are data points with uncorrelated random errors. It then follows that an estimate of the probable error in the stress component δg at the grid point is

$$\delta_g = \sqrt{\sum_{i=1}^{N_e} (W_i \delta_{ei})^2} = \frac{1}{\sqrt{N_e}} \sqrt{\sum_{i=1}^{N_e} (\delta_{ei})^2}$$

Equation 15-4.

where $\delta_{ei} = \sigma_{ei} - \sigma_g$. Thus, the probable error δ_g is the root mean square error in δ_{ei} divided by $\sqrt{N_e}$.

It should be noted that the root mean square error is a reasonable measure of precision in many practical cases, but it is easy to provide examples in which this value is a poor measure of the concentration of the distribution about the mean. This subject is discussed further in the section "Discussion of Error Measures." Eq. 15-4 is assumed to provide an approximate error estimator for the grid point stress data output by NX Nastran.

This stress error assessment is often referred to as stress discontinuity, and it is evaluated in two different ways in NX Nastran: as grid point and element stress discontinuities.

Grid Point Stress Discontinuities

To illustrate the computations described in the previous section, consider the following portion of a finite element model consisting of CQUAD4 elements. For clarity, only a portion of the model is plotted. These elements lie on a surface (defined as surface 91 in this case).

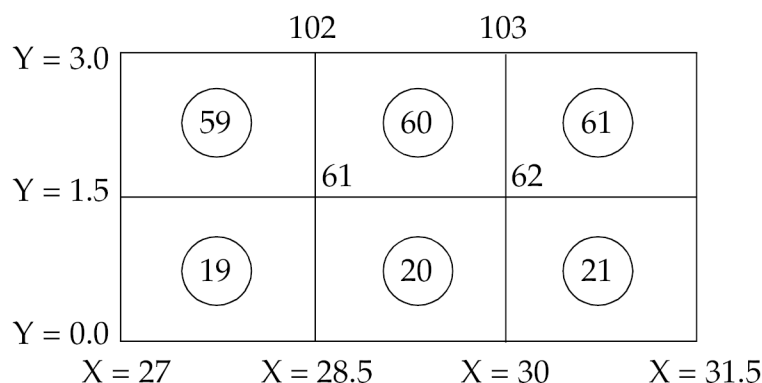


Figure 15-24. Partial Model for the Stress Discontinuity Calculations

The following data is extracted from the NX Nastran output. [Table 15-3](#) and [Table 15-4](#) contain partial listings of the element stresses and grid point stresses for the above model. The grid point stresses are obtained following the procedures described in [“Reviewing Grid Point Stresses”](#).

Table 15-3. Element Stresses for CQUAD4			
Element ID	Stresses in the Element Coordinate System		
	NORMAL-X	NORMAL-Y	SHEAR-XY
19	-4.006485E+03	-8.949428E-09	-1.045506E+02
20	-3.820136E+03	2.255547E-09	-1.045506E+02
.			
59	-1.335495E+03	8.658390E-09	-2.287827E+02
60	-1.273379E+03	-1.818989E-09	-2.287827E+02

Table 15-4. Grid Point Stresses for Surface 91

Grid ID	Stresses in the Element System		
	NORMAL-X	NORMAL-Y	SHEAR-XY
20	-5.218E+03	-6.730E-09	-4.243E+01
21	-4.969E+03	1.692E-09	-4.243E+01
.			
.			
.			
60	-2.733E+03	-3.274E-10	-1.667E+02
61	-2.609E+03	-3.638E-11	-1.667E+02
62	-2.485E+03	1.273E-10	-1.667E+02
63	-2.360E+03	-5.457E-11	-1.667E+02
.			
.			
.			
102	-6.104E-05	6.803E-09	-2.909E+02
103	-9.155E-05	-1.437E-09	-2.909E+02

The probable error for stresses at both grid points 20 and 61 can be computed using [Eq. 15-4](#). The probable error δ_g is expected to provide some measure of the error in the magnitude of the grid point stress values. You must first compute the values $\delta_{ei} = \sigma_{ei} - \sigma_g$ for each stress component and then compute the probable error, δ_g . This calculation is done for both grid points 20 and 61. The symbols “GID” and “EID”, which represent grid point ID and element ID, respectively, are used extensively throughout this section. Consider an example calculation using the NORMAL-X stress component.

NORMAL-X Stress

GID = 20			
EID	σ_{ei}	σ_g	δ_{ei}
19	-4.006485E+03	-5.218E+03	1.212E+03
20	-3.820136E+03	-5.218E+03	1.398E+03

Using [Eq. 15-4](#),

$$\delta_g = \frac{1}{\sqrt{2}} \sqrt{\frac{(1212)^2 + (1398)^2}{2}} = 924.9$$

GID = 61			
EID	σ_{ei}	σ_g	δ_{ei}
19	-4.006485E+03	-2.609E+03	-1.397E+03
20	-3.820136E+03	-2.609E+03	-1.211E+03
59	-1.335495E+03	-2.609E+03	1.274E+03
60	-1.273379E+03	-2.609E+03	1.336E+03

Again, using Eq. 15-4,

$$\delta_g = \frac{1}{\sqrt{4}} \sqrt{\frac{(-1397)^2 + (-1211)^2 + (1274)^2 + (1336)^2}{4}} = 653.1$$

The other stress components (σ_y and τ_{xy}) can then be obtained in a similar manner.

This same finite element model is then remodeled using different element types—CQUAD8, CHEXA(8) as well as CQUAD4 elements with the corner output option. The values of δ_g for each of the three stress components are then calculated at two locations: $x = 28.5$ and $y = 0.0$, as well as $x = 28.5$ and $y = 1.5$. Note that these two locations correspond to GID 20 and GID 61, respectively, for the above CQUAD4 model. This process is then repeated for each of the above element types. The results (δ_g and ERROR ESTIMATE) from NX Nastran are summarized in Table 15-5. Note that these values may be slightly different than the hand-calculated values since the computer provides more precision than the hand calculation.

The probable error δ_g provides an estimated error for each of the three stress components at each of the two grid point locations considered. It is probably more useful and more desirable to combine these three estimated errors into a single representative error measure at each grid point. An approximate root mean square value of the three estimated errors for each of the three stress components offers such a representative error measure. That is,

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{gi})^2}{N_c}}$$

Equation 15-5.

where N_c , which is the number of stress components, is equal to three for plate elements and six for solid elements.

Table 15-5. The Values of δ_g and ERROR ESTIMATE for Different Element Types(X = 28.5, Y = 0.0)

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL-Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	9.247E+02	4.628E-09	4.392E+01	5.345E+02
CQUAD8	0.000E+00	1.204E-09	0.000E+00	9.840E-10
CHEXA (8)	8.839E+01	2.841E+01	0.000E+00	5.729E+01
CQUAD4 (CORNER)	9.653E+01	2.896E+01	3.548E+02	2.129E+02

Table 15-6. The Values of δ_g and ERROR ESTIMATE for Different Element Types (X = 28.5, Y = 1.5)

Element Type		X = 28.5	Y = 1.5	
	NORMAL-X (δ_g)	NORMAL-Y (δ_g)	SHEAR-XY (δ_g)	ERROR ESTIMATE
CQUAD4	6.531E+02	3.196E-09	3.106E+01	3.775E+02
CQUAD8	0.000E+00	3.634E-09	0.000E+00	2.098E-09
CHEXA (8)	3.125E+01	1.497E+01	3.125E+01	2.148E+01
CQUAD4 (CORNER)	7.298E+01	2.153E+02	2.532E+02	1.965E+02

The following remarks should be noted in regard to [Table 15-5](#) and [Table 15-6](#).

1. By default, the stress components for the CQUAD4 elements are only output at the element centroids, while the stress components for the CQUAD8, and CHEXA(8) elements are output at the element centroids as well as the element vertices.
2. If vertex stresses are also desired for the CQUAD4 elements, then use the STRESS(CORNER) = x option in the Case Control Section. This output corresponds to the last rows of [Table 15-5](#) and [Table 15-6](#).
3. The calculation of δ_g for the CQUAD8 and CHEXA(8) elements involves grid point and element vertex stresses that are defined at the same points.
4. The calculation of δ_g for the CQUAD4 elements involve grid point stresses and element stresses provided at a different geometric location if the default center option is requested. They are evaluated at the same location when using the corner option.
5. The ERROR ESTIMATE for the CHEXA(8) also includes the contribution of σ_z , σ_{xz} , and σ_{yz} . These three values are not listed in [Table 15-5](#) and [Table 15-6](#). The procedure is identical to that of the plate element with the exception that it includes six instead of three stress components.

[Table 15-5](#) and [Table 15-6](#) indicate that the smaller values of δ_g and the ERROR ESTIMATE are obtained from the models composed of elements for which vertex stresses are computed by NX Nastran. A detailed discussion of the error estimator used by NX Nastran is presented later in the section "Discussion of Error Measures."

In addition to values of δ_g for each stress component and the ERROR ESTIMATE of Eq. 15-5 NX Nastran output also contains values of δ_g for each of the stress invariants. These quantities are output for each grid point in the defined SURFACE (plate elements) or VOLUME (solid elements).

Element Stress Discontinuities

Other statistical error estimates may be generated by associating the estimates with the elements rather than with the grid points. For example, root mean square errors for each stress component of an element may be computed from the values of δ_{ei} that are computed for each of the N_g connected vertex grid points where $\delta_{ei} = (\sigma_e - \sigma_{gi})$. This computation is done for each stress component for all of the elements of interest.

Next is an example calculating the NORMAL-X stress components for elements 20 and 60 in Figure 15-24.

NORMAL-X Stress

EID=20			
GID	σ_e	σ_{gi}	δ_{ei}
20	-3.820136E+03	-5.218E+03	1.398E+03
21	-3.820136E+03	-4.969E+03	1.149E+03
62	-3.820136E+03	-2.485E+03	-1.335E+03
61	-3.820136E+03	-2.609E+03	-1.211E+03

$$\delta_e = \sqrt{\frac{\sum_{i=1}^{N_g} (\delta_{ei})^2}{N_g}}$$

$$\delta_g = \sqrt{\frac{(1398)^2 + (1149)^2 + (-1335)^2 + (-1211)^2}{4}} = 1277$$

Equation 15-6.

EID = 60			
GID	σ_e	σ_{gi}	δ_{ei}
61	-1.273379E+03	-2.609E+03	1.336E+03
62	-1.273379E+03	-2.485E+03	1.212E+03
103	-1.273379E+03	-9.155E-05	-1.273E+03
102	-1.273379E+03	-6.104E-05	-1.273E+03

$$\delta_e = \sqrt{\frac{(1336)^2 + (1212)^2 + (-1273)^2 + (-1273)^2}{4}} = 1274$$

The other two components (σ_y and τ_{xy}) can be calculated in a similar manner.

The above data provides an estimated error for each of the three stress components for each of the two elements considered. As discussed in the previous section, it is desirable to combine these three estimated errors into a single representative error estimate for each element. The root mean square value of the three error estimates for each of the three stress components offers such a representative error measure. In other words,

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{ei})^2}{N_c}}$$

Equation 15-7.

(The results (δ_e and ERROR ESTIMATE) from NX Nastran for this model, using CQUAD4, CQUAD8, and CQUAD4 with the corner option, are summarized in [Table 15-7](#) and [Table 15-8](#).

Table 15-7. The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types				
Element Type	Location at EID = 20			
	NORMAL-X (δ_e)	NORMAL-Y (δ_e)	SHEAR-XY (δ_e)	ERROR ESTIMATE
CQUAD4	1.277E+03	4.757E-09	6.212E+01	7.382E+02
CQUAD8	0.000E+00	4.014E-09	0.000E+00	2.318E-09
CQUAD4 (CORNER)	1.407E+02	2.990E+02	4.932E+02	3.428E+02

Table 15-8. The Values of δ_e and ERROR ESTIMATE for Different Plate Element Types				
Element Type	Location at EID = 60			
	NORMAL-X (δ_e)	NORMAL-Y (δ_e)	SHEAR-XY (δ_e)	ERROR ESTIMATE
CQUAD4	1.274E+03	4.520E-09	6.212E+01	7.365E+02
CQUAD8	5.038E-10	3.362E-09	0.000E+00	2.020E-09
CQUAD4 (CORNER)	1.003E+02	2.969E+02	4.915E+02	3.366E+02

The CHEXA(8) element is not included in the above tables; however, the calculations are similar with the exception that six rather than three components of stress need to be considered. In addition to values of $\bar{\sigma}_e$ for each stress component and the ERROR ESTIMATE of Eq. 15-7, NX Nastran output also contains values of for each of the stress invariants. For plate elements, all of these quantities are output at the neutral plane and at fiber distances Z1 and Z2 for each element in the defined SURFACE.

Discussion of Error Measures

An examination of Table 15-5 through Table 15-8 clearly shows that the error estimates for CQUAD4 elements without vertex stress data are substantially larger than the corresponding error estimates for those elements that provide vertex stress data as a default. In view of the fact that the finite model under discussion was constructed with a rather high mesh density to produce results that agreed very closely with theory, you may perhaps be surprised by the large values of the error estimates for those elements that only provide stress data at the element centroid.

These seemingly large values of error estimators for CQUAD4 elements are not uncommon occurrences in statistical error measures. For example, the mean and variance have direct analogies in engineering mechanics. The mean is analogous to the centroid of a body and is thus a measure of where the mass is centered. The variance is the second moment about the mean, and it tends to be small if the majority of the mass is concentrated about the centroid. As the mass is dispersed further from the centroid, the moment of inertia tends to increase. If the mass is concentrated at the centroid (as it might be in some idealized lumped mass models), the moment of inertia becomes zero.

Thus, the error estimates for CQUAD4 elements tend toward larger values because the data used to compute the estimators is dispersed relatively far from the mean. For those elements that provide stress data at vertices, the estimators tend toward smaller values because the stress data tends to be concentrated about the mean in well-designed finite element models.

Generally, you should design a finite element mesh for static analysis so that all important stress gradients are adequately represented. If the mesh is not of sufficient detail, the stress data whether available at element vertices or at element centroids results in the relatively large values of the error estimators discussed here.

You are cautioned that very inaccurate values of these error estimators may occur at the edges of defined SURFACES and on the faces of defined VOLUMES. In summary, the error estimators under discussion can, in some cases, be highly inaccurate. Nevertheless, these data are quite useful when interpreted properly.

User Interface

The element and grid point stress discontinuity output can only be obtained if the grid point stress output is requested via the STRFIELD Case Control Command. The STRFIELD Case Control command is also used for graphical postprocessing of grid point stress, element stress discontinuity, and grid point stress discontinuity. The STRFIELD command, however, does not provide printed output for the grid point stress. The GPSTRESS Case Control command is needed if printed output for the grid point stress is desired. Similar to the GPSTRESS command, the STRFIELD command also requires that you define all applicable SURFACES and VOLUMES in the OUTPUT(POST) portion of the Case Control Section. The element stresses must also be requested for those elements that lie on the SURFACES and/or VOLUMES. For a more detailed description of GPSTRESS, refer to “[Reviewing Grid Point Stresses](#).”

Two other Case Control commands are needed—GPSDCON for grid point stress discontinuity and ELSDCON for element stress discontinuity. Note that you can also request grid point stress discontinuity without requesting element stress discontinuity, and vice versa, by omitting the appropriate Case Control command (ELSDON or GPSDCON).

The following remarks should be noted relative to the use of the GPSDCON and/or ELSDCON commands.

1. The GPSDCON and ELSDCON commands are supported only in Solution Sequence101.
2. The GPSDCON and ELSDCON commands may be placed above the SUBCASE level or in individual SUBCASEs and/or SUBCOMs. The use of these commands above the SUBCASE level causes the stress discontinuity data to be output for all SUBCASEs in the NX Nastran input file. If you wish to restrict stress discontinuity output to specific SUBCASEs, GPSDCON and/or ELSDCON commands should appear only under these specific SUBCASEs.
3. You are cautioned that these commands can produce a substantial amount of data. Therefore, you should be judicious in their use.
4. Stress continuity data is not provided when both plate and solid elements are connected to a grid point that is involved in stress discontinuity calculations.

The following is a listing of the input file used to generate the results for the CQUAD4 element model used in [Table 15-5](#) through [Table 15-8](#). The example problem in [Listing 15-7](#) contains the full list of Case Control commands required to obtain Grid Point/Element Stress Discontinuity output.

```

$
$  FILENAME - q4sdcon.dat
$
ID STRESS DISCON
TIME 25
SOL 101
CEND
TITLE = CANTILEVER BEAM WITH PLATES
$
SPC=1
LOAD=1000
$
$
$  ELEMENT OUTPUT REQUIRED
$
SET 22 = 19,20,21,59,60,61
$
ELSTRESS = 22
$
$  FOLLOWING REFERENCES SURFACE 91
$
SET 21 = 91
GPSTRESS=21
STRFIELD=21
$
$  FOLLOWING ARE FOR STRESS DISCONTINUITIES
$
GPSDCON=21  $ FOR GRID POINT DISCONTINUITIES
ELSDCON=21  $ FOR ELEMENT DISCONTINUITIES
$
$  FOLLOWING REQUIRED FOR GRID POINT STRESS
$
OUTPUT (POST)
SET 95 = 19,20,21,59,60,61
$
SURFACE 91  SET 95  NORMAL Z  SYSTEM ELEMENT
$
BEGIN BULK
$
$  BRING IN THE REST OF THE BULK DATA FILE
$
INCLUDE 'q4sdcon.blk'
$
$
ENDDATA

```

Listing 15-7. Input File for the Error Estimators for CQUAD4

15.6 Postprocessor Checks

Once you have completed your NX Nastran job, you must interpret the results. For a large model, it is impractical to go through every page of your output to look at the results. Graphical postprocessors can help you more easily examine and interpret large amounts of output data. This section describes some of the basic output quantities that you may want to consider when using a postprocessor.

Examining Displacements

When you post process your model, you should create a displacement plot for each of your loading conditions. If there are abrupt changes in the displacements at a certain region, you may want

to zoom in to inspect that local region and see if they are justifiable. One possible cause can be improper modeling at that location or certain elements not being connected.

Examining Stress Contours

Stress contour plots are calculated using a stress averaging technique of neighboring elements. Once these average stresses are calculated, the postprocessor assigns different colors to these stresses based on the stress ranges. These stresses can then be plotted as stress contour plots with various colors designating different stress ranges. This feature is probably the most frequently used postprocessing option. When used properly, it can help you quickly identify high stress regions and provide you with insight regarding your model.

When you create stress contour plots of component stresses, you should ensure that the component that you are plotting is in a consistent direction for all the elements that are included in your plot. The default component direction may be a function of element type. Some postprocessing packages may also transform them into a common coordinate system. The following example problem in [Figure 15-25](#) illustrates the importance of consistent direction. The stresses in a CQUAD4 element are in terms of the element coordinate system.

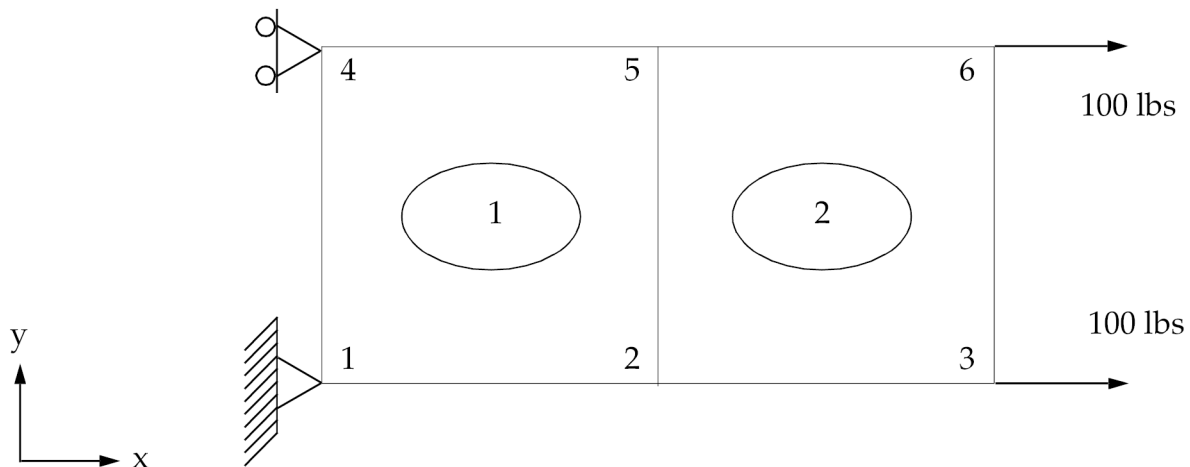


Figure 15-25. CQUAD4 Model

The complete input file used for this problem is shown in [Listing 15-8](#).

```

$
$ FILENAME - (q4consc.dat)
$
SOL      101
TIME     5
CEND
TITLE = CONSISTENT CQUAD4 CONNECTIVITIES
SPC =1
LOAD =1
STRESS = ALL
BEGIN BULK
$
$
PARAM, POST, 0
$
$ THIS SECTION CONTAINS GRID POINT LOCATIONS
$
GRID     1           0.0      0.0      0.0
GRID     2           10.      0.0      0.0
GRID     3           20.      0.0      0.0
GRID     4           0.0      10.      0.0
GRID     5           10.      10.      0.0
GRID     6           20.      10.      0.0
$
$ THIS SECTION CONTAINS ELEMENT CONNECTIVITIES
$
CQUAD4   1          10      1      2      5      4
CQUAD4   2          10      2      3      6      5
$
$CQUAD4   2          10      3      6      5      2
$
$ LOADS
$
FORCE    1          3           100.    1.
FORCE    1          6           100.    1.
$
$ CONSTRAINTS
$
SPC       1          1      12345    0.0
SPC       1          4      1345    0.0
$
$ SECTION AND MATERIAL PROPERTIES
$
PSHELL   10          10      .1      10
MAT1     10          1.+7      .32
$
ENDDATA

```

Listing 15-8. Input File for CQUAD4 Stress Contour Plots with Consistent Connectivity

In this example, the connectivities for both CQUAD4 elements are described in a consistent manner as shown in [Figure 15-26](#). Due to this consistency, the corresponding component stresses (e.g.,) point in the same direction for both elements. Note that for this model is equal to zero for both elements.

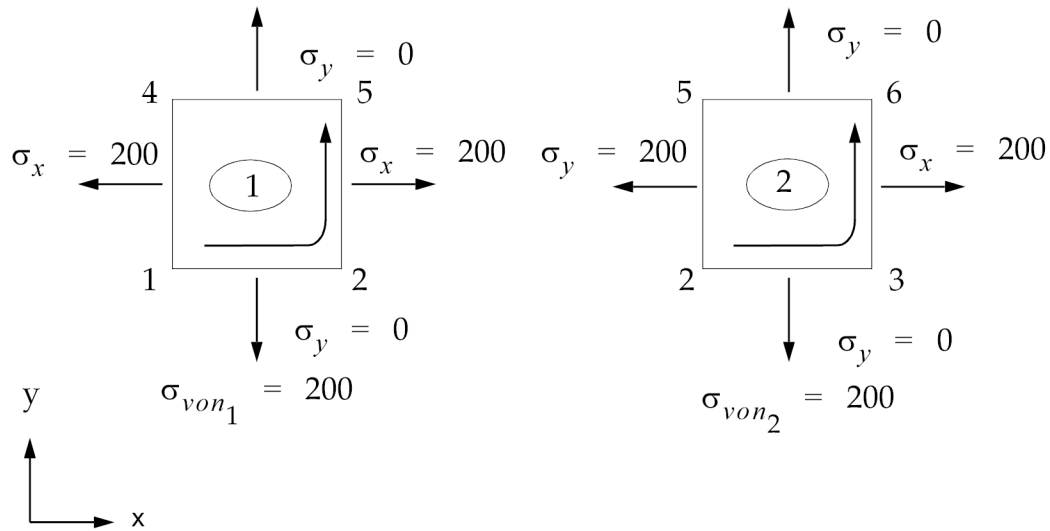


Figure 15-26. Consistent Connectivity

If you perform a stress contour on this model using σ_x , the contour plot for σ_x across the two elements can be calculated as follows:

$$\sigma_x = \frac{(\sigma_{x1} + \sigma_{x2})}{2} = \frac{(200 + 200)}{2} = 200.$$

The contour plot for σ_{von} across the two elements can be calculated as follows:

$$\sigma_{von} = \frac{(\sigma_{von1} + \sigma_{von2})}{2} = \frac{(200 + 200)}{2} = 200.$$

On the other hand, if the two elements are not connected in a consistent matter as shown in [Figure 15-27](#), then the corresponding component stresses (e.g. σ_x ,) do not necessarily point in the same directions for both elements. As shown in [Figure 15-27](#), the σ_x component stresses point in the directions of the x- and y-axis for elements 1 and 2, respectively.

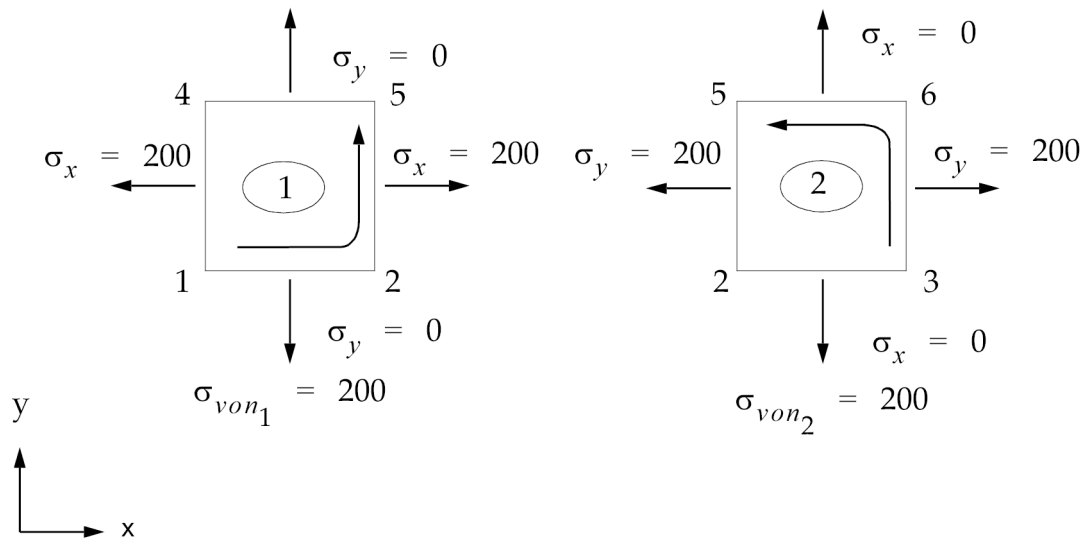


Figure 15-27. Inconsistent Connectivity

Once again, if you perform a stress contour on this model using , the value used in the contour plot for across the two elements in this case is as follows:

$$\sigma_x = \frac{(\sigma_{x1} + \sigma_{x2})}{2} = \frac{(200 + 0)}{2} = 100.$$

As you can see, the σ_x value used in the contour plot for this case is basically meaningless and could lead you to an erroneous conclusion.

The von Mises stresses used for the contour plot can be calculated as follows:

$$\sigma_{von} = \frac{(\sigma_{von1} + \sigma_{von2})}{2} = \frac{(200 + 200)}{2} = 200.$$

As you can see, σ_{von} remains the same since it is an invariant quantity and is independent of the order of connectivity. Note that the input file used for [Figure 15-27](#) is identical to [Listing 15-8](#) with the exception of the shaded CQUAD4 entry. In other words, replace the following CQUAD4 entry:

1	2	3	4	5	6	7	8	9	10
CQUAD4	2	10	2	3	6	5			

with the following:

1	2	3	4	5	6	7	8	9	10
CQUAD4	2	10	3	6	5	2			

and the two input files are then identical.

As you have probably observed by now, using contour plots of component stresses can lead to misleading results unless they are transformed into a common coordinate system. Therefore, in most instances, it is generally easier and more meaningful to look at the invariant stress quantities, such as

von Mises stresses, instead of the component stresses. However, there are situations where you may want to look at component stresses. An example is a cylindrical model whereby you are interested in looking at the hoop stress. In this case, you should be aware of the consequences if your model is not created in a consistent manner. Alternate invariant stress quantities are the principal stresses.

See also

- “[CQUAD4 and CTRIA3 Elements](#)” in the *NX Nastran Element Library*

Creating Grid Point Stress and Stress Discontinuity Plots

The grid point stress feature enables you to obtain stress output at the grid points instead of the elements. If the contour plot using grid point stress is substantially different from the contour plot using element stress, it is probably an indication that the model needs to be refined or that you may have inconsistent element orientations as described in the previous section.

Stress discontinuity is a measure of the probable error of the stresses across the grid points or elements. This type of plot assists you in the identification of regions of your model that may require refinement. This feature is discussed in detail in “[Stress Error Estimators](#).”

See also

- “[Reviewing Grid Point Stresses](#)”

Viewing Element Strain Energy Plots

The element strain energy plot is an excellent tool for identifying areas that are most effective in influencing your design. For a large model, this output request can generate a large amount of printed output. The plot option can be used to generate plots without generating printed output. This feature is discussed in detail in “[Strain Energy Output](#).”

Chapter 16: Restarts

- *Introduction to Restarts*
- *Types of Restarts*
- *Read-write Restarts*
- *Read-only Restarts*
- *Mini-database Restarts*
- *Determining the Version for a Restart*

16.1 Introduction to Restarts

A restart is a logical way of continuing from a previous run without having to start from the beginning. The savings can be in terms of both time and money. In the case of a static analysis, the most expensive and time consuming part of the run is the decomposition of the stiffness matrix, especially with large models. Now suppose after completing the original run, you want to obtain additional output (e.g., stresses, displacements, etc.) or add more load cases. You can always start from the beginning and redo the whole problem, or you can perform a restart at a fraction of the time and cost. In the case of additional output requests and additional load conditions, the decomposition of the stiffness matrix, which was performed in the previous run, is not redone if you use the restart feature.

However, a restart may not always be the best option compared to rerunning the job. Determining whether to save the database for restarting or rerun the analysis is based on:

- The amount of available disk space
- Computer speed
- The types of restarts
- The size of your model

For conventional static analysis runs (i.e, those not involving superelements), if a restart involves model changes, the savings is probably minimal. However, in the case of additional output requests or additional load cases, the savings can be substantial as indicated by [Table 16-1](#).

In addition to needing to rerun a job, you may also want to use NX Nastran's restart capabilities if you need to access data from an old file and the input file is no longer available. If you still have the database archive for that solve, the following sample run ([Listing 16-1](#)) shows how you can use restart to either print or punch a copy of the Bulk Data from the database that you can subsequently use to create a new input file.

```
$ FILE - bulklist.dat
$
restart
assign master='archive1.MASTER'
sol 101
time 10
cend
title = run to get listing of bulk data
echo = sort
$
$ replace echo=sort with echo=punch if a
$ punched output is desired instead of a listing
$
param,serst,semi
begin bulk
$
enddata
```

Listing 16-1. Input File for Printing/Punching a Listing of Bulk Data File from the Database

See also

- [“Restart Procedures”](#) in the *NX Nastran User's Guide*

16.2 Types of Restarts

There are two types of restarts available in NX Nastran:

- Automatic
- Manual

With automatic restarts, NX Nastran automatically determines which operations need to be performed during the current run based on the changes that are made in your input file. Automatic restarts are easy to use, and they can potentially save you a substantial amount of time—both CPU and elapsed time—by not having to resolve a problem from the beginning.

With manual restarts, in addition to providing the necessary input file changes, you are also responsible for telling NX Nastran which major operations need to be performed. Manual restarts are more difficult to use and are more prone to user error.

Restarts can use the same solution sequence or a different solution sequence. Restarting from a statics run and requesting stress output is an example of a restart using the same solution sequence. Restarting from a statics run into a normal modes run is an example of a restart across solution sequences.

Within automatic restart, there are two different options:

- A read-write restart
- A read-only restart

Each of these is described below.

Understanding Restart Terminology

When you're performing restart analyses in NX Nastran, your analyses are categorized into cold start and restart runs.

Cold Start Runs

Your initial run is known as a “cold start” run and is identical to a regular NX Nastran job, except that you must save your database. In addition to your normal output files (e.g., the .F06 file), NX Nastran creates four database files as a result of this run. The naming convention for the filenames is machine dependent—you should refer to *NX Nastran Installation and Operations Guide* for the exact syntax for your machine. For a NX Nastran input file called “stat1.dat”, the following sample submittal command can be used:

```
nastran stat1 scr=no
```

In this case, nastran is the name of the shell script for executing NX Nastran. The “scr=no” option in the above submittal command is not required if you have not modified the default value of the submittal command as provided on the delivery media. By default, the following two database files are created as a result of the above command:

- stat1.DBALL
- stat1.MASTER

Restart Runs

Once you've performed the cold start run and saved the database, subsequent runs are referred to as "restart" runs.

16.3 Read-write Restarts

The following sections describe the changes you'll need to make to your input file to restart the analysis for a read-write restart.

NASTRAN Statement Section

This section is normally the same as your cold start run. You shouldn't change the BUFFSIZE in a restart run.

File Management Section (FMS)

This section tells NX Nastran that you are performing a restart run. In the FMS Section, only two statements are needed: the "restart" statement to tell NX Nastran that this current run is a restart run and the "assign" statement that tells NX Nastran the name of the database you want to attach.

The RESTART statement is required in any restart run. The general format for the RESTART statement is:

```
RESTART VERSION=a,b
```

where, "a" is the version from which you restart (the default value for "a" is LAST), and "b" indicates whether version "a" will be kept (KEEP) or deleted (NOKEEP) at the end of the run. The default value for "b" is NOKEEP. Due to the default values, the following two restart statements are identical:

```
RESTART VERSION=LAST,NOKEEP
RESTART
```

Whenever you perform, the software automatically creates a new version number in the database. For each restart, the current version number is incremented by one, regardless of whether the job ran successfully or not. However, there are two exceptions to this rule (which are discussed later on in the chapter).

Restart-Method 1

You also need to tell NX Nastran the database that you want to attach to your current run. There are two ways of accomplishing this goal. One way is to use the ASSIGN statement. For example, if you are restarting from the database created by stat1.dat, the following FMS statements can be used in your current run

```
RESTART
assign master= 'stat1.MASTER'
$
$   the following statement is optional
$
assign dball='stat1.DBALL'
```

Depending on the machine, the filenames may be case sensitive. They should be entered exactly as they were created in the cold start run and enclosed with a single quotes as shown above. Note

that the use of the second ASSIGN statement is optional since the “master” DBset can find the associated DBALL database.

Restart–Method 2

An alternate way to attach a restart database is to use the DBS keyword on the submittal command instead of the ASSIGN statement. Assuming the current run is called stat2.dat, then the equivalent submittal command is as follows:

```
nastran stat2 dbs=stat1
```

The ASSIGN statement is not needed in this case; however, the RESTART statement is needed using either option. While the first method requires more input than the second method, it also provides you with information about the database used for the restart run although this information is also provided in the Execution Summary Table (.F04 for most machines).

Executive Control Section

In a restart analysis, the executive control section is identical to one in a cold start analysis.

Case Control Section

When you're performing a restart analysis, the software's automatic restart logic compares the modifications you made to the Case Control and Bulk Data Sections to determine which operations need to be processed upon restart. Therefore, you must be very careful in the changes that you make in your restart run. Adhering to the following rules will avoid unnecessary reprocessing of previously completed operations:

1. You must include in your restart run all solution-type related Case Control commands that are unchanged as compared to the cold start run. In other words, do not make unnecessary load, SPC, or MPC command changes or remove them from the Case Control Section unless these are actual changes. This process is clarified later on with the example problems.
2. Output requests can be modified. A typical example can be a request of the element stress output that was not requested in the cold start run.

Bulk Data Section

The software stores a copy of the Bulk Data for each version. Your restart run must not contain any Bulk Data entries that were included with the previous version that you are restarting from since they are already saved in the database for that particular version. The Bulk Data section in the current restart run should only contain the addition of new entries, the modification of old entries, or the deletion of old entries from the database. The following is a series of three runs illustrating an application of a restart. The first run ([Listing 16-2](#)) is a sample statics run with one loading condition, and it requests displacement output.

```

$
$  filename -  stat1.dat
$
ID ROD,TEST
SOL 101
TIME 20
DIAG 8
CEND
TITLE=  COLD START RUN
$
SPC = 1
$
SUBCASE 1
LOAD = 1
DISP = ALL
$
BEGIN BULK
$
GRID,1,,0.,0.,0.
GRID,2,,5.,0.,0.
GRID,3,,10.,0.,0.
CROD,1,1,1,2
CROD,2,1,2,3
PROD,1,1,.2
MAT1,1,1.+7,,.32
FORCE,1,2,,1000.,1.,0.,0.
SPC1,1,123456,1
PARAM,AUTOSPC,YES
$
ENDDATA

```

Listing 16-2. Input File for a Cold Start Run

The second run ([Listing 16-3](#)) is a restart run with an additional load condition. Displacement and stress outputs are desired for this new loading condition. Furthermore, stress output for all the elements is desired for Subcase 1. However, since you already printed the displacement for Subcase 1 in your original run, you may elect not to reprint this information by removing the displacement request from Subcase 1. There are no model changes as compared to the original run. Note that the Bulk Data entries that were present in the original run must not be included in the current run. The only entry in the Bulk Data Section is the new FORCE entry for Subcase 2, which was not included in the original run. Since this load is a new loading condition, the FORCE entry must have a unique ID (2 in this case) as compared to the existing FORCE entry ID (1 in this case) that is stored in the database.

This type of restart run constitutes an efficient restart since the re-decomposition of the stiffness matrix is not performed; for a large problem, this can result in substantial CPU savings.


```

$ filename - stat2.dat
$
restart
assign master='stat1.MASTER'
ID ROD,TEST
SOL 101
TIME 20
DIAG 8
CEND
TITLE= RESTART RUN
$
SPC = 1
$
SUBCASE 1
LABEL = ADDITIONAL STRESS OUTPUT REQUEST
LOAD = 1
STRESS = ALL
$
SUBCASE 2
LABEL = ADDITIONAL LOAD CASE
LOAD = 2
DISP = ALL
STRESS = ALL
$
BEGIN BULK
$
$
$ NOTE THAT THE FORCE ID IS UNIQUE
$ WITH RESPECT TO EXISTING LOAD ID IN
$ THE DATABASE.
$
FORCE,2,2,, -500.,1.,0.,0.
$
ENDDATA

```

Listing 16-3. Input File for a Restart Run with an Additional Load Case

The third run ([Listing 16-4](#)) is a restart run with a modification in the material property. The same two loading conditions are used as run number two. A new material entry is added in this run and the old material entry (the eighth sorted Bulk Data from the stat2.dat run) is deleted. The general format for deleting a range of entries from the database is

```

/,x,y

```

where x and y are the range of the first and last sorted Bulk Data count to be deleted from the database, respectively. You can have as many of these entries as you need. If you are only deleting one entry ($y=x$), then either one of the following two formats can be used:

```

/,x,x
/,x

```

The sorted Bulk Data is output by default. If the ECHO Case Control command is specified in the cold start run, it should be set to output the sorted Bulk Data (e.g., ECHO=SORT).

There may be times when you may be making substantial changes to the model. In this case, it may be easier to delete the whole Bulk Data listing from the database and insert a complete new Bulk Data file as shown below:

```

/,1,z
$   include a complete new bulk data listing
$

```

In this case, z stands for an integer larger than the total number of Bulk Data entries (e.g., 999999). Alternatively, you could delete the database files and run the problem as a cold start run.

```

$ filename - stat3.dat
restart
assign master='stat1.MASTER'
ID ROD,TEST
SOL 101
TIME 20
DIAG 8
CEND
TITLE= RESTART RUN
$
SPC = 1
$
SUBCASE 1
LABEL = DISPLACEMENT AND STRESS OUTPUT REQUEST
LOAD = 1
DISP = ALL
STRESS = ALL
$
SUBCASE 2
LABEL = ADDITIONAL LOAD CASE
LOAD = 2
DISP = ALL
STRESS = ALL
$
BEGIN BULK
$
/,8
mat1,1,3.+7,,.32
$
ENDDATA

```

Listing 16-4. Input File for a Restart Run with Modified Material Property Entry

In this example, the old MAT1 entry is the eighth sorted Bulk Data (from run stat2.dat) as shown by the following partial output:

S O R T E D B U L K D A T A E C H O														
CARD COUNT	.	1	..	2	..	3	..	4	..	5	..	6	..	7
1-	CROD	1		1		1		2						
2-	CROD	2		1		2		3						
3-	FORCE	1		2				1000.	1.			0.		
4-	FORCE	2		2				-500.	1.			0.		
5-	GRID	1				0.		0.	0.					
6-	GRID	2				5.		0.	0.					
7-	GRID	3				10.		0.	0.					
8-	MAT1	1		1.+7				.32						
9-	PARAM	AUTOSPC	YES											
10-	PROD	1		1		.2								
11-	SPC1	1		123456	1									
	ENDDATA													
	TOTAL COUNT=			12										

Listing 16-5. Sorted Bulk Data Listing from a Cold Start Run

This type of restart (run number 3) requires the reformulation of the stiffness matrix since the material was changed. For a nonsuperelement run, this type of restart does not save you much in terms of runtime, if any. This run (run number 3) is included merely to show you how to delete information from the database. However, for superelement type runs, this type of restart can still save you time since you may only be resolving a portion of the structure.

The model used in the above series of runs is quite simple. In most cases, such simple models do not warrant the use of a restart since the savings are insignificant for such a small model. This simple example is chosen to illustrate the mechanics of how simple the restart feature is without being distracted by the details of a complex model.

The same restart procedure can now be applied to a larger model. The model used is "BCLL16". "BCLL16" is one of the standard test problems that is run for each new version of NX Nastran. It is basically a $16 \times 16 \times 16$ cube of CHEXA elements overlaid with a CQUAD4 elements on the face of each CHEXA element to simulate a relatively dense stiffness matrix. The model consists of 29,400 degrees of freedom. The following table illustrates the benefit of using restarts for large models.

Table 16-1. BCLL16 Performance Summary		
Run Number	Description	CPU Time (Sec)
1	Static analysis with one load case and the following output request:	372.1
	SET n = a,b,c,d DISP = n STRESS (PLOT) = All	
2	After the completion of run number 1, you now decide that you want additional printed stress output for a few selective elements. One option is to rerun the same job from the beginning with the following modified output request in the Case Control Section: SET m = x,y,z STRESS = m	364.5
3	The second option is to restart from run number 1 with the following modified output request in the Case Control Section: SET m = x,y,z STRESS = m	5.0

Note

Note the tremendous savings in the CPU time (364.5 versus 5 seconds) between the run numbers 2 and 3 with both runs accomplishing the same goal.

The cold start and restart input files used for run numbers 1 and 3 are shown in [Listing 16-6](#) and [Listing 16-7](#).

See "Mini-Database Restarts for efficient method of restarts when only data recovery is desired."

```

$ FILENAME - BCLL16A.DAT
INIT DBALL, LOGICAL=(DBALL(200000))
INIT SCRATCH, LOGICAL=(SCRATCH(200000))
ID NXN, BCLL16A
TIME 300
SOL 101
CEND
TITLE = STATIC ANALYSIS OF A CELLULAR CUBE          BCLL16A
SUBTITLE = 16 X 16 X 16 CUBE
ECHO=NONE
STRESS(PLOT)=ALL
SET 100 = 30080,40088,70880,80888
DISP = 100
LOAD=100
SPC=10
BEGIN BULK
$   BRING IN THE REST OF THE BULK DATA FILE
$
include 'bcllblk.dat'
ENDDATA

```

Listing 16-6. Input File for a BCELL16 Cold Start Run

```

$ FILENAME - RBCLL16A.DAT
$
RESTART
ASSIGN MASTER='bcll16a.MASTER'
$
$   THE ABOVE STATEMENT ASSUMES THAT YOUR
$   COLDSTART INPUT FILENAME AS
$
$   bcll16a.dat
$
ID NXN, BCLL16A
TIME 100
SOL 101
CEND
TITLE = STATIC ANALYSIS OF A CELLULAR CUBE          BCLL16A
SUBTITLE = 16 X 16 X 16 CUBE
ECHO=NOSORT
SET 100 = 30080,40088,70880,80888
set 200 = 1001,2001,3001,4001
$DISP = 100
stress = 200
LOAD=100
SPC=10
BEGIN BULK
$
$   SINCE THERE IS NO MODEL CHANGES, NO BULK
$   DATA ENTRIES ARE NEEDED
$
ENDDATA

```

Listing 16-7. Input File for a BCELL16 Restart Run

Note the user interface is quite straightforward considering the tremendous benefits in terms of CPU reduction.

16.4 Read-only Restarts

Read-only restart is very similar to read-write restart, except that original database is accessed in a read-only mode. No new data is added to the existing database. Instead, all new data is written

to the new database, leaving the original database intact. You can delete the new database or save it for another restart. The new database created during the restart run retains an internal link to the original database.

To use read-only restart for the same example problem as in [Listing 16-3](#), all you need to do is modify the FMS statements from:

Restart

```
Assign master='stat1.MASTER'
```

to:

```
Restart logical=run2
Assign run2='stat1.MASTER'
```

In this case, run2 is an arbitrary logical name.

16.5 Mini-database Restarts

You can save on the database only those data blocks that are required for data recovery. The size of the database, called a mini-database, is on average about 25% the size of a full database. For pure data recovery run (no SPC or loading changes), this type of restart is very efficient.

To use this feature, use the following submittal command in the cold start run:

```
nastran jobname scr=mini
```

In the restart run, no special input is required because NX Nastran automatically detects a mini-database and will process any output request from the table below.

The mini-database restart processes the following Case Control commands and user parameters:

DISPLACEMENT	STRAIN
ELSDCON	STRFIELD
ESE	STRESS
FORCE	SUBCOM
GPFORCE	SUBSEQ
GPSDCON	SURFACE
GPSTRESS	SYMCOM
MPCFORCES	SYMSEQ
PARAM,POST	THERMAL
REPCASE	VOLUME
SPCFORCES	

For cyclic symmetry analysis, you can't change the HOUTPUT and NOUTPUT in the restart run.

16.6 Determining the Version for a Restart

Not all versions in the database are restartable. For each run, NX Nastran prints a message near the top of the .F04 file indicating the version number of the current job. In general, if a job completes without any fatal messages, and the database is saved, then that particular version is restartable. It is a good idea to keep a brief log of all the restartable versions since this is probably the most reliable source of information regarding whether a version is restartable. If a restart job fails, e.g., due to Bulk Data error, then this newly created version is not restartable, and it is indicated as such by the following error message at the bottom of the .F06 file.

```
*** USER WARNING MESSAGE 784 (XCLNUP)
VERSION =   yy PROJECT =  "zzz"  OF THIS DATA BASE IS NOT VALID FOR RESTART
PURPOSES.
USER ACTION:
SUBSEQUENT RESTARTS SHOULD REFERENCE VERSION =   xxx OR A PRIOR VALID VERSION
```

In this case, you must restart from a previous valid version. The “xxx” and “yy” above denote version numbers. The “zzz” denotes a project description provided by you. This project description is alphanumeric and can contain up to 40 characters with the default being blank. The project description is rarely used and is an optional statement.

If for some reason the records for the old runs are no longer available, then the DBDIR FMS statement can be used to query the database contents to find out which versions are stored in the database. The following simple setup is all that is required for this purpose:

```
ASSIGN MASTER='dddd.MASTER'
DBDIR VERSION=*,PROJECT=*
ENDJOB
```

Neither the Executive, Case Control, nor Bulk Data Section is required in this case. Furthermore, a new version is not created because the ENDJOB statement is present. Near the top of the .F06 output, a PROJECT VERSION DIRECTORY TABLE is printed listing all the versions in the database. A “***” next to a version number indicates that this particular version was deleted from the database. This deletion may be due to the “NOKEEP” option or the use of the DBCLEAN statement when performing a restart run. A version number with a “***” next to it is not restartable. A version number without a “***” next to it is restartable if the run that created the version did not terminate with UFM 784. This DBDIR FMS statement can also be used to check the database directory for the existence of data blocks (e.g., UG). If the job fails very early in the run (e.g., error in the FMS section), then a new version may not be created.

Always back up your database on a regular basis. If the system aborts the run, then your database may get corrupted. Another good practice, which ensures that only good models are retained in the database, is to perform the following:

1. Use RESTART VERSION=a,KEEP
where “a” is a version number.
2. If a version contains errors, or is no longer of interest, then you can use the FMS statement DBCLEAN to remove obsolete or incorrect versions of the model from the database. Using DBCLEAN allows the Executive System to reuse some of this space for new versions.

Chapter 17: Database Concepts

- *Introduction to Database Concepts*
- *Understanding DBsets*
- *Delivery Database*
- *Deleting DBsets*
- *Database Autoassignment*
- *Database Project and Version*
- *Migrating Databases*
- *Database Archival, Compression, and Transfer*
- *Using the File Management Section with Databases*
- *Guidelines for Large Problems*

17.1 Introduction to Database Concepts

All NX Nastran runs require a database. A database allows for the storage and retrieval of data: data blocks, parameters, scratch files, and DMAP object and source files.

During a run, data may be written onto only one database. This is called the primary database which is attached automatically or by the INIT MASTER statement. Additional databases may be attached for “read-only” purposes. This includes a “delivery” database and one or more “located” databases.

- The delivery database contains solution sequences and is automatically attached by the SOL Executive statement or the ACQUIRE FMS statement.
- “Located” databases are attached by the DBLOCATE statement.

Before presenting the details of the database structure, the definitions for some of the basic terms that are used throughout this chapter need to be presented.

DBset	Database set. DBset consists of an NX Nastran logical name, such as MASTER, DBALL, etc., which refers to an entity within the NX Nastran database. It consists of a collection of physical files.
Database	The collection of all DBsets assigned to a run.
Data block	A matrix or table (e.g., KAA, LAMA) that is stored in the database.
Logical name	The local internal name (log-name) used by NX Nastran for a file or DBset.
Word	For 32- and 64-bit machines, each word is equivalent to four and eight bytes, respectively.
BUFFSIZE	The length of an NX Nastran buffer in terms of words (32- or 64-bit words). It controls the physical record size for data storage/transfer that is contained in many NX Nastran logical units. The default and maximum allowable BUFFSIZE is machine dependent. The default value is recommended except for large problems. It can be modified by using the following NASTRAN statement: <code>NASTRAN BUFFSIZE = xxxxx</code>
Block	A block is often referred to as an NX Nastran GINO block. Each GINO block contains one NX Nastran buffer. The size of each block is equal to BUFFSIZE-1 words. (GINO is an acronym for general input/output.)
{ }	A brace indicates that the quantity within this bracket is mandatory. The underlined item within { } is the default value.
[]	A square bracket indicates that the quantity within this bracket is optional.
_____	The underlined value is the default value for a particular option.

17.2 Understanding DBsets

A database is divided into several DBsets which are initialized by the INIT file management statement. There are two types of DBsets:

- Permanent DBsets, which you can save at the end of the run and reuse in a restart run
- Scratch DBsets, which are automatically deleted at the end of the run

A DBset may be composed of up to twenty concatenated physical files or DBset members. Each member is assigned to a physical file by the ASSIGN statement.

For an NX Nastran input file named “stat1.dat”, the following sample submittal command can be used:

```
nastran stat1 scr=no
```

In the command above, nastran is the name of the shell script used to execute NX Nastran. The following two permanent physical database files are created as a result of the above command.

- stat1.MASTER
- stat1.DBALL

Note that the temporary scratch files are allocated and used by NX Nastran during the solve and are deleted at the end. Unless otherwise stated, the input filename is assumed to be “stat1.dat” in this chapter.

By default, there are four DBsets—two permanent and two scratch—which are predefined and automatically assigned by the program. The DBset names of the predefined permanent DBsets are: MASTER and DBALL. The DBset-names of the predefined scratch DBsets are: SCRATCH and OBJSCR. The DBsets are briefly described below:

- The MASTER DBset is the directory of the database. It contains the names of all DBsets, DBset members and their physical file names, and a directory of projects, versions, data blocks, parameters, DMAP source and object files. It also contains the NDDL scheme used to describe the database. The default maximum size for MASTER is 5,000 blocks.
- The DBALL DBset contains all the DMAP data blocks which may be saved permanently for reuse in a subsequent run. In the solution sequences in which restarts are not possible, this DBset is empty.
- The SCRATCH DBset is a temporary DBset for all scratch data blocks and files. It has two partitions: one for DMAP data blocks and one for DMAP module internal scratch files.
- The SCR300 DBset is used as the temporary workspace for the modules. This space is released at the end of the module execution to be reused by subsequent modules. The default size is 250,000 blocks
- The OBJSCR DBset is a temporary DBset for DMAP compilation.
- If you want to create your own DMAP source and object files, you need to allocate the USROBJ and USRSOU DBsets. They contain DMAP source and object files that may be saved permanently for execution in a subsequent run. See the COMPILE Executive Control statement.

If an INIT and ASSIGN statement aren't specified for one or more of the predefined DBsets, then the corresponding statement(s) are provided by default.

```

ASSIGN MASTER='dbs-name.MASTER'
      INIT    MASTER (RAM)    LOGICAL=(MASTER(5000))
ASSIGN DBALL='dbs-name.DBALL'
      INIT    DBALL LOGICAL=(DBALL(25000))
ASSIGN USROBJ='dbs-name.USROBJ'
      INIT    USROBJ LOGICAL=(USROBJ(5000))
ASSIGN USRSOU='dbs-name.USRSOU'
      INIT    USRSOU LOGICAL=(USRSOU(5000))
ASSIGN SCRATCH='temp-name.SCRATCH'
ASSIGN SCR300='temp-name.SCR300'
      INIT    SCRATCH (MEM) LOGICAL=(SCRATCH(175000)),
      SCR300=(SCR300(175000))
ASSIGN OBJSCR='temp-name.OBJSCR'
      INIT    OBJSCR LOGICAL=(OBJSCR(5000))

```

The numbers inside the parentheses are the maximum sizes of the DBsets in GINO blocks. By default, one member with the same log-name as the DBset-name is assigned to each DBset. (For a description of a GINO block, DBset-name and log-name, see the INIT statement). For example, the DBALL DBset has one member also called DBALL. The dbs-name is determined from the dbs keyword on the nastran command and temp-name is generated by the command procedure. The scratch DBsets are assigned special names on a scratch disk. (For a description of the dbs-name and temp-name, see the *NX Nastran Installation and Operations Guide*).

The filenames above are those that would be typically generated on Linux computers. For example, if the name of the input file is called MYJOB.DAT and the dbs keyword isn't specified, then the default filenames for the permanent DBsets would be:

- MYJOB.MASTER
- MYJOB.DBALL

Also, they would have the same directory or path as the input file.

A physical filename is automatically created for any new DBsets specified by an INIT statement. For example, to create a new DBset called DBUP for split database operations, specify:

```
INIT DBUP
```

The physical filename will then be: MYJOB.DBUP. For another example, to create two members for DBALL, specify

```
INIT DBALL LOGI=(DB1 ,DB2)
```

Their physical filenames will be MYJOB.DB1 and MYJOB.DB2.

You can use the ASSIGN statement to override these default names. For example, to rename MYJOB.DBUP to FENDER.DBUP, specify:

```

ASSIGN DBUP='FENDER.DBUP'
INIT DBUP

```

To rename MYJOB.DB1 and MYJOB.DB2, specify

```

ASSIGN DB1 ='FENDER.DBALL1'
ASSIGN DB2='FENDER.DBALL2'
INIT DBALL LOGI=(DB1,DB2)

```

The INIT statement may be used to change the maximum size of the predefined DBsets. (The size may be specified in blocks, kilowords, megawords, kilobytes, megabytes, gigawords, or gigabytes.) For example, to specify a size of 50,000 blocks for DBALL,

```
INIT DBALL LOGICAL=(DBALL(50000))
```

or 50,000 bytes,

```
INIT DBALL LOGICAL=(DBALL(50KB))
```

The INIT and ASSIGN statement may be used to specify more than one member for a DBset which may exist on different physical devices.

```
ASSIGN DB1 ='physical file name of DB1 on disk 1'
ASSIGN DB2='physical file name of DB2 on disk 2'
INIT DBALL LOGICAL=(DB1(5000),DB2(5000))
```

17.3 Delivery Database

The Delivery Database contains the solution sequences. This database, in addition to the primary database, is automatically assigned in order to execute an NX Nastran solution sequence. For example, the filenames of the database are

```
SSS.MASTERA
SSS.MSCOBJ
SSS.MSCSOU
```

If the solution sequence is not an NX Nastran solution sequence, as in the case of a user's solution sequence; then the ACQUIRE FMS statement may be used to select the appropriate database. ACQUIRE NDDL selects the SSS database. The ACQUIRE statement is required when the DBLOAD FMS statement is used with the ENDJOB FMS statement.

17.4 Deleting DBsets

There are several different ways that you can control what happens to permanent DBsets in the primary database at the end of a solve.

- If scr=yes is specified on the nastran command or the INIT MASTER(S) statement appears in the FMS Section, then NX Nastran deletes all DBsets in the primary database.
- An individual DBset member is automatically deleted at the end of the run by specifying the TEMP keyword on the ASSIGN statement. For example, to delete the USROBJ and USRSOU DBsets at the end of the run, specify:

```
ASSIGN USROBJ='A' TEMP
ASSIGN USROBJ='B' TEMP
```

Note: If you use this method to delete a DBset, you can't recreate it for a future run.

- The DBSETDEL FMS statement provides a better method than the TEMP keyword on the ASSIGN statement except that the DBsets are deleted at the beginning of the run. The deleted DBsets may be recreated in a future run. For example:

```
DBSETDEL USROBJ,USRSOU
```

17.5 Database Autoassignment

When a database is created, the physical filenames of all DBset members are loaded into the MASTER DBset. Therefore, when using DBLOCATE or RESTART, it is sufficient to assign the MASTER DBset member. All other DBset members previously initialized, i.e., DBALL, DBUP, etc., are automatically assigned by the program. For example, on restart, the following input will automatically assign the DBALL, USROBJ, and USRSOU DBsets:

```
ASSIGN MASTER='filename of the MASTER DBset member'
RESTART
```

If DBLOCATE is used, then:

```
ASSIGN MSTR1='filename of the MASTER DBset member'
DBLOCATE LOGI=MSTR1
```

The autoassignment feature cannot assign the physical file of a DBset member that has been renamed. This is because the MASTER DBset member contains the old physical filename. Therefore, an additional ASSIGN statement is required to specify the new filename of the DBset member. If in the previous restart example DBALL is renamed, then the following input is required:

```
ASSIGN MASTER='filename of the MASTER DBset member'
ASSIGN DBALL='new filename of the DBALL DBset member'
RESTART
```

In split database operations, it is often necessary to assign only some of the DBsets because the others are “offline.” The NASTRAN AUTOASGN statement is used to specify the types of databases that will be autoassigned:

1. AUTOASGN = 0: No databases will be autoassigned. This includes the primary database, Delivery Database, and any located databases.
2. AUTOASGN = 1: All databases will be autoassigned. This is the default.
3. AUTOASGN = 2: Only the Delivery Database will be autoassigned.
4. AUTOASGN = 4: Only located databases will be autoassigned.

If some combination of the options above is desired, then sum their values. For example, if the delivery and located databases are to be autoassigned, then specify NASTRAN AUTOASGN = 6.

17.6 Database Project and Version

All data stored on the primary database is assigned to a particular project identification string (ID) and version identification number (ID). The current project ID is specified on the PROJECT FMS statement. The current version ID is an integer value assigned by the program each time the database is attached as the primary database. When a primary database is initialized or a new project is specified, then all items stored on the database during the run are stored under a version ID of integer 1. Then, in subsequent runs a new version is created with an ID equal to the last version ID incremented by one. If the RESTART FMS statement is specified, then a new version of data is created which is equivalent to the last version. In other words, all items in the old version are made available for reuse in the new version.

17.7 Migrating Databases

Databases between versions in general aren't compatible and must be converted, or migrated, before you can use them in a higher version of the software. This is required primarily because NDDL sequences change. For example, some qualifiers may have been added to the paths of some data blocks in order to support new capabilities in the solution sequences. Also, NDDL description of some data blocks change.

Importantly, a database created in nonlinear analysis, heat transfer analysis, or optimization analysis cannot be fully migrated. This is because of changes in Bulk Data entry formats and/or data blocks. Also, it would be inefficient to recreate because it is created by modules which perform the bulk of a nonlinear analysis.

You can migrate a database in a single run which consists of two phases.

- In the first phase, the DBLOCATE FMS statement with the CONVERT and COPY keywords is used to copy most of the data blocks and all of the parameters from the earlier version database to the current version primary database.
- In the second phase, the remaining data blocks which cannot simply be copied into the current version have to be regenerated. This involves the execution of a short solution sequence that regenerates those data blocks: specify SOL DBTRANS (or SOL 190).

For example, the input file necessary to migrate a database created by one of the structured solution sequences (SOLs 100 through 200) is:

```
ASSIGN MIG='physical filename of the MASTER DBset'
REFINCL DBTRANS
SOL DBTRANS
CEND
```

With the DBTRANS solution sequence, it is assumed that the early version database contains all of the data from a completed and successful analysis. If not, then the solution sequences may fail and the migration may not be successful. In this case, it is possible to alter DBTRANS with the ALTER statement to obtain the appropriate data.

If the earlier version database wasn't created by an NX Nastran solution sequence, then it may be necessary to develop new DBLOCATE statements and a new migration solution sequence.

17.8 Database Archival, Compression, and Transfer

Database archival, compression and transfers are performed using the DBUNLOAD and DBLOAD FMS statements. You can select all items on the database or selected items according to the WHERE clause.

Database Archival and Retrieval

The DBUNLOAD FMS statement is used to archive the primary database. It writes out the data blocks and parameters stored in the database to a sequential file in a binary format which is similar to that of the OUTPUT2 module (see *NX Nastran DMAP Programmer's Guide*). A database can only be unloaded in a run separate from the run in which it is created or modified. The default action is to archive all data blocks and parameters under all projects and versions in the database. The ASSIGN FMS statement is also required to assign a FORTRAN unit on which to unload the database. The default unit number is 50. This unit number and other default attributes are predefined under the logical keyword DBUNLOAD. For example, a simple request to unload or archive a database is as follows:

```
ASSIGN DBUNLOAD='physical filename of archive file'
DBUNLOAD
ENDJOB
```

The DBLOAD FMS statement is used to read an archived database and copy it to the primary database. The ACQUIRE statement is required to select the appropriate NDDL sequence. The ASSIGN FMS statement is also required to assign a FORTRAN unit to the unloaded database. The default unit number is 51. This unit number and other default attributes are predefined under the logical keyword DBLOAD. For example, a simple request to reload the archived database above is as follows:

```
ACQUIRE NDDL
ASSIGN DBLOAD='physical filename of archive file'
DBLOAD
ENDJOB
```

The example above makes use of the ENDJOB statement, which means that only the FMS statements will be processed in the run. A new BUFFSIZE may be assigned to the retrieved database with the NASTRAN statement. SOL DBTRANS or DBTRANU must also be executed:

```
NASTRAN BUFFSIZE=xxxx
ASSIGN DBLOAD='physical filename of archive file'
DBLOAD
SOL DBTRANS
CEND
```

By default, the DBLOAD and DBUNLOAD statements will load and unload all the data under all projects and versions found in the database. The WHERE clause may be used to be more selective according to the project ID, version ID, item name, qualifier values, and/or DBset. Also, the CONVERT clause may be used to modify the project ID, version ID, item name, qualifier values, and/or DBset.

Database Compression

When data is deleted from the database, its space is released for the storage of new data. This old space will be reused before any new space is used. However, even if the space is not reused, the size of the database is not reduced. After several restarts or the execution of the DBCLEAN FMS statement, the database may contain a significant amount of released or “dead” space. There are two methods of removing dead space or “compressing” the database. The first method involves the DBUNLOAD and DBLOAD FMS statements, and the examples shown above will also perform a database compression. The second and simplest method involves the DBLOCATE statement specified with the COPY keyword. For example, if the database was created with SOLs 100 through 200, then the input file is:

```
ACQUIRE NDDL
ASSIGN MYDB='physical file name of the MASTER DBset'
DBLOCATE LOGI=MYDB COPY
ENDJOB
```

It is also possible to compress an individual DBset and change its maximum size. For example, the input file is:

```
ACQUIRE NDDL
ASSIGN MYDB='physical file name of the MASTER DBset'
DBLOCATE LOGI=MYDB COPY WHERE (DBSET='DBALL')
INIT DBALL LOGICAL=(DBALL(10000))
ENDJOB
```

Transferring Databases Across Different Platforms

You can also use the DBUNLOAD and DBLOAD statements to copy a database from one computer platform to another.

- On the first computer, the DBUNLOAD statement converts the database to a “neutral” file on the first computer. This neutral file is now suitable for a simple copy to the second computer or may be accessed through a network if the two computers are so connected.
- On the second computer, the DBLOAD statement converts the neutral file back to a database. It is also necessary to execute the solution sequence DBTRANS (or 190) as described under Database Migration, because some data blocks cannot be converted to a neutral format. (In SOL DBTRANS; data blocks EST, EMAP, ETT, PTELEM, SLT, KDICT, KELM, MDICT, MELM, BDICT, BELM, and ACPT are regenerated.)

The ASSIGN statement is also required to assign the physical filename of the neutral file and specify that the file is formatted since the default format (UNFORMATTED) is not transferable across dissimilar computers.

The following example unloads a database created by one of the unstructured solution sequences on the first computer to a neutral file:

```
ASSIGN DBUNLOAD='physical filename of neutral file' FORMATTED
DBUNLOAD FORMAT=NEUTRAL
ENDJOB
```

Then the neutral file is copied to, or accessed through a network by, the second computer:

```
ASSIGN DBLOAD='physical filename of neutral file' FORMATTED
DBLOAD FORMAT=NEUTRAL
SOL DBTRANU
CEND
```

With the DBTRANS solution sequence it is assumed that the neutral file contains all of the data from a completed and successful analysis. If not, then the solution sequences may fail and the transfer may not be successful. In this case, it is possible to alter DBTRANS with the ALTER statement to obtain the appropriate data.

If the database was created in a heat transfer analysis then NASTRAN HEAT = 1 must be specified.

If the database on the first computer was not created by a NX Nastran solution sequence, then it may be necessary to develop a new transfer solution sequence.

17.9 Using the File Management Section with Databases

The File Management Section (FMS) is intended primarily for the attachment and initialization of NX Nastran database sets (DBsets) and FORTRAN files. For many problems due to the default values, the FMS is handled automatically and is not required to be specified by you.

You can allocate the database file size in any of the following units:

- Bytes
- Words
- Megabytes (1,000,000 bytes)
- Megawords (1,000,000 words)
- Kilobytes (1,000 bytes)
- Kilowords (1,000 words)
- NX Nastran GINO blocks (default)
- Mixture of the above units

Internally, NX Nastran converts these units to GINO blocks. If an FMS statement is longer than 72 characters, it must be continued on the next line. You can have a maximum of 200 files (FORTRAN and DBsets) assigned (online) at a time. This limit may be less for some machines due to their operating system limit.

Using the INIT Statement

INIT creates/initializes permanent and/or temporary DBsets. The INIT statement has two basic formats: one for all the DBsets and one specifically for the SCRATCH DBsets.

Format (Simplified) for All DBsets Except SCRATCH DBsets

```
INIT DBset-name [LOGICAL=(log-name1(max-size1) ,
log-name2(max-size2) , ...log-namei(max-sizei) ,
...log-namen(max-sizen) ) ]
```

DBset-name The logical name of the DBset being used (e.g., DBALL).

log-name <i>i</i>	The <i>i</i> -th logical name for the DBset-name referenced in this INIT statement. You can have up to 20 logical names for each DBset ($1 \leq i \leq 20$). An ASSIGN statement may be used to attach a physical file to the <i>i</i> -th logical file.
max-size <i>i</i>	The maximum allowable size that may be written to the <i>i</i> -th file.

Example A

The following statement creates the DBALL DBset with a logical name of DBALL and the maximum size of 50,000 NX Nastran blocks instead of 250,000 blocks, which is the default value for DBALL:

```
INIT DBALL LOGICAL=(DBALL(50000))
```

Assuming that your input file is called "stat1.dat", the physical file has the name stat1.DBALL unless an ASSIGN statement is also used.

The following statement creates the DBALL DBset with logical names of DB1 and DBTWO:

```
INIT DBALL LOGICAL=(DB1(35000),DBTWO(60000))
```

The two physical files stat1.DB1 and stat1.DBTWO are created with a maximum of 35,000 and 60,000 NX Nastran blocks, respectively.

Format (Simplified) for the SCRATCH DBset

```
INIT SCRATCH [LOGICAL=(log-name1(max-size1) ,
log-name2(max-size2),...log-namei(max-sizei)) ,
SCR300=(log-namei+1(max-sizei+1),...log-namen(max-sizen))]
```

Log-name1 through log-name*i* are allocated for regular scratch files as temporary workspace. This space is not released until the end of the job. SCR300 is a special keyword that indicates the log-names are members reserved for DMAP module internal scratch files. The space occupied by these files is for the duration of the execution of the module. This space is released at the end of the module execution. You can have up to a combined total of 20 logical names for the SCRATCH DBset ($1 \leq i \leq 20$).

Example B

The following statement creates the SCRATCH DBset with logical names of SCR1, SCR2, SCRA, and SCRB:

```
INIT SCRATCH LOGICAL=(SCR1(150MW),SCR2(100MW)) ,
SCR300=(SCRA(250MW),SCRB(300MW))
```

The two physical files stat1.SCR1 and stat1.SCR2 are created with maximum sizes of 150 and 100 megawords, respectively. These two files are regular scratch files. Two additional physical files stat1.SCRA and stat1.SCRB are created with maximum sizes of 250 and 300 megawords, respectively. These last two files are SCR300-type files.

Using the ASSIGN Statement

Use ASSIGN to assign a physical filenames to logical filenames or special FORTRAN files that are used by other FMS statements or DMAP modules.

Format (Simplified) to Assign Logical Files

```
ASSIGN log-namei='filenamei' [TEMP DELETE]
```

log-namei	The i-th logical name for the DBset created by the INIT statement.
TEMP	Requests that filenamei be deleted at the end of the job.
DELETE	Requests that filenamei be deleted if it exists before the start of the run. This is optional; however, if this option is not used and the file exists prior to the current run, then the job may fail with the following messages:

```
*** USER FATAL MESSAGE 773 (DBDEF)
    THE FOLLOWING PHYSICAL FILE ALREADY EXISTS.
    LOGICAL NAME   = xxxx
    PHYSICAL FILE  = yyyy
    USER INFORMATION: NO ASSOCIATED DEFAULT FILES OR ASSIGNED DBSETS CAN
    EXIST PRIOR TO THE DATA BASE INITIALIZATION RUN.
    USER ACTION: DELETE THIS FILE AND RESUBMIT THE JOB.
```

DELETE is not a suggested option if you are using RESTART since you can delete your database inadvertently. Manual deletion of unwanted databases is a safer approach.

Example C

The following statements create a logical name DB1 for the DBset DBALL in the current directory:

```
ASSIGN DB1='sample.DB1'
INIT DBALL LOGICAL=(DB1(50000))
```

The physical file sample.DB1 is created in this case. Without the ASSIGN statement, the physical filename created is called stat1.DB1, assuming once again that your input file is called stat1.dat.

The following statements create the two logical names DB1 and DB2 for the DBset DBALL:

```
ASSIGN DB1='/mydisk1/se/sample.DB1'
ASSIGN DB2='/mydisk2/sample.DB2'
INIT DBALL LOGICAL=(DB1(50000),DB2(40000))
```

DB1 points to a physical file called sample.DB1 that resides in the file system (directory) /mydisk1/se. DB2 points to a physical file called sample.DB2 that resides in the file system (directory) /mydisk2.

Format (Simplified) to Assign FORTRAN Files

```
ASSIGN log-key='filenamef' [STATUS={new,old},UNIT=u ,
FORM={FORMATTED,UNFORMATTED},DELETE]
```

log-key	<p>This is the logical keyword for the FORTRAN file being assigned. This file may already exist. The default value depends on the keyword. Acceptable keywords are</p> <p>DBC, DBMIG, INPUTT2, INPUTT4, OUTPUT2, OUTPUT4, DBUNLOAD, DBLOAD, and USERFILE.</p>
---------	---

See the *NX Nastran Quick Reference Guide* for detailed descriptions of these keywords and their default values.

filenamef	The physical name of the FORTRAN file.
STATUS	Specifies whether the FORTRAN file is created (STATUS = new) or is an existing file (STATUS = old).
UNIT	Specifies the FORTRAN unit (e.g., UNIT = 12).
FORM	Specifies whether the file written is in ASCII (FORM = FORMATTED) or binary (FORM = UNFORMATTED) format. Note that if the log-key is OUTPUT2, the FORMATTED output is in compressed ASCII format.
DELETE	Requests that filenamef be deleted, if it exists before the start of the run.

Example D

The following example creates a new FORTRAN file to be used for OUTPUT2 operations:

```
ASSIGN OUTPUT2='sample.out',STATUS=NEW,UNIT=11,FORM=FORMATTED
```

This FORTRAN file is in compressed ASCII format with a filename of sample.out and is assigned to unit 11. Note that a compressed ASCII file can be transferred directly across machines, but it should not be edited.

Using the EXPAND Statement

Expand concatenates files into an existing DBset in order to increase the allowable disk space available for NX Nastran. The EXPAND statement is normally used in a restart run when you exceed the disk space allocation in your previous run.

Format

```
EXPAND DBset-name LOGICAL=(log-namei(max-sizei),... )
```

DBset-name	The logical name of the DBset to be expanded by the addition of new members to an existing DBset previously defined with an INIT statement.
log-namei	The logical name of the i-th member of the DBset. An ASSIGN statement should be used to point this logical name to a physical file.
max-sizei	The maximum size of the i-th member.

Only one EXPAND statement is allowed per run.

Example E

The original run creates a database with the name stat1.DBALL. However, this database was filled and the job failed with the following error messages in the .F06 file:

```
*** USER FATAL MESSAGE 1012 (GALLOC)
DBSET DBALL IS FULL AND NEEDS TO BE EXPANDED.
```

For small to medium problems, it is best to rerun the job from the beginning with a larger file allocation. For large problems, if rerunning the job is not practical, then the database can be expanded with the following FMS statements:

```
RESTART
```

```

ASSIGN MASTER='stat1.MASTER'
ASSIGN DBADD='morespace.DB'
EXPAND DBALL LOGICAL=(DBADD(50000))

```

These statements assign an additional member, with a logical name of DBADD, to the existing DBset DBALL. This member points to a new physical file called morespace.DB, which may contain up to a maximum of 50,000 NX Nastran blocks. You are restarting from “stat1.MASTER” in this case.

The EXPAND statement cannot be used for the scratch files since they are deleted at the end of each job.

Using the RESTART Statement

RESTART allows you to continue from the end of the previous run without solving the problem from the beginning.

Format

```
RESTART [PROJECT='proj-ID', VERSION={version-ID, LAST}, {KEEP, NOKEEP}]
```

proj-ID	Project identifier used in the original run. It can have up to 40 characters. This option is optional and is normally not used. The default proj-ID is blank.
version-ID	The version number that you are restarting from.
KEEP	If this option is used, then the version that you are restarting from is also saved at the end of the current run.
NOKEEP	If this option is used, then the version that you are restarting from is deleted at the end of the current run.

Example F

The following statement causes the current run to use the last version in the database for the restart:

```
RESTART
```

At the end of the run, this last version that you are restarting from is deleted from the database. This is probably the most commonly used form for RESTART.

The following statement instructs the current run (version 6 or higher) to use version 5 in the database for restart:

```
RESTART VERSION=5,KEEP
```

At the end of the run, version 5 is also retained in the database. This format is used most often when you want to ensure that a specific version is saved in the database (e.g., a large run from which you may want to request additional data recovery in the future).

The following statement specifies that the current run (version 4 or higher with a proj-ID of xyz) uses version 3 with a proj-ID of xyz in the database for restart:

```
RESTART PROJ='xyz' VERSION=3
```

At the end of the run, version 3 with a proj-ID of xyz is deleted from the database.

Using the DBCLEAN Statement

DBCLEAN deletes unwanted versions from the database. The DBCLEAN statement does not reduce the size of the database. This statement deletes the data blocks for the specified versions to allow other data blocks to reuse this space for subsequent restarts without potentially increasing the database size. Up to ten DBCLEAN statements can be used in the FMS for each run.

Format

```
DBCLEAN VERSION={version-ID,*} [PROJECT={'project-ID',*}]
```

version-ID	The version number that you want to remove from the database.
project-ID	The identifier of the project to be deleted. The default is blank, which is the most commonly used form.
*	This is a wildcard command. It removes all versions or projects from the database.

Example G

The following statements delete versions 3 and 6 with a blank proj-ID from the database in the current run:

```
DBCLEAN VERSION=3
DBCLEAN VERSION=6
```

Using the DBDIR Statement

The DBDIR statement prints the database directory.

Format

```
DBDIR [VERSION={version-ID,*}] [PROJECT={proj-ID,*}] [FORMAT={format-no}]
```

version-ID	Version number.
proj-ID	Project number.
*	This is a wildcard command. It refers to all versions or projects.
	This controls the type of directory information printed. The value is the sum of the desired types listed below. The default value is 63 (sum of first six values), which means the first six types will be printed.
format-no	<ul style="list-style-type: none"> 1 = Project version table 2 = NDDL data blocks 4 = NDDL parameters 8 = NDDL empty data blocks 16 = NDDL data block trailers 32 = NDDL path value table for NDDL entries 64 = Scratch data blocks

Example H

The following statement causes the printing of the database directory for all versions of the current project-ID in the database:

```
DBDIR
```

Using the INCLUDE Statement

The INCLUDE statement inserts an external file at the location where the INCLUDE statement is used. It is not a pure FMS statement because it can be used anywhere in the input file, not just in the FMS section.

Format

```
Include 'filename'
```

filename Physical filename of external file to be inserted at this location.

The following run reads a file called sub1.dat with all the Case Control commands contained in it:

```
Sol 101
cend
include 'sub1.dat'
begin bulk
$
include 'bulk1.dat'
include 'bulk2.dat'
$
$           rest of bulk data file
$
.
.
enddata
```

This run also brings two additional files (bulk1.dat and bulk2.dat) into the Bulk Data Section. You may, for example, want to include all your grid entries in file bulk1.dat and all your element connectivities in bulk2.dat. As you can see, the INCLUDE statement can be a handy tool. For parametric studies, you can potentially save a tremendous amount of disk space by using the INCLUDE statement instead of having multiple files with duplicate input data.

17.10 Guidelines for Large Problems

In general, you can solve small to medium sized problems in NX Nastran with very little knowledge of FMS statements and database structure. However, when you're solving large problems, knowledge of FMS statements and database structure can help you to allocate and optimize your computer resources.

When running large problems, you should have some idea of the resource requirements—both disk space and computer time—prior to engaging in the actual analysis. A utility program called ESTIMATE provides a set of recommended resource values for running a particular job.

For most small to medium problems, the default values for all the file assignments should be more than adequate. However, for a large problem, larger disk space allocation may be necessary. Several examples are shown in this section that illustrate various options and consequences with each option. The submittal command for each example is shown followed by a partial listing of the corresponding input file. In each case, it is assumed that you are running on a Linux machine with “nastran” as the name of the shell script executing NX Nastran.

See also

- “ESTIMATE” in the *NX Nastran Installation and Operations Guide*

Example I

The following submittal command and FMS statements allocate 300,000 and 5,000 NX Nastran GINO blocks for DBALL and MASTER, respectively:

```
nastran runa scr=no
$
$ filename - runa.dat
$
INIT DBALL,LOGICAL=(DBALL(300000))
INIT SCRATCH,LOGICAL=(SCRATCH(200000)) ,
    SCR300=(SCR300(200000))
$
$ THE REST OF YOUR INPUT FILE
$
```

Furthermore, the following physical files are created and saved in your current directory:

- runa.DBALL
- runa.MASTER

These statements also allocate 200,000 and 200,000 NX Nastran GINO blocks for SCRATCH and SCR300, respectively. These two files are allocated to the default scratch directory during the course of the run and are deleted at the end of the run. Since the database is saved, this job is restartable. Other than the allocation size of the databases, this is identical to the default setup.

When using the “scr = yes” option on the submittal command, the permanent data blocks that are normally written to the DBALL DBset are now written to the SCRATCH DBset.

Example J

The submittal command and FMS statements below allocate 50, 500, and 800 megabytes for DBALL, SCRATCH, and SCR300, respectively. Furthermore, by default, 5,000 NX Nastran GINO blocks are allocated each for the MASTER dbset. During the course of the run, all of these files are allocated to your default scratch directory. Because you are using the “scr=yes” option on the submittal command, the permanent data blocks that are normally written to the DBALL DBset are now written to the SCRATCH DBset. In other words, you must allocate the disk space to the SCRATCH DBset, which you would normally allocate to the DBALL DBset, when you use the “scr=yes” option on the submittal command. At the end of the run, all of these files are deleted automatically. Since the database is not saved, this job is not restartable.

```
nastran runb scr=yes
$
```

```

$ filename - runb.dat
$
INIT DBALL, LOGICAL=(DBALL(50MB))
INIT SCRATCH, LOGICAL=(SCRATCH(500MB)) ,
    SCR300=(SCR300(800MB))
$
$ THE REST OF YOUR INPUT FILE
$

```

Example K

If you have sufficient disk space in your current and default scratch file systems (disk packs), the method shown in “[Example A](#)” or “[Example B](#)” is the preferred procedure since it is the simplest method. However, if you do not have sufficient disk space in these two file systems, but have other file systems (e.g., /disk2, /disk3, and /disk4) mounted to your system, then the procedure listed below can be used. You must have read and write privileges in order to use these file systems. For this example, the three new file systems are /disk2, /disk3, and /disk4.

```

nastran runc scr=no

$
$ filename - runc.dat
$
ASSIGN DBALL1='runc.db1'
ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrch2'
ASSIGN SC3B='/disk4/user_guide/statics/runc.sc3b'
$
INIT DBALL, LOGICAL=(DBALL1(200000KB), DBALL2(100000KB))
INIT SCRATCH, LOGICAL=(SCRATCH1(100000), SCRATCH2(100000)) ,
    SCR300=(SC3A(150000), SC3B(50000))
$
$ THE REST OF YOUR INPUT FILE
$

```

The permanent database is broken up into two separate logical files (DBALL1 and DBALL2). The above run allocates 200,000 and 100,000 kilobytes for DBALL1 and DBALL2, respectively. The run also allocates 5,000 GINO blocks for the MASTER dbset. DBALL1 is allocated to the current directory that you are running on with the physical filename of “runc.db1”. DBALL2 is allocated to the “/disk2/user_guide/statics” directory with a physical filename of “runc.db2”. The following additional physical file is also created in your current directory:

```
runc.MASTER
```

These three files are saved at the end of the run and the job is restartable.

The regular SCRATCH and SCR300 files are each divided into two separate files. Two logical files (SCRATCH1 and SCRATCH2) are assigned to the regular SCRATCH file. SCRATCH1 is assigned to the default scratch directory requesting 100,000 GINO blocks. SCRATCH2 is assigned to the “/disk3/user_guide/statics” directory with a filename of “runc.scrch2” and requesting 100,000 GINO blocks. The SCR300 file is also divided into two files (SC3A and SC3B). SC3A requests 150,000 blocks and assigns this disk space to the default scratch directory. SC3B requests 50,000 blocks and assigns this disk space to the “/disk4/user_guide/statics” directory with a filename of “runc.sc3b”. All four of these scratch files are deleted at the end of the run.

```
nastran rund scr=yes
```


Example L

Even though the setup in “[Example D](#)” looks virtually the same as that in “[Example C](#),” the “scr=yes” keyword causes the permanent data blocks—that are normally written to the DBALL DBset—to be written to the SCRATCH DBset. In other words, except for the amount used for overhead, the space that you have allocated to DBALL1 and DBALL2 is not used at all. You must adjust the disk space allocation to the SCRATCH DBset that you normally would allocate to the DBALL DBset when you use the “scr=yes” option on the submittal command. The correct action in this case is to increase the disk space allocation for SCRATCH1 and SCRATCH2, and decrease the disk space allocation for DBALL1 and DBALL2. Once again, since the database is not saved, this job is not restartable.

```
$
$ filename - rund.dat
$
$ ASSIGN DBALL1='rund.db1'
$ ASSIGN DBALL2='/disk2/user_guide/statics/runc.db2'
$ ASSIGN SCRATCH2='/disk3/user_guide/statics/runc.scrch2'
$ ASSIGN SC3B='/disk4/user_guide/statics/runc.sc3b'
$
$ INIT DBALL, LOGICAL=(DBALL1(200000KB), DBALL2(100000KB))
$ INIT SCRATCH, LOGICAL=(SCRATCH1(100000), SCRATCH2(100000)) ,
$ SCR300=(SC3A(150000), SC3B(50000))
$
$ THE REST OF YOUR INPUT FILE
$
```

BUFFSIZE

BUFFSIZE is the length of an NX Nastran buffer in terms of words (32- or 64-bit words). It controls the physical record size for data storage/transfer that is contained in many NX Nastran logical units. The default and maximum allowable BUFFSIZE is machine dependent. The default value is recommended except for large problems. Each 32-, and 64-bit word contains 4 and 8 bytes, respectively. This feature can be invoked by including the following NASTRAN statement in your input file.

```
NASTRAN BUFFSIZE=xxxxx
```

where $xxxxx = (n \cdot \text{disk-block-size}) + 1 \leq \text{limit}$ and n is a positive integer.

The acceptable values of disk-block-size and limit are machine dependent and are defined in [Table 17-1](#) for most machines.

A larger BUFFSIZE takes fewer requests of the computer operating system to transfer the same amount of data, with each transfer involving a larger physical record size. This can reduce the I/O time which, in turn, reduces the elapsed time. In general, the effect on CPU time is insignificant. A larger BUFFSIZE decreases the number of I/O operations but may increase the database size. Since each data block uses at least one buffer and the read/write operation requires a minimum of one buffer, the default BUFFSIZE is recommended except for large problems. See *NX Nastran Installation and Operations Guide* for the default BUFFSIZE for various machines.

Disk space allocation is affected by the BUFFSIZE if the disk space units are in terms of GINO blocks. GINO blocks are related to words by the following equation:

$$\text{Number of words} = \text{Number of GINO blocks} \cdot (\text{BUFFSIZE} - 1)$$

Equation 17-1.

“[Example E](#)” illustrates the use of a non-default BUFFSIZE.

Example M

Assume the machine that you are running on has a default BUFFSIZE of 2049. If you increase the BUFFSIZE to 17291, using [Eq. 17-1](#), this example allocates the same maximum amount of physical disk space as “[Example A](#),” which uses the default BUFFSIZE.

```
$
$ filename - rune.dat
$
NASTRAN BUFFSIZE=17291
INIT DBALL,LOGICAL=(DBALL(35536))
INIT SCRATCH,LOGICAL=(SCRATCH(23691)),SCR300=(SCR300(23691))
$
$ THE REST OF YOUR INPUT FILE
$
```

Assuming you have disk space, is there any other reason why you would not want to use a larger BUFFSIZE for a large problem? The answer is “no” for most machines, provided you have sufficient memory available on your machine. For some machines, optimum values are preselected, and you should use these default values. See Reference 16. for further details regarding your particular machine. Total memory allocation can be controlled by the “mem” keyword on the submittal line. The following example allocates 100 mb of total memory for the run.

```
nastran stat1 mem=100mb
```

A portion of this total memory is allocated to the Executive System. The rest of the memory is then available to the functional modules in NX Nastran. Assuming that you are using the default SCRATCH(MEM) and BUFFERPOOL, the portion of the memory that is available for your problem is denoted as “User OPENCORE (HICORE)”, and can be estimated by [Eq. 17-2](#).

$$\text{User OPENCORE} \cong \text{mem} - (\text{BUFFSIZE} \cdot 148) - 89,315$$

Equation 17-2.

As you can see from [Eq. 17-2](#), the amount of memory allocated to the Executive System is a function of the BUFFSIZE. As the BUFFSIZE increases, a larger portion of the total memory is allocated to the Executive System. A summary of the NX Nastran memory utilization is printed near the beginning of the .F04 file. Some typical examples of the memory summary tables are shown in [Listing 17-1](#), [Listing 17-2](#), and [Listing 17-3](#). You can also let NX Nastran estimate the amount of required memory by using the following submittal command:

```
nastran stat1 mem=estimate
```

** MASTER DIRECTORIES ARE LOADED IN MEMORY.			
USER OPENCORE (HICORE)	=	1607959	WORDS
EXECUTIVE SYSTEM WORK AREA	=	81854	WORDS
MASTER (RAM)	=	30000	WORDS
SCRATCH (MEM) AREA	=	204900	WORDS (100 BUFFERS)
BUFFER POOL AREA (EXEC)	=	76183	WORDS (37 BUFFERS)
TOTAL NX NASTRAN MEMORY LIMIT	=	2000896	WORDS

Listing 17-1. Memory Summary Table(mem = 2MW, bufsize = 2049)

```

** MASTER DIRECTORIES ARE LOADED IN MEMORY.
  USER OPENCORE (HICORE)      =    3607831  WORDS
EXECUTIVE SYSTEM WORK AREA    =      81854  WORDS
MASTER (RAM)                  =      30000  WORDS
SCRATCH (MEM) AREA            =    204900  WORDS (    100 BUFFERS)
BUFFER POOL AREA (EXEC)       =      76183  WORDS (     37 BUFFERS)
TOTAL NX NASTRAN MEMORY LIMIT =    4000768  WORDS

```

Listing 17-2. Memory Summary Table(mem = 4MW, buffsize = 2049)

```

** MASTER DIRECTORIES ARE LOADED IN MEMORY.
  USER OPENCORE (HICORE)      =    1352015  WORDS
EXECUTIVE SYSTEM WORK AREA    =     249516  WORDS
MASTER (RAM)                  =      30000  WORDS
SCRATCH (MEM) AREA            =    1729100  WORDS (    100 BUFFERS)
BUFFER POOL AREA (EXEC)       =     640137  WORDS (     37 BUFFERS)
TOTAL NX NASTRAN MEMORY LIMIT =    4000768  WORDS

```

Listing 17-3. Memory Summary Table(mem = 4MW, buffsize = 17291)

Chapter 18: Inertia Relief in Linear Static Analysis

- *Introduction to Inertia Relief*
- *Description of Inertia Relief Using PARAM,INREL,-1*
- *Implementation of Inertia Relief Using PARAM,INREL,-1*
- *Automatic Inertia Relief*

18.1 Introduction to Inertia Relief

Inertia relief is an advanced option that allows you to simulate unconstrained structures in the linear static solution SOL 101, and in the linear static solution portions of SOLs 105 and 200. Typical applications of inertia relief include modeling an aircraft in flight, an automobile on a test track, or a satellite in space. The inertia relief method is a tool that you may not be familiar with. To help understand the method, the first section provides a brief description of inertia relief. The last section shows its implementation within NX Nastran.

Although the internal processing is somewhat complex, the interface to the inertia relief option is quite straightforward. There are two ways to invoke the inertia relief.

- The first method is to specify the SUPORT entry explicitly by including “PARAM,INREL,-1” in the Bulk Data Section.
- The second method (recommended) is to let NX Nastran select the SUPORT degrees of freedom automatically by including “PARAM,INREL,-2” in the Bulk Data Section.

An optional “PARAM,GRDPNT,x”— where x is a grid point ID— can be specified in the model. If “PARAM,GRDPNT,x” is used, the loads and accelerations will be summed about this point. If “PARAM,GRDPNT,x” is not specified, then the loads and accelerations will be summed about the origin of the basic coordinate system.

18.2 Description of Inertia Relief Using PARAM,INREL,-1

Static analysis by the finite element method assumes that the model contains no mechanisms and may not move as a rigid body (strain free). If either of these conditions exists in a conventional finite element analysis, the stiffness matrix for the model becomes singular. When NX Nastran attempts to decompose a singular matrix, a fatal message or unreasonable answers result.

Consequently, you can't perform conventional finite element static analysis on unconstrained structures. However, a method called inertia relief is provided in NX Nastran for analyzing these conditions. A simple description of inertia relief is that the inertia (mass) of the structure is used to resist the applied loadings, that is, an assumption is made that the structure is in a state of static equilibrium even though it is not constrained. Two examples are a spacecraft in orbit or an aircraft in flight. In these cases, the structure is in state of static equilibrium, although it is capable of unconstrained motion.

To invoke inertia relief, you must provide a SUPORT Bulk Data entry with a list of up to six non-redundant degrees of freedom that describe the possible unconstrained motion. The easiest way to describe how to use the SUPORT entry in static analysis is if you hold the SUPORT degrees of freedom constrained, there is no possible rigid body motion. If all possible rigid body motion is not described on the SUPORT entry, then the stiffness matrix is singular, and the problem either fails in decomposition or gives unreasonable answers.

When you specify inertia relief, NX Nastran calculates the forces that result from a rigid body acceleration about the point specified on the “PARAM,GRDPNT,x” in the specified directions. NX Nastran then calculates the summation of all applied loadings in the same directions. Accelerations are applied to the structure in the appropriate directions to “balance” the applied loadings. The structure is now in a state of static equilibrium, i.e., the summation of all applied loads is 0.0. Since the problem is not constrained, rigid body displacement is still possible.

NX Nastran next constrains the SUPORT degrees of freedom to a displacement of 0.0 and provide the relative motion of all other grid points with respect to that reference point. Hence, the term “reference” degree of freedom is used to describe the SUPORT degrees of freedom in NX Nastran. The set of degrees of freedom described on the SUPORT entry belong to the *r*-set or reference set for the solution. The computed solution is the correct one, and it is relative to any rigid body motion that is occurring. A simple way to think of this is that the solution coming from NX Nastran represents the deformation of the structure you would see if you were standing at the SUPORT degrees of freedom.

18.3 Implementation of Inertia Relief Using PARAM,INREL,-1

Solution 101 requires that the model does not contain mechanisms. You specify the reference degrees of freedom using the SUPORT entry as follows:

1	2	3	4	5	6	7	8	9	10
SUPPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Field

Contents

IDi Grid or scalar point identification number.

CI Component numbers.

In addition to the PARAM,INREL,-1 entry, a SUPORT entry is also needed. If PARAM,GRDPNT,x is specified, the rigid body mass matrix is calculated about grid point x; otherwise, the rigid body mass is calculated about the origin of the basic coordinate system. The inertia relief method requires that a realistic mass distribution exists, and the degrees of freedom listed on the SUPORT entry must be connected elastically to the model in all specified degrees of freedom. (For example, degrees of freedom 4, 5, and 6 on a grid point with only solid elements attached cannot be used since solid elements have no stiffness in these degrees of freedom.)

Whenever a SUPORT entry is used in static analysis, the epsilon and strain energy printed in the table from UIM 3035 should be checked. The values printed for epsilon and strain energy in the UIM 3035 table should all be numeric zero. The strain energy printed in this table for matrix KLR represents the strain energy in the model when the one SUPORT degree of freedom is moved 1.0 unit, while all other SUPORT degrees of freedom are constrained. If the SUPORT degrees of freedom are properly specified, the model should be capable of rigid body motion (strain-free) with no strain energy.

The values printed for the strain energy indicate the ability of the model to move as a rigid body. These values should always be checked. If the structure is not constrained, the values should be numeric zero, but roundoff almost always results in a small nonzero value. Acceptable values are a function of the units, size of the structure, and precision of the hardware; therefore, a recommended value is not provided in this user's guide.

An additional feature allowed in Solution 101 is the solution of a problem under uniform acceleration. This problem is posed using the DMIG,UACCEL entry in addition to the previously mentioned requirements. In this case, uniform accelerations are applied to the model, and the solution is found. Uniform accelerations are useful for situations, such as spacecraft liftoff and landing loadings, which are often specified as static accelerations.

As an example of inertia relief, consider the three CBEAM model shown in [Figure 18-1](#).

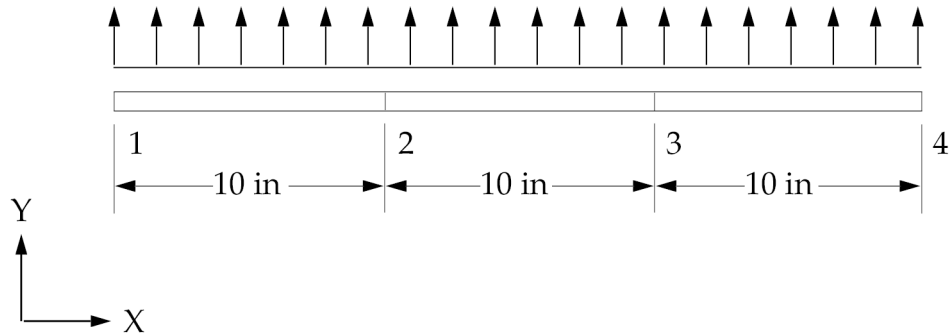


Figure 18-1. Inertia Relief Analysis of a CBEAM Model

The three CBEAM structure is to be analyzed as a free-free structure with a line load acting in the Y-direction as shown and using inertia relief. To show the effect of the SUPORT point, two runs are made, each with a different SUPORT grid point. The input file is given in [Listing 18-1](#). Note that the SUPORT information is shown for both runs; however, one set is commented out. For the first run, "inertia1.dat", the SUPORT point is located at grid point 1. All six degrees of freedoms for the SUPORT point are placed on one grid point as required for Solution 101. The parameter INREL is set to -1, and the parameter GRDPNT is set to the SUPORT point, which is grid point 1 in this case.


```

$ FILENAME - INERTIA1.DAT
ID      LINEAR,INERTIA
SOL      101
TIME     5
CEND
TITLE = BAR WITH SUPORT ENTRY
LOAD = 1
DISP = ALL
SPCF = ALL
STRESS = ALL
BEGIN BULK
PARAM    POST      0
$
$
$ SUPORT ENTRY FOR INERTIA1.DAT
$
SUPORT   1          123456
PARAM    GRDPNT    1
PARAM    INREL     -1
$
$ SUPORT ENTRY FOR INERTIA2.DAT
$
$SUPORT   3          123456
$PARAM    GRDPNT    3
$PARAM    INREL     -1
$
GRID     1          0.0      0.0      0.0
GRID     2          10.      0.0      0.0
GRID     3          20.      0.0      0.0
GRID     4          30.      0.0      0.0
$
CBEAM    1          1          1          2          1.          1.          0.0
CBEAM    2          1          2          3          1.          1.          0.0
CBEAM    3          1          3          4          1.          1.          0.0
$
.5
PBEAM    1          1          1.          .667          .167          .1
.5
.1          1.5          -.1          -.5          .1          -.5          -.1
PLOAD1   1          1          FY          FR          0.          1000.          1.          1000.
PLOAD1   1          2          FY          FR          0.          1000.          1.          1000.
PLOAD1   1          3          FY          FR          0.          1000.          1.          1000.
$
MAT1     1          1.+7          .3          7.43E-3
ENDDATA

```

Listing 18-1. Three CBEAM Model with Inertia Relief

A partial listing of the output is shown in [Figure 18-2](#).

```

1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 7
O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
0
                                REFERENCE POINT = 1
                                M O
      * 2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 *
      * 0.000000E+00  2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.343500E+00 *
      * 0.000000E+00  0.000000E+00  2.229000E-01  0.000000E+00 -3.343500E+00  0.000000E+00 *
      * 0.000000E+00  0.000000E+00  0.000000E+00  1.858986E-01  0.000000E+00  0.000000E+00 *
      * 0.000000E+00  0.000000E+00 -3.343500E+00  0.000000E+00  7.058500E+01  0.000000E+00 *
      * 0.000000E+00  3.343500E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.058500E+01 *
                                S
      * 1.000000E+00  0.000000E+00  0.000000E+00 *
      * 0.000000E+00  1.000000E+00  0.000000E+00 *
      * 0.000000E+00  0.000000E+00  1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
      X      2.229000E-01      0.000000E+00  0.000000E+00  0.000000E+00
      Y      2.229000E-01      1.500000E+01  0.000000E+00  0.000000E+00
      Z      2.229000E-01      1.500000E+01  0.000000E+00  0.000000E+00
                                I (S)
      * 1.858986E-01  0.000000E+00  0.000000E+00 *
      * 0.000000E+00  2.043250E+01  0.000000E+00 *
      * 0.000000E+00  0.000000E+00  2.043250E+01 *
                                I (Q)
      * 1.858986E-01
      * 2.043250E+01
      * 2.043250E+01
                                Q
      * 1.000000E+00  0.000000E+00  0.000000E+00 *
      * 0.000000E+00  1.000000E+00  0.000000E+00 *
      * 0.000000E+00  0.000000E+00  1.000000E+00 *

1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 9
0 RESULTANTS ABOUT      1 IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                OLOAD      RESULTANT
SUBCASE/      LOAD
DAREA ID      TYPE      T1      T2      T3      R1      R2      R3
0      1      FX      0.000000E+00      ----      ----      ----      0.000000E+00  0.000000E+00
      FY      ----      3.000000E+04      ----      0.000000E+00      ----      4.500000E+05
      FZ      ----      ----      0.000000E+00  0.000000E+00  0.000000E+00      ----
      MX      ----      ----      ----      0.000000E+00      ----      ----
      MY      ----      ----      ----      ----      0.000000E+00      ----
      MZ      ----      ----      ----      ----      ----      5.456968E-11
      TOTALS  0.000000E+00  3.000000E+04  0.000000E+00  0.000000E+00  0.000000E+00  4.500000E+05
0      2      FX      -2.229000E-01      ----      ----      ----      0.000000E+00  0.000000E+00
      FY      ----      0.000000E+00      ----      0.000000E+00      ----      0.000000E+00
      FZ      ----      ----      0.000000E+00  0.000000E+00  0.000000E+00      ----
      MX      ----      ----      ----      0.000000E+00      ----      ----
      MY      ----      ----      ----      ----      0.000000E+00      ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS -2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
0      3      FX      0.000000E+00      ----      ----      ----      0.000000E+00  0.000000E+00
      FY      ----      -2.229000E-01      ----      0.000000E+00      ----      -3.343500E+00
      FZ      ----      ----      0.000000E+00  0.000000E+00  0.000000E+00      ----
      MX      ----      ----      ----      0.000000E+00      ----      ----
      MY      ----      ----      ----      ----      0.000000E+00      ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS  0.000000E+00 -2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00 -3.343500E+00
0      4      FX      0.000000E+00      ----      ----      ----      0.000000E+00  0.000000E+00
      FY      ----      0.000000E+00      ----      0.000000E+00      ----      0.000000E+00
      FZ      ----      ----      -2.229000E-01  0.000000E+00  3.343500E+00      ----
      MX      ----      ----      ----      0.000000E+00      ----      ----
      MY      ----      ----      ----      ----      0.000000E+00      ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00

```

Figure 18-2. Inertia Relief Output When Grid Point 1 Is the SUPORT Point

```

TOTALS 0.000000E+00 0.000000E+00 -2.229000E-01 0.000000E+00 3.343500E+00 0.000000E+00
0 5 FX 0.000000E+00 -----
FY ----- 0.000000E+00 ----- 0.000000E+00 ----- 0.000000E+00
FZ ----- 0.000000E+00 0.000000E+00 0.000000E+00 -----
MX ----- -1.858986E-01 ----- 0.000000E+00 -----
MY ----- 0.000000E+00 ----- 0.000000E+00 -----
MZ ----- 0.000000E+00 ----- 0.000000E+00
TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00
0 6 FX 0.000000E+00 -----
FY ----- 0.000000E+00 ----- 0.000000E+00 ----- 0.000000E+00
FZ ----- 3.343500E+00 0.000000E+00 -7.058500E+01 -----
MX ----- 0.000000E+00 ----- 0.000000E+00 -----
MY ----- 0.000000E+00 ----- 0.000000E+00 -----
MZ ----- 0.000000E+00 ----- 0.000000E+00
TOTALS 0.000000E+00 0.000000E+00 3.343500E+00 0.000000E+00 -7.058500E+01 0.000000E+00
0 7 FX 0.000000E+00 -----
FY ----- -3.343500E+00 ----- 0.000000E+00 ----- -7.058500E+01
FZ ----- 0.000000E+00 0.000000E+00 0.000000E+00 -----
MX ----- 0.000000E+00 ----- 0.000000E+00 -----
MY ----- 0.000000E+00 ----- 0.000000E+00 -----
MZ ----- 0.000000E+00 ----- 0.000000E+00
TOTALS 0.000000E+00 -3.343500E+00 0.000000E+00 0.000000E+00 0.000000E+00 -7.058500E+01
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 10
0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
FOR DATA BLOCK KLR
SUPPORT PT.NO. EPSILON STRAIN ENERGY EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1 2.3344220E-15 0.0000000E+00
2 2.3344220E-15 7.2759576E-12
3 2.3344220E-15 0.0000000E+00
4 2.3344220E-15 0.0000000E+00
5 2.3344220E-15 0.0000000E+00
6 2.3344220E-15 5.3551048E-09
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 11
0
INTERMEDIATE MATRIX ... QRR
COLUMN 1
1 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 6
COLUMN 2
1 0.000000E+00 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 3.343500E+00 6
COLUMN 3
1 0.000000E+00 0.000000E+00 2.229000E-01 0.000000E+00 -3.343500E+00 0.000000E+00 6
COLUMN 4
1 0.000000E+00 0.000000E+00 0.000000E+00 1.858986E-01 0.000000E+00 0.000000E+00 6
COLUMN 5
1 0.000000E+00 0.000000E+00 -3.343500E+00 0.000000E+00 7.058500E+01 0.000000E+00 6
COLUMN 6
1 0.000000E+00 3.343500E+00 0.000000E+00 0.000000E+00 0.000000E+00 7.058500E+01 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 12
0
INTERMEDIATE MATRIX ... QRL
COLUMN 1
1 0.000000E+00 -3.000000E+04 0.000000E+00 0.000000E+00 0.000000E+00 -4.500000E+05 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 13

```

Figure 18-3. Inertia Relief Output When Grid Point 1 Is the SUPORT Point (Continued)

```

0
                                INTERMEDIATE MATRIX ... URA

                                COLUMN      1
1      1      0.000000E+00      1.345895E+05      0.000000E+00      0.000000E+00      8.246463E-
13      6

*** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1      -1.6152625E-14      1.5617192E+02

1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 15
0

                                D I S P L A C E M E N T   V E C T O R

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      1      G      0.0      0.0      0.0      0.0      0.0      0.0
2      2      G      0.0      -6.246877E-02      0.0      0.0      0.0      -1.249375E-02
3      3      G      0.0      -2.498751E-01      0.0      0.0      0.0      -2.498751E-02
4      4      G      0.0      -5.622189E-01      0.0      0.0      0.0      -3.748126E-02
1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 16
0

                                F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      1      G      0.0      -8.807738E-13      0.0      0.0      0.0      -6.462349E-27
1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 17
0
1  BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 18
0

                                S T R E S S E S   I N   B E A M   E L E M E N T S      ( C B E A M )

ELEMENT-ID  GRID  STAT DIST/  SXC      SXD      SXE      SXF      S-MAX      S-MIN      M.S.-
T  M.S.-C
0      1
      1      0.000      0.0      0.0      0.0      0.0      0.0      0.0
      2      1.000      0.0      0.0      0.0      0.0      0.0      0.0
0      2
      2      0.000      0.0      0.0      0.0      0.0      0.0      0.0
      3      1.000      0.0      0.0      0.0      0.0      0.0      0.0
0      3
      3      0.000      0.0      0.0      0.0      0.0      0.0      0.0
      4      1.000      0.0      0.0      0.0      0.0      0.0      0.0

```

Figure 18-3. Inertia Relief Output When Grid Point 1 Is the SUPORT Point

The GPWG (Grid Point Weight Generator) output is calculated with respect to the grid point specified on the parameter GRDPNT. The total mass of the structure in this model is 0.229, which is used by NX Nastran to develop the inertia loads. Inspection of the GPWG output should be part of your routine model checkout.

The OLOAD RESULTANT contains seven sections—consisting of seven lines per section. The first section is the resultant of the applied loads about the GRDPNT, which is the basic coordinate system in this case. The remaining six sections are the loads necessary to impose unit acceleration about the SUPORT point. Subcase 2 corresponds to the X-direction; Subcase 3 corresponds to Y-direction,

etc. The first six lines of each section represent the detailed contributions, whereas the seventh line represents the total.

Following the OLOAD output is User Information Message 3035 showing the strain energy and epsilon due to the imposed unit accelerations about the SUPORT point. As discussed earlier, the epsilons and strain energies should be small, which is the case for this example. Small epsilons and strain energy tell you that you do not have any unwanted constraints or poorly defined MPCs causing a constraint in your model. Always inspect User Information Message 3035 to make sure that the epsilons and strain energies are small.

The intermediate matrix QRR is printed following the UIM 3035. The QRR matrix is the total rigid body mass of the total structure. It is a 6 x 6 matrix measured about the PARAM,GRDPNT point in the global coordinate system. Masses on the scalar points are not included. Following the QRR matrix is the QRL matrix. The QRL matrix is the resultant of the “apparent reaction loads,” measured at the SUPORT point. This resultant is equal and opposite to the OLOAD resultant shown earlier in the output, if the SUPORT point is the same as the PARAM,GRDPNT point. There is one column for each loading condition. The last matrix output is the URA matrix, which is the rigid body acceleration matrix that is computed from the applied loads.

The displacement and stress output shown is the standard output as requested through the Case Control Section. Note that the displacement at the SUPORT point is exactly 0.0. This SUPORT point should be 0.0 since this point is constrained for the solution, and the forces of constraint should be numeric zero. The SPC forces are shown in the output to confirm that the loads are balanced at the SUPORT point. The displacement of all of the other points in the model are relative to the SUPORT point.

The displacement solution calculated for the model is not a true deflected shape, but rather a solution of the inertia relief calculations that balance the acceleration of the center of gravity with the applied loads. In the “Automatic Inertia Relief” section, the model is run using the automatic inertia relief method by specifying PARAM,INREL,-2 instead of PARAM,INREL,-1 and removing the SUPORT entry. The results shown in [Figure 18-6](#) indicate that the “balance” displacement solution is in the direction of the applied load and is centered around the center of gravity of the model.

For the PARAM,INREL,-1 method, the “balance” solution is fixed at Grid 1, which is the grid point specified on the SUPORT entry. To satisfy this constraint, the balance shape is rotated clockwise. Because the model is free-free and the applied load is a constant pressure load along the entire beam, the true deflected shape is a rigid body motion in the direction of the applied load. Such a solution produces no bending and generates zero stress, which are confirmed by the results.

An interesting effect occurs when the SUPORT point is changed in the previous example. Suppose you change the location of the SUPORT point from 1 to 3 (see the commented SUPORT entries in [Listing 18-1](#)).

The resulting displacement and stress output is shown in [Figure 18-4](#).

```

1 BAR WITH SUPORT ENTRY                                OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 7

      OUTPUT FROM GRID POINT WEIGHT GENERATOR
0      REFERENCE POINT = 3
      M O
      * 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 -1.114500E+00 *
      * 0.000000E+00 0.000000E+00 2.229000E-01 0.000000E+00 1.114500E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 0.000000E+00 1.858986E-01 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.114500E+00 0.000000E+00 2.600500E+01 0.000000E+00 *
      * 0.000000E+00 -1.114500E+00 0.000000E+00 0.000000E+00 0.000000E+00 2.600500E+01 *
      S
      * 1.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 1.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.000000E+00 *
      DIRECTION
      MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
      X      2.229000E-01      0.000000E+00 0.000000E+00 0.000000E+00
      Y      2.229000E-01      -5.000000E+00 0.000000E+00 0.000000E+00
      Z      2.229000E-01      -5.000000E+00 0.000000E+00 0.000000E+00
      I (S)
      * 1.858986E-01 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 2.043250E+01 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 2.043250E+01 *
      I (Q)
      * 1.858986E-01 *
      * 2.043250E+01 *
      * 2.043250E+01 *
      Q
      * 1.000000E+00 0.000000E+00 0.000000E+00 *
      * 0.000000E+00 1.000000E+00 0.000000E+00 *
      * 0.000000E+00 0.000000E+00 1.000000E+00 *

0      CLOAD      RESULTANT
      SUBCASE/      LOAD
      DAREA ID      TYPE      T1      T2      T3      R1      R2      R3
0      1      FX      0.000000E+00      ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      3.000000E+04      ----      0.000000E+00 ----      -1.500000E+05
      FZ      ----      ----      0.000000E+00 0.000000E+00 ----      ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      5.456968E-11
      TOTALS 0.000000E+00 3.000000E+04 0.000000E+00 0.000000E+00 0.000000E+00 -1.500000E+05
0      2      FX      -2.229000E-01 ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00 ----      0.000000E+00 ----      0.000000E+00
      FZ      ----      ----      0.000000E+00 0.000000E+00 ----      ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS -2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
0      3      FX      0.000000E+00 ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      -2.229000E-01 ----      0.000000E+00 ----      1.114500E+00
      FZ      ----      ----      0.000000E+00 0.000000E+00 ----      ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS 0.000000E+00 -2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 1.114500E+00
0      4      FX      0.000000E+00 ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00 ----      0.000000E+00 ----      0.000000E+00
      FZ      ----      ----      -2.229000E-01 0.000000E+00 -1.114500E+00 ----
      MX      ----      ----      ----      0.000000E+00 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00
      TOTALS 0.000000E+00 0.000000E+00 -2.229000E-01 0.000000E+00 -1.114500E+00 0.000000E+00
0      5      FX      0.000000E+00 ----      ----      ----      0.000000E+00 0.000000E+00
      FY      ----      0.000000E+00 ----      0.000000E+00 ----      0.000000E+00
      FZ      ----      ----      0.000000E+00 0.000000E+00 0.000000E+00 ----
      MX      ----      ----      ----      -1.858986E-01 ----      ----
      MY      ----      ----      ----      ----      0.000000E+00 ----
      MZ      ----      ----      ----      ----      ----      0.000000E+00

```

Figure 18-4. Inertia Relief Output When Grid Point 3 Is the SUPORT Point (Continued)

```

TOTALS 0.000000E+00 0.000000E+00 0.000000E+00 -1.858986E-01 0.000000E+00 0.000000E+00
0 6 FX 0.000000E+00 ---- ---- 0.000000E+00 0.000000E+00 0.000000E+00
FY ---- 0.000000E+00 ---- 0.000000E+00 ---- 0.000000E+00
FZ ---- -1.114500E+00 0.000000E+00 -2.600500E+01 ----
MX ---- ---- 0.000000E+00 ---- ----
MY ---- ---- 0.000000E+00 ---- ----
MZ ---- ---- 0.000000E+00 ----
TOTALS 0.000000E+00 0.000000E+00 -1.114500E+00 0.000000E+00 -2.600500E+01 0.000000E+00
0 7 FX 0.000000E+00 ---- ---- 0.000000E+00 0.000000E+00 0.000000E+00
FY ---- 1.114500E+00 ---- 0.000000E+00 ---- -2.600500E+01
FZ ---- 0.000000E+00 0.000000E+00 0.000000E+00 ----
MX ---- ---- 0.000000E+00 ---- ----
MY ---- ---- 0.000000E+00 ---- ----
MZ ---- ---- 0.000000E+00 ----
TOTALS 0.000000E+00 1.114500E+00 0.000000E+00 0.000000E+00 0.000000E+00 -2.600500E+01
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 10

0
*** SYSTEM INFORMATION MESSAGE 6916 (DFMSYN)
DECOMP ORDERING METHOD CHOSEN: BEND, ORDERING METHOD USED: BEND
*** USER INFORMATION MESSAGE 3035 (SOLVER)
FOR DATA BLOCK KLR
SUPPORT PT.NO. EPSILON STRAIN ENERGY EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1 8.8574680E-16 0.0000000E+00
2 8.8574680E-16 1.4551915E-11
3 8.8574680E-16 0.0000000E+00
4 8.8574680E-16 0.0000000E+00
5 8.8574680E-16 1.1641532E-10
6 8.8574680E-16 3.2596290E-09
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 11

0
INTERMEDIATE MATRIX ... QRR
COLUMN 1
1 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 6
COLUMN 2
1 0.000000E+00 2.229000E-01 0.000000E+00 0.000000E+00 0.000000E+00 -1.114500E+00 6
COLUMN 3
1 0.000000E+00 0.000000E+00 2.229000E-01 0.000000E+00 1.114500E+00 0.000000E+00 6
COLUMN 4
1 0.000000E+00 0.000000E+00 0.000000E+00 1.858986E-01 0.000000E+00 0.000000E+00 6
COLUMN 5
1 0.000000E+00 0.000000E+00 1.114500E+00 0.000000E+00 2.600500E+01 0.000000E+00 6
COLUMN 6
1 0.000000E+00 -1.114500E+00 0.000000E+00 0.000000E+00 0.000000E+00 2.600500E+01 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 12

0
INTERMEDIATE MATRIX ... QRL
COLUMN 1
1 0.000000E+00 -3.000000E+04 0.000000E+00 0.000000E+00 0.000000E+00 1.500000E+05 6
1 BAR WITH SUPORT ENTRY OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 13

0
INTERMEDIATE MATRIX ... URA
COLUMN 1
1 0.000000E+00 1.345895E+05 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 6
*** USER INFORMATION MESSAGE 5293 (SSG3A)
FOR DATA BLOCK KLL
LOAD SEQ. NO. EPSILON EXTERNAL WORK EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1 -2.6191993E-15 1.5617192E+02

```

Figure 18-4. Inertia Relief Output When Grid Point 3 Is the SUPORT Point (Continued)

1	BAR WITH SUPORT ENTRY				OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 14					
D I S P L A C E M E N T V E C T O R										
POINT ID.		TYPE	T1	T2	T3	R1	R2	R3		
1		G	0.0	-2.498751E-01	0.0	0.0	0.0	2.498751E-02		
2		G	0.0	-6.246877E-02	0.0	0.0	0.0	1.249375E-02		
3		G	0.0	0.0	0.0	0.0	0.0	0.0		
4		G	0.0	-6.246877E-02	0.0	0.0	0.0	-1.249375E-02		
1	BAR WITH SUPORT ENTRY				OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 16					
0										
1	BAR WITH SUPORT ENTRY				OCTOBER 16, 2004 NX NASTRAN 4/ 9/04 PAGE 17					
0										
S T R E S S E S I N B E A M E L E M E N T S (C B E A M)										
ELEMENT-ID		GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T M.S.-C
0	1									
		1	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		2	1.000	0.0	0.0	0.0	0.0	0.0	0.0	
0	2									
		2	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		3	1.000	0.0	0.0	0.0	0.0	0.0	0.0	
0	3									
		3	0.000	0.0	0.0	0.0	0.0	0.0	0.0	
		4	1.000	0.0	0.0	0.0	0.0	0.0	0.0	

Figure 18-4. Inertia Relief Output When Grid Point 3 Is the SUPORT Point

Note that the displacement vector changes as the displacements are now relative to grid point 3, which is the new SUPORT point. However, the stresses in the elements are the same because the stresses are based on the relative displacement between the grid points, which is independent of the grid point used for the SUPORT point.

A final comment on this simple beam model: suppose you used CBAR elements instead of CBEAM elements for this model. The run would fail because the CBAR element does not have any torsional inertia mass, whereas the CBEAM element does. This model is basically a one-dimensional structure so that the rotational inertia about the X-axis is the mass inertia of the elements. When the inertia relief method is used, the structure must have mass in all six directions. In general, however, there are no restrictions on using a CBAR element with inertia relief. Most structures include CBARs that are not co-linear. The example above was introduced merely to emphasize the need to have mass in all six degrees of freedom.

Suppose you wish to impose a 10 g acceleration at grid point 2 of the beam structure shown in [Figure 18-1](#). This acceleration can be applied using the DMIG,UACCEL Bulk Data entry as shown below.

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	"0"	"9"	TIN					
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

Field	Contents
TIN	Type of matrix being input.
L	Load sequence number.
Gi	Grid point identification number of a single reference point.
Ci	Component number for Gi in the basic coordinate system.
Xi	Value of enforced acceleration term in the basic coordinate system.

The input file for this example is shown in [Listing 18-2](#).

```

$ FILENAME - UACCEL
ID      LINEAR,UACCEL
SOL     101
TIME    5
CEND
TITLE = BAR WITH UACCEL INPUT
OLOAD = ALL
DISP = ALL
BEGIN BULK
PARAM   POST      0
$
$
$ SUPORT INFORMATION
$
SUPORT  2          123456
PARAM   GRDPNT    2
PARAM   INREL     -1
DMIG    UACCEL    0      9      1
DMIG    UACCEL    1              2      2      3864.0
$
GRID    1          0.0      0.0      0.0
GRID    2          10.      0.0      0.0
GRID    3          20.      0.0      0.0
GRID    4          30.      0.0      0.0
$
CBEAM   1          1      1      2      1.      1.      0.0
CBEAM   2          1      2      3      1.      1.      0.0
CBEAM   3          1      3      4      1.      1.      0.0
$
PBEAM   1          1      1.      .667      .167      .1
      .1      .1      .1      -.1      -.1      .1      -.1      -.1
$
MAT1    1          1.+7      .3      7.43E-3
ENDDATA

```

Listing 18-2. Imposing a 10 g Acceleration Using the PARAM,UACCEL

The acceleration of 10 g is entered on the DMIG,UACCEL entry. When a DMIG (Direct Matrix Input) entry is used, UACCEL, the first entry is the header entry, which is denoted by the 0 in field 3. Field 4 must be a 9, and field 5 is either a 1 for single precision input (the most common) or a 2 for double precision input. For this example, the input is in single precision.

The second entry is where the applied acceleration is entered. Field 3 must be a 1, indicating that this is the first load case (and the only load case in this model). The grid point component and value of the enforced acceleration are entered in fields 6, 7, and 8. If acceleration is to be enforced in more than one component, then the continuation entries are used. For this example, the acceleration of 3864. in/sec² is entered in field 8. The 3864.0 is 10 g in the English system-you must ensure that the units are consistent.

When using the DMIG,UACCEL, the intermediate matrix URACCEL is output as shown in [Figure 18-5](#). This matrix is the rigid body acceleration that you input using the DMIG,UACCEL entry. Also shown is the OLOAD output and the displacement vector. As can be seen, the SUPORT point is constrained, which is what you would expect, and the total load applied to the structure corresponds to the mass of the structure (see the GPWG output in [Figure 18-5](#)) times the enforced acceleration.

Although not commonly done, you can perform multiple inertia relief analyses in a single run. This feature can be activated with the SUPORT1 entries instead of the SUPORT entry. Unlike the SUPORT entry, which is automatically activated, the SUPORT1 entry must be called out by the SUPORT1 Case Control command in order for it to be applied.

```

O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
                                REFERENCE POINT =      2
                                M O
* 2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  2.229000E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.114500E+00 *
* 0.000000E+00  0.000000E+00  2.229000E-01  0.000000E+00 -1.114500E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  0.000000E+00  1.858986E-01  0.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00 -1.114500E+00  0.000000E+00  2.600500E+01  0.000000E+00 *
* 0.000000E+00  1.114500E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.600500E+01 *
                                S
* 1.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  1.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
X      2.229000E-01      0.000000E+00  0.000000E+00  0.000000E+00
Y      2.229000E-01      5.000000E+00  0.000000E+00  0.000000E+00
Z      2.229000E-01      5.000000E+00  0.000000E+00  0.000000E+00
                                I (S)
* 1.858986E-01  0.000000E+00  0.000000E+00 *
* 0.000000E+00  2.043250E+01  0.000000E+00 *
* 0.000000E+00  0.000000E+00  2.043250E+01 *
                                I (Q)
* 1.858986E-01
* 2.043250E+01
* 2.043250E+01
                                Q
* 1.000000E+00  0.000000E+00  0.000000E+00 *
* 0.000000E+00  1.000000E+00  0.000000E+00 *
* 0.000000E+00  0.000000E+00  1.000000E+00 *
INTERMEDIATE MATRIX ... URA
COLUMN      1
1      0.000000E+00      3.864000E+03      0.000000E+00      0.000000E+00      0.000000E+00      0.000000E+00      6
DISPLACEMENT VECTOR
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      G      .0      -7.547017E-03      .0      .0      .0      1.076069E-03
2      G      .0      .0      .0      .0      .0      .0
3      G      .0      -3.340174E-02      .0      .0      .0      -5.380345E-03
4      G      .0      -9.475221E-02      .0      .0      .0      -6.456414E-03
LOAD VECTOR
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1      G      .0      -1.435476E+02      .0      .0      .0      .0
2      G      .0      -2.870952E+02      .0      .0      .0      .0
3      G      .0      -2.870952E+02      .0      .0      .0      .0
4      G      .0      -1.435476E+02      .0      .0      .0      .0

```

Figure 18-5. Partial Output of the UACCEL Example

18.4 Automatic Inertia Relief

In the previous sections, manual inertia relief is discussed where the SUPORT entry must be specified explicitly along with PARAM,INREL,-1. The SUPORT entry lists the DOFs in the r-set, the reference set. Some skill is required to use this entry correctly. When the r-set is chosen such that structure is not constrained in a statically determinate manner, or if the stiffness attached to these points is inadequate, the rigid body mode shapes may be poor, leading to low accuracy in inertial load calculations. A poor static analysis solution may also result from the poor constraints, independent of the loading problems. Both effects may result in poor overall accuracy. Poor results can occur with little warning when the SUPORT entry is misused.

The automatic inertia relief method is the recommended inertia relief method. With the automatic method, the specification of the SUPORT entry is no longer needed. To turn it on, simply add PARAM,INREL,-2 to the input file. The reference frame is selected automatically, in a manner that poor solutions are unlikely because of the choice of reference frame variables.

The constraints associated with the reference frame are distributed to all points with mass. This means that structures with modeling errors, such as a region of elements left out through oversight, will still give reasonable results that can aid in diagnosing the modeling errors. A model in development may contain many disjoint parts, inadvertently. Any part with three non co-linear points with mass will be adequately constrained for solution.

In the manual support option, the free stiffness matrix K_{aa} is constrained by removing the r-set DOFs from the a-set and imposing zero motion on them,

$$[K_{aa}] \cdot [u_a] = \{P_a\}$$

Equation 18-1.

$$[u_r] = \{0\}$$

Equation 18-2.

With the automatic inertia relief, the u_r variables are no longer a subset of the u_a variables. The constraint equation (Eq. 18-2) is replaced by a more general constraint equation,

$$[M_{aa} \cdot D_{a6}]^T \cdot \{u_a\} = \{u_r\} = \{0\}$$

Equation 18-3.

The variables u_r are a set of generalized coordinates that represent the average motion of the u_a variables, as weighted by the mass matrix. The zero value for this equation implies that the average motion of all a-set points is zero, although the displacement variables are free to move relative to this least-squares fit reference frame.

The manual style of inertia relief allows use of PARAM,GRDPNT to define the origin used in determining the shape functions D_{a6} . The presence of this parameter causes the output of the Grid Point Weight Generator (GPWG) table, which lists the c.g. location of the model, its moments of inertia, and other related data. This point in space is used as the reference point when computing rigid body mass and moments of inertia, and rigid body accelerations. If the parameter is not present, the basic origin is used, in the basic coordinate system. It is good practice (but not required by NX Nastran) to use the same grid point on PARAM,GRDPNT and on the SUPORT entry.

Unlike the manual option, PARAM,GRDPNT is not used as the inertia relief reference point for the automatic support option. The basic origin is used. If the PARAM,GRDPNT is present, it causes the generation of the GPWG table, but does not influence the inertia relief calculations in any way.

The constraint forces caused by the distributed constraints are printed with the SPCFORCE case control command. They are computational zeros for a well-defined model. PARAM,TINY (default value is 1.E-3) is traditionally used to discard element strain energies with values less than “tiny”. The

same parameter requests that small SPC forces of the reference frame constraint type be replaced with binary zeros, for the automatic support option only. This filtering is not used on conventional SPC forces from selected SPC entries or AUTOSPC, and manual support DOFs.

Limitations for the Automatic Inertia Relief Method

Following are limitations for the Automatic Inertia Relief Method:

- The DMIG,UACCEL option is not supported.
- The Iterative Solver is not supported.
- There must be 6 rigid body modes (i.e. free-free).

EXAMPLE

Let's revisit the CBEAM model in [Figure 18-1](#) using the automatic inertia relief method. This is accomplished by removing the SUPORT entry and replacing PARAM,INREL,-1 with PARAM,INREL,-2. [Figure 18-6](#) summarizes the displacements, spcforces, and element stresses output.

The spcforces and stresses are both zeros, similar to the other two runs. As for the displacements, the mass-weighted average motion of all a-set points must be zero. Since the mass is evenly distributed, u_r can be calculated for the T2 components as follows:

$$U_r = \{0.5 \cdot (-8.329169\text{E-}2) + 1.0 \cdot (4.164584\text{E-}2) + 1.0 \cdot (4.164584\text{E-}2) + 0.5 \cdot (-8.329169\text{E-}2)\} = 0$$

Equation 18-4.

See TAN 4002 for further details regarding the Automatic Inertia Relief Method.

```

0
      DISPLACEMENT VECTOR
      POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
      1      G      0.0      -8.329169E-02  0.0      0.0      0.0      1.874063E-02
      2      G      0.0      4.164584E-02  0.0      0.0      0.0      6.246876E-03
      3      G      0.0      4.164584E-02  0.0      0.0      0.0      -6.246876E-03
      4      G      0.0      -8.329169E-02  0.0      0.0      0.0      -1.874063E-02
1  BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 17
0
      FORCES OF SINGLE-POINT CONSTRAINT
      POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
      1      G      0.0      6.251190E-13  0.0      0.0      0.0      0.0
      2      G      0.0      8.820207E-13  0.0      0.0      0.0      0.0
      3      G      0.0      5.138035E-13  0.0      0.0      0.0      0.0
      4      G      0.0      7.279315E-14  0.0      0.0      0.0      0.0
1  BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 18
0
1  BAR USING AUTOMATIC SUPORT ENTRY - GRDENT AT 1      AUGUST 19, 2001 MSC.NASTRAN 4/ 9/01 PAGE 19
0
      STRESSES IN BEAM ELEMENTS (CBEAM)
      ELEMENT-ID  GRID  STAT DIST/  SXC  SXD  SXE  SXF  S-MAX  S-MIN  M.S.-T  M.S.-C
      1          1      LENGTH
0      1          1      0.000  0.0    0.0    0.0    0.0    0.0    0.0
      2          2      1.000  0.0    0.0    0.0    0.0    0.0    0.0
0      2          2      0.000  0.0    0.0    0.0    0.0    0.0    0.0
      3          3      1.000  0.0    0.0    0.0    0.0    0.0    0.0
0      3          3      0.000  0.0    0.0    0.0    0.0    0.0    0.0
      4          4      1.000  0.0    0.0    0.0    0.0    0.0    0.0

```

Figure 18-6. Abridged Output Using the Automatic Inertia Relief Method

Chapter 19: Contact

NX Nastran provides a contact capability for SOL 101 linear static analysis, and also in consecutive SOLs 103, 105, 111 and 112. NX Nastran also provides a contact capability for SOL 401 multi-step nonlinear analysis. Most of the contact inputs used by SOL 101 are supported by SOL 401. Contact for the SOLs 601 and 701 is also available and is discussed in the *Advanced Nonlinear Theory and Modeling Guide*.

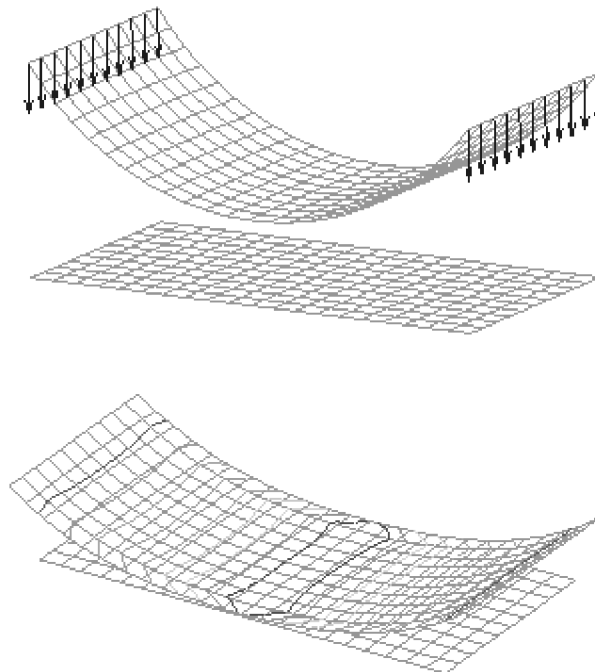
Surface-to-Surface Contact Summary

Surface-to-surface contact can be defined on the faces of the following elements.

- Shell elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR.
- 3-D Solid elements CHEXA, CPENTA, CPYRAM, and CTETRA.

Surface-to-surface contact conditions allow the solution to search and detect when a pair of element faces come into contact. The contact conditions prevent the faces from penetrating and allow finite sliding with optional friction effects. The surface-to-surface contact source and target regions consist of shell and/or solid element faces. From element faces in the source region, a top and bottom normal is projected. The software creates a contact element if:

- Any of the source element normals intersect with an element in the target region.
- The distance between the two faces is equal to or less than the defined separation distance.



Edge-to-Edge Contact Summary

Edge-to-edge contact can be defined on the edges of the following elements:

- Axisymmetric elements CTRAX3, CQUADX4, CTRAX6, CQUADX8.
- Plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8.
- Plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.

Edge-to-edge contact conditions allow the solution to search and detect when a pair of element edges come into contact. The software creates contact elements if the distance between the edges is equal to or less than the search distance which you specify. The contact elements prevent the edges from penetrating and allow finite sliding with optional friction effects.

The axisymmetric, plane stress, and plane strain elements can be defined in either the XZ plane or in the XY plane. Edge-to-edge contact is supported in either orientation.

19.1 Contact Regions

Edge Contact Regions

An edge contact region is a collection of axisymmetric, plane strain, and plane stress element free edges in a section of the model where you expect contact to occur. These regions can be created using either the BEDGE bulk entry or the BLSEG bulk entry.

- To define a contact edge using the BEDGE entry, you enter the element ID along with the corner GRID IDs. You can define the edges on the BEDGE entry in any order.
- To define a contact edge using the BLSEG entry, you enter the GRID IDs in a continuous topological order. You must include both midside and corner GRID IDs on the BLSEG entry.

All region IDs defined with the BEDGE, BLSEG, BSURF, BSURFS, BCPROP, or BCPROPS entries must be unique.

Surface Contact Regions

A surface contact region is a collection of element free faces in a section of the model where you expect contact to occur. These regions can be created using the shell elements (BSURF and BCPROP) and using solid element free faces (BSURFS and BCPROPS).

- The BSURF entry is defined by its own unique ID and is a list or range of shell element IDs to include in the region.
- The BCPROP entry is defined by its own unique ID and is a list of shell element property IDs. Shell elements which use any of these listed property IDs will be included in the region.
- The BSURFS entry is defined by its own unique ID and is a list of solid element IDs each followed by 3 grid points defining which face of the 3-D element to include in the contact region.
- The BCPROPS entry is defined by its own unique ID and is a list of solid element property IDs. The free faces of the solid elements selected with a property ID are automatically determined by the software.

A contact surface can be defined as any type of shell or any face of a solid element. Although parabolic faces with omitted midside nodes are permitted, their use could affect accuracy.

All region IDs defined with the BEDGE, BLSEG, BSURF, BSURFS, BCPROP, and BCPROPS entries must be unique.

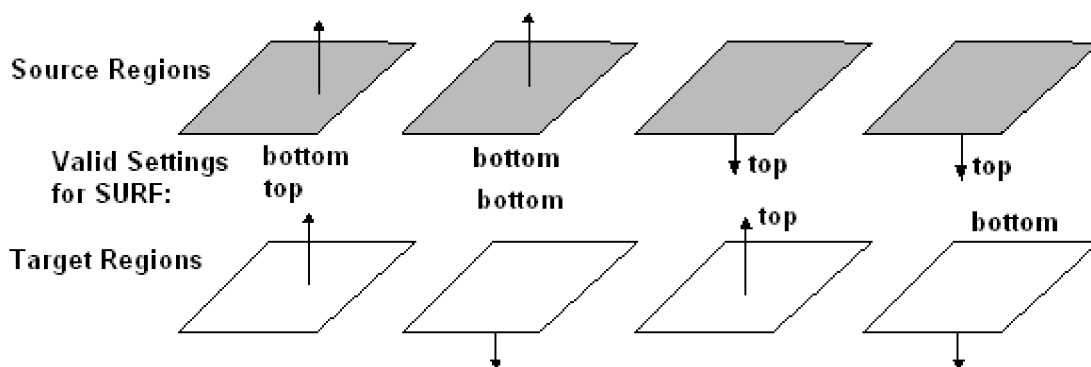
19.2 Contact Region Parameters

Contact region parameters are defined using the BCRPARA bulk entry. The CRID field on BCRPARA must match the ID used on one of the BSURF, BCPROP, BSURFS, BCPROPS, BEDGE, and BLSEG bulk entries to be considered by the solution. The parameters SURF and OFFSET are supported with a surface-to-surface contact definition. Only OFFSET is supported with an edge-to-edge contact definition.

- SURF is used to define the contact side of shell element regions. When SURF is "TOP", the contact side is consistent with the shell element normal and when "BOT" the opposite. In a SOL 101 (including consecutive solutions 103, 111 and 112), SURF must be defined so that source and target contact sides either face one another to represent a separation condition, or oppose one another to represent an interference condition.

Understanding shell element normals, and making sure those in the same regions are consistent is very important to ensuring that contact elements will be created as expected. See [“Using Consistent Normals”](#) in the *NX Nastran User's Guide* for information on using consistent shell normals.

The figure below shows examples of how the SURF option is used to modify the contact side when the element normals vary. The arrow on each face shows the element normal. Source regions are shaded while target regions are white.



- Use the OFFSET field to account for a rigid layer between contact face or edge regions. For example, a model which has two metal surfaces coming into contact, and one of these has a ceramic coating. If the ceramic material stiffness is not significant enough to be included in the analysis, it may not have been specifically modeled, but the thickness it adds to the face of the metal may be important when considering the contact problem.

You can also use the OFFSET field to analyze an interference fit problem if unconnected elements are modeled coincident. The offset value in this example can represent the theoretical interference.

19.3 Contact Pairs

A contact pair combines two contact regions, source and target, in which contact will be analyzed during the solution. Each contact pair can have its own unique friction value (if desired) and search distance.

The BCTSET bulk entry is used to define both edge-to-edge and surface-to-surface contact pairs. The CID field will need to match the value of 'n' on the BCSET case control entry for the solution to recognize this contact definition. The SIDi and TIDi fields are used to define source and target regions respectively for a pair. As many pairs as desired can be included on a single BCTSET entry. Each pair can have a unique friction value (optional), a minimum search distance, and a maximum search distance. You can define the optional Coefficient of Friction field (FRICi) for each contact pair if you expect finite sliding to occur. When contact is detected, the solver uses this value to calculate any tangential contact forces by multiplying the normal contact force by FRICi.

The regions you select with SID and TID depend on the type of contact:

- For edge-to-edge contact pairs, SID and TID are contact regions defined with the BEDGE and BLSEG entries.
- For surface-to-surface contact pairs, SID and TID are contact regions defined with the BCPROP, BCPROPS, BSURF, and BSURFS entries.

The minimum and maximum search distance fields (MINDi and MAXDi) define a range in which the solver can initially determine if the distance between element edges or faces in a particular pair are within the threshold for creating contact elements. These values are used once, at the beginning of SOL 101, to determine where contact elements need to be initially created. The minimum distance can be negative if there is an interference condition modeled as overlapping regions.

Since SOL 101 assumes small deflection, the number of contact elements created in the initial step will remain the same for the entire solution. Although, each contact element status can change during the solution between active and inactive as the contact condition iterates.

Combining Contact Sets – BCTADD

You can optionally define multiple BCTSET/BCTPARAM bulk entry sets, each set with unique contact set IDs (CSID), and then combine them with a single BCTADD bulk entry. The multiple BCTSET/BCTPARAM bulk entry sets are created to adjust certain contact parameters locally. Contact parameters can also be adjusted globally with a BCTPARAM bulk entry having the same CSID as the BCSET case control command.

The following example demonstrates the inputs.

```
CASE CONTROL
$CSID on the BCSET case control matches CSID on BCTADD
BCSET = 108
...
BULK DATA
$Local Contact Set definitions
BCTSET 1 1 2 0.0 1.0
BCTSET 2 3 4 0.15 0.0 0.1
5 6 0.15 0.0 0.1
...
$Local Contact Parameters
BCTPARAM 1 PENN 10 PENT 1
BCTPARAM 2 PENN 1.0 PENT 0.1
...
$Local Contact Sets are combined with BCTADD
BCTADD 108 1 2
```

```

...
$Global Contact Parameters
BCTPARM 108 MAXS 30 NCHG 0.02

```

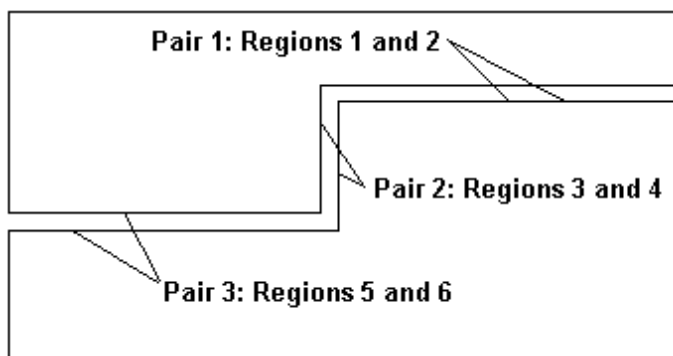
See the section “Contact Control Parameters - BCTPARM” in this chapter for more information on contact parameters.

Contact with Composite Solid Faces

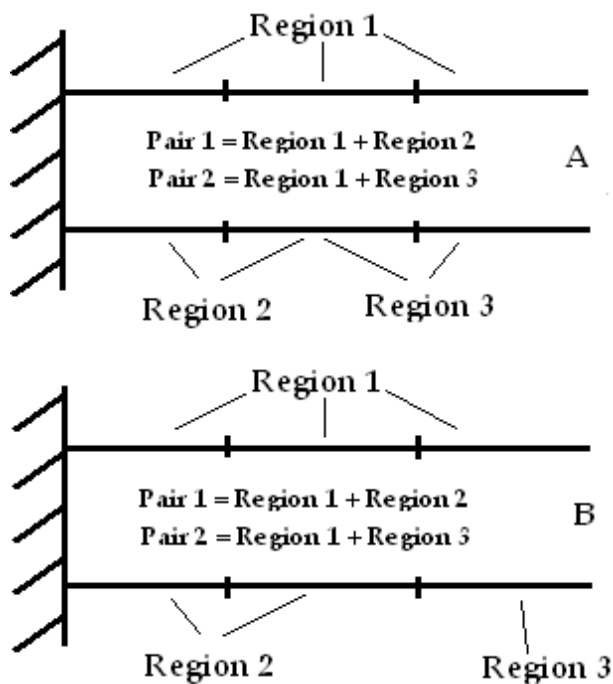
Defining contact regions and pairs on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the contact definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BCREULTS case control command.

Additional Recommendations

When defining contact regions and pairs on geometry which is not tangent continuous, creating single contact regions which cross corner transitions can result in non-uniform stress results around the corners. It is recommended to break these areas into multiple regions and pairs as shown below.



When defining contact regions and pairs, it is recommended to not include the same element face in multiple regions. In “A” below, an element is repeated in regions 2 and 3. In “B”, the same element only exists in region 2. “B” is recommended. Repeating element faces multiple times in the same or different regions can significantly increase memory requirements and degrade performance.



19.4 Contact Control Parameters

The contact control parameters on the BCTPARM bulk entry (optional) can help you adjust the contact algorithm when you are having problems getting a solution to converge and complete, or when the contact results are not as expected. Also see the section [Penalty Factor Adjustment and Contact Stiffness](#) below. For many solutions, the default settings are appropriate, and the BCTPARM entry is not required.

- For surface-to-surface contact definitions, all of the parameters on the BCTPARM entry are supported.
- For edge-to-edge contact definitions, the parameters CTOL, MAXF, MAXS, NCHG, INIPENE, PENN, PENT, PENTYP, RESET, CSTRAT, and PREVIEW are supported. The parameters REFINE, SHLTHK, ZOFFSET, and INTORD are ignored.

You can optionally define multiple BCTSET/BCTPARM bulk entry sets, each set with unique contact set IDs (CSID), and then combine them with a single BCTADD bulk entry. The multiple BCTSET/BCTPARM bulk entry sets are created to adjust certain contact parameters locally. Contact parameters can also be adjusted globally with a BCTPARM bulk entry having the same CSID as the BCSET case control command. Also see the section [Contact Pairs](#) which includes an input example.

Global and local contact parameters have the following definition and rules:

- Global Contact Parameters

The BCTPARM bulk entry, which uses the same CSID entered on the BCSET case control command, defines global parameters.

Any of the parameters on the BCTPARM bulk entry can be defined globally. A parameter's default value is used if it is not defined globally or locally.

- Local Contact Parameters

The BCTPARM bulk entries associated to individual BCTSET bulk entries, which are then combined with a BCTADD bulk entry, define local parameters.

Following is a description of the BCTPARM input parameters. Local contact parameters have “*” next to their names.

- PENN*: Penalty factor for normal direction. PENN and PENT are automatically calculated by default. When PENT is defined but PENN is undefined, $PENN = 10 * PENT$.
- PENT*: Penalty factor for transverse direction. PENN and PENT are automatically calculated by default. When PENN is defined but PENT is undefined, $PENT = PENN / 10$.
- PENTYP*: Changes how contact element stiffness is calculated:
 When PENTYP=1 (default), PENN and PENT have units of $1/(\text{Length})$, and the contact element stiffness is calculated by $K = e * E * dA$ where e represents PENN or PENT, E is the elastic modulus of the softer material in each contact pair, and dA is area. A physical interpretation is that it is equivalent to the axial stiffness of a rod with area dA , modulus E , and length $1/e$.
 When PENTYP=2, PENN and PENT become a spring rate per area $\text{Force}/(\text{Length} \times \text{Area})$, and the contact element stiffness is calculated as $K = e * dA$. The spring rate input is a more explicit way of entering contact stiffness since it is not dependent on the modulus.
- CTOL: The contact force convergence tolerance, found by comparing changes in traction between the two contacting bodies. When the convergence tolerance is less than the CTOL, the contact condition is assumed to be converged. (default = 0.01)
- MAXF: The contact algorithm iterates in an inner and outer loop. This parameter specifies the maximum number of iterations of the inner or force loop. The inner loop forces a condition of zero penetration between the contacting bodies. (default = 10)
- MAXS: Specifies the maximum number of iterations for the outer or status loop. This loop determines which contact locations (or contact elements) are active. (default = 20)
- NCHG: Specifies a number of contact elements which can be active from one iteration to the next, yet the software will consider the solution converged. If NCHG is a real number and is < 1.0 , the software treats it as a fraction of the number of active contact elements in each outer loop of the contact algorithm. The number of active contact elements is evaluated at each outer loop iteration. If NCHG is an integer ≥ 1 , the value defines the allowable number of contact changes. If $NCHG = 0$, no contact status changes can exist.
- SHLTHK: This is the shell thickness offset flag. If you set this value to 0, the contact surface is assumed to be offset $t/2$ from the nodes defining a shell element. If the FE mesh represents the outer surface, set this value to 1 so the thickness offset will be ignored. (default = 0)
- RESET: This is a flag to indicate if the contact status for a specific subcase is to start from the final status of the previous subcase. If you set this value to 0, the contact status will start from a previous subcase. If you set it to 1, the contact status will start from an initial state. (default = 0)
- INIPENE*: Controls definition of initial gap or penetration of the generated contact elements. If you set this value to 0 or 1, the contact solution will use the value calculated from the grid

coordinates. If you set the value to 2, it will initially work the same as above, but if penetration is detected, the penetration will be set to zero. If you set the value to 3, the gap and penetration will be zero for all contact elements. Option 3 is very useful when the contact edges or surfaces are intended to be coincident but due to irregularity in the meshes, some grids are not exactly on the edge or surface. This can lead to contact pressures that have a few hot spots rather than a smooth distribution. This option will ensure that a smooth distribution is obtained and in many cases the number of iterations will be reduced along with run time. (default = 0 or 1)

- **REFINE:** Determines if the mesh on the source region is refined during the contact solution. Refinement occurs when set to 2 (default). REFINE=0 turns the refinement off. REFINE=2 turns on refinement.
- **INTORD:** Determines the number of contact evaluation points for a single element face on the source region. The number of contact evaluation points is dependent on the value of INTORD, and on the type of element face. A higher number of contact evaluation points can be used to increase the accuracy of a contact solution. Inaccuracies sometimes appear in the form of nonuniform contact pressure and stress results. There may be a penalty associated with using more evaluation points since the time for a contact problem to converge may be longer. The table below shows how the number of contact evaluation points is dependent on the element type, and how it can be adjusted using the INTORD option. The “Face Type” column applies to shell elements, and to the solid element with the associated face type.

Face Type	Number of Contact Evaluation Points		
	INTORD=1	INTORD=2	INTORD=3
Linear Triangle	1	3	7
Parabolic Triangle	3	7	12
Linear Quad	1	4	9
Parabolic Quad	4	9	16

- **ZOFFSET:** Determines if the shell element z-offset is included in the contact solution. By default, ZOFFSET=0 and shell z-offsets are included. ZOFFSET=1 will prevent them from being included.
- **CSTRAT:** Under certain conditions, all of the contact elements could become inactive which may lead to singularities. Setting the parameter CSTRAT=1 will reduce the likelihood of all contact elements becoming inactive. By default, CSTRAT=0, and all contact elements can become inactive.
- **PREVIEW:** Requests the export of a bulk data representation of the element edges and faces where contact elements are created.

19.5 Penalty Factor Adjustment and Contact Stiffness

The stiffness between contact surfaces is obtained from the values of PENN, PENT, and from PENTYP. PENN and PENT, which can optionally be defined on the BCTPARM bulk entry, are automatically calculated by the software by default. The penalty factors influence the rate of convergence, and to a lesser extent, the accuracy of the contact solution. The automatic penalty factor calculation works well for most instances, but manual adjustments may be necessary, particularly if a contact problem fails to converge. The automatic penalty factor calculation is turned off if either PENN or PENT are defined.

The automatic penalty factor calculation estimates geometry characteristics using element edge lengths in the vicinity of the contact regions. The software then uses the resulting estimated lengths to automatically calculate contact penalty stiffness values PENN and PENT. The software calculated penalty factor is included in the *.f06 output.

Below are some guidelines for manually setting the BCTPARM bulk entry inputs PENTYP, PENN and PENT, along with examples to help determine characteristic geometry lengths.

PENTYP Description

When PENTYP=1 (default), PENN and PENT have units of 1/(length), and the contact element stiffness is calculated by:

$$K = e * E * dA$$

where e represents the parameters PENN or PENT

E is the elastic modulus of the softer material in each contact pair.

and dA is the area associated with the particular contact point.

A physical interpretation is that it is equivalent to the axial stiffness of a rod with area dA, modulus E, and length 1/e.

When PENTYP=2, PENN and PENT become a spring rate per area Force/(Length x Area), and the contact element stiffness is calculated by:

$$K = e * dA$$

where e represents the parameters PENN or PENT

and dA is the area associated with the particular contact point.

The spring rate input is a more explicit way of entering contact stiffness since it is not dependent on the modulus.

When PENTYP=1, PENN and PENT have units of 1/(Length), and the following is recommended.

- For solid element regions and for edge regions:
 $PENN = 10/L$ and $PENT = PENN/10$
 where L is a length characteristic of the model.
- For shell element regions:
 $PENN = 10^5 t^3 / L^4$ and $PENT = PENN/10$
 where t and L are the thickness and characteristic length of the softer of the 2 contacting surfaces, respectively.

Following are some additional guidelines when PENTYP=1:

- Very high penalty factors (in excess of 1E4) aren't recommended and will cause numerical problems, even for flat surfaces with very regular meshes.
- Very low penalty factors (less than 0.01) produce extremely slow convergence rates and aren't recommended.

When **PENTYP=2**, **PENN** and **PENT** have units of Force/(Length x Area), and the following is recommended.

- For solid element regions: $PENN=10 \cdot E/L$ and $PENT=PENN/10$
where E is the elastic modulus of the softer of the 2 contacting faces, and L is a length characteristic of the model.
- For shell element regions: $PENN=10^5 E t^3/L^4$ and $PENT=PENN/10$
where E , t , and L are the elastic modulus, thickness, and characteristic length of the softer of the two contacting shell surfaces.

The following examples may help you determine a reasonable characteristic length for penalty stiffness calculations. Flexibility exists in the definition of the length.

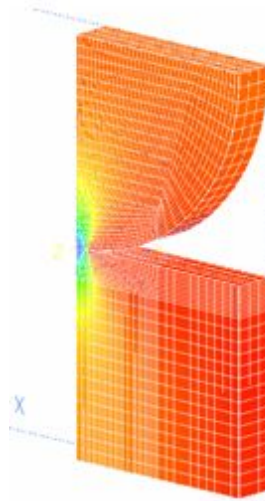


Figure 19-1. Example 1 – Use the Radius of the Contacting Cylinder or the Radius plus the block Thickness as the Length

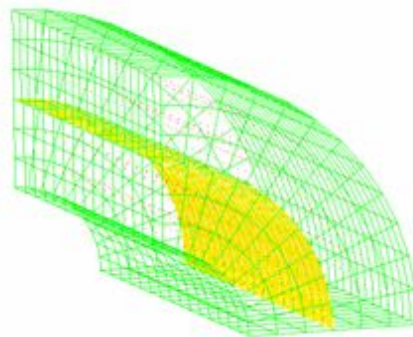


Figure 19-2. Example 2 – Use the Radius of the Inner or Outer Cylinder as the Length

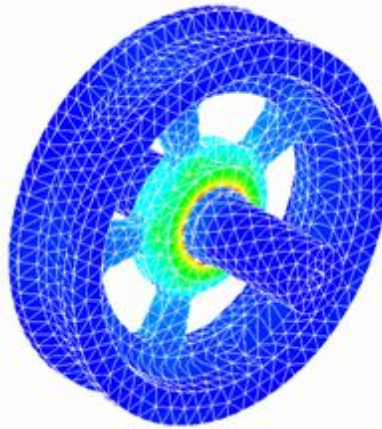


Figure 19-3. Example 3 – Use the Inner or Outer Radius of the Wheel as the Length

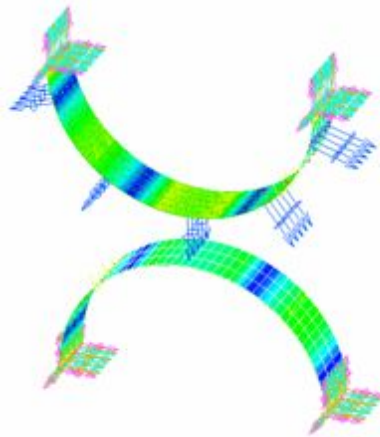


Figure 19-4. Example 4 – Use the Radius or Diameter of the Arch as the Length

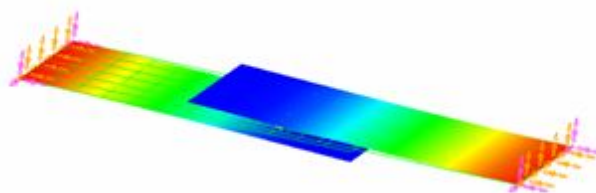


Figure 19-5. Example 5 – Use the Length of One of the Cantilever Shells as the Length

19.6 Contact Output Request

Contact results can be requested for edge-to-edge and surface-to-surface contact with the BCRESULTS case control command. The output can be requested as PRINT, PLOT, and PUNCH

for ALL grid points, or for a set (n) of grids defined with the SET command. The available output requests are:

TRACTION: Contact pressure (scalar) and in-plane contact tractions (vector in basic coordinate system) are output for each contact grid point.

FORCE: Contact force vector is output for each contact grid point.

SEPDIS: The separation distance output (SEPDIS) is a scalar quantity representing the source side normal distance to the target. During the solution, the separation distance is known at the element integration points, but is written to the grids when output. The result at each source grid is the value of separation distance at the closest contact element. If there are two or more contact elements equidistant from the grid, then the minimum value of separation distance is used at the grids rather than the average, since the average gives unexpected results for coarse meshes. SEPDIS is only supported by solution 101.

The force output for edge-to-edge contact for axisymmetric elements is “Force / Radian”. The force output for plane stress and strain elements is “Force”. This is consistent with the applied and reaction force unit for these elements.

19.7 Contact Stiffness Output

You can output the final contact stiffness matrix in DMIG format. Request the contact stiffness matrix by including the parameter KGGCPCH=1 in a SOL 101 input file. The solver writes the contact stiffness matrix from the final contact iteration (not necessarily converged) into a PUNCH file in the format required for DMIG (Direct Matrix Input at Grids).

This option is only available with the default sparse solver. The element iterative solver does not support this option.

Note

You can include differential stiffness conditions, which include contact stiffness, in a normal modes solution (SOL 103) by using the STATSUB case control command.

Similarly, you can also use the new DMIG bulk entry option to add contact stiffness to a normal modes solution.

The eigenvalues computed in a normal mode solution using the STATSUB option may be different than the eigenvalues computed using the DMIG option. The STATSUB option includes both the differential stiffness of the structure and the final contact stiffness, while the DMIG option includes only the final contact stiffness.

19.8 Contact Preview Output

NX Nastran internally creates contact elements using the contact regions and search distance you define. Although these contact elements may become inactive as the solution iterates on the contact conditions, it can be helpful to understand where they are initially created or perhaps not created.

The option to export a bulk data representation of the element edges and faces where contact elements are created is available. Setting the PREVIEW parameter on the BCTPARM bulk entry to “1” makes the request. For example:

```

$*
BCTSET      100      1      20.100000  0.00000.001000
BCTPARM     100  PREVIEW      1
$*

```

The software writes a bulk data file containing dummy shell element entries for face locations, and dummy PLOTTEL entries for edge locations. Dummy GRID, property and material entries are also written. You can import the file into a preprocessor to display both source and target contact locations.

The preview file has the following naming convention:

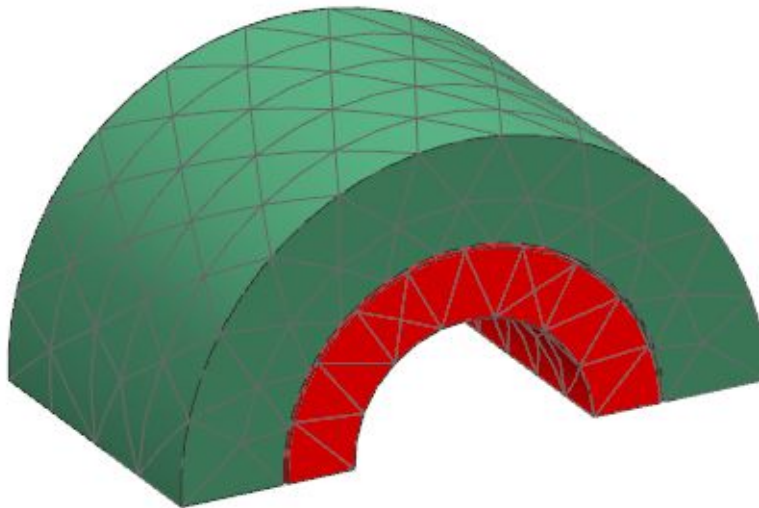
```
<input_file_name>_cnt_preview_<subcaseid>_<contactsetid>.dat
```

For example, if an input file named *test.dat* includes a subcase numbered 101 and a contact set numbered 201, the resulting preview file name would be:

```
test_cnt_preview_101_201.dat
```

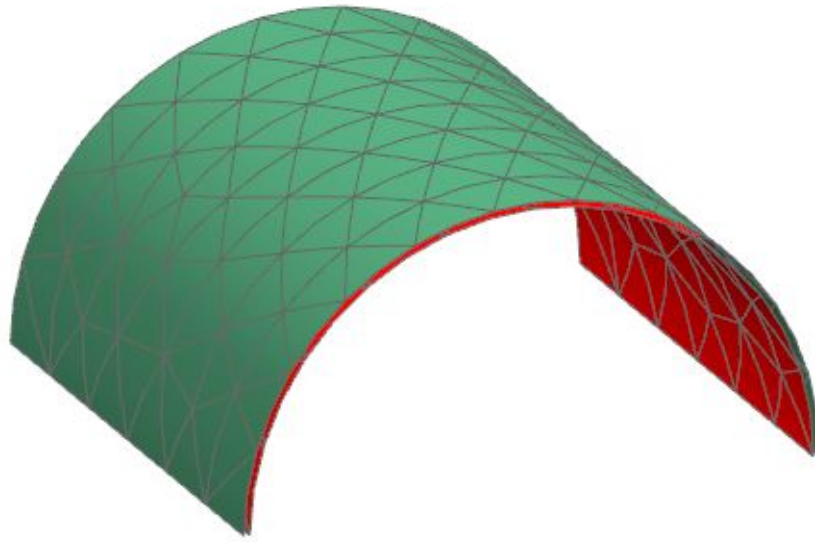
At the end of a solution, some contact elements may become inactive because they did not participate in the converged condition. To view the locations of the final active contact elements, you can request contact tractions with the BCRESULTS case control command, and then view them in a post processor.

In the following simple example, the red mesh is forced vertically into the green.

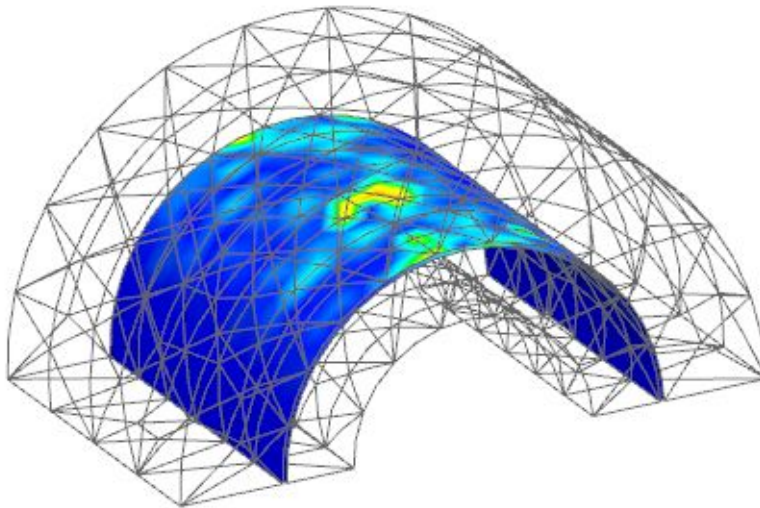


Before the software applies any loading, the initial, open contact condition is evaluated. This is the point in the solution in which the preview output is written.

The following illustration shows how the preview appears after you import the preview file into the NX preprocessor. Note that the colors were manually modified in NX after importing the preview bulk data file.



The traction results show the final contact condition after importing the results .op2 file.



19.9 Including CGAP Elements in a Contact Solution

When CGAP elements are included with linear contact, the system cell (412) OLDGAPS determines how CGAPs are treated.

- When a BCSET case control command exists, CGAP elements are treated as linear contact elements if the system cell (412) OLDGAPS is set to 0 (default). You can use CGAP elements this way with surface-to-surface contact defined (BCTSET bulk entries exist), or without (no BCTSET bulk entries exist). In the case where no surface-to-surface contact is defined, “n” on the BCSET case control command can point to a BCTPARAM bulk entry which optionally defines PENN, PENT or PENTYP for the CGAP/linear contact elements, or to nothing if a BCTPARAM bulk entry does not exist (an integer value for “n” is still required in this case).

If you do not want the CGAP to be treated as a contact element and want it handled as in previous releases as described in the *NX Nastran Element Library*, set the system cell OLDGAPS = 1.

- If a CGAP is included in a model where contact is not defined (no BCSET card), it will be treated as described in the *NX Nastran Element Library*, regardless of the value of the system cell OLDGAPS. In other words, the system cell OLDGAPS is only considered when a BCSET case control command exists.

When CGAP elements are treated as contact elements, some of the properties on the CGAP and PGAP bulk data entries are used to determine the CGAP/contact element stiffness. For contact in a linear solution, the CGAP element will use U0, MU1, KA, and KT. The following rules apply to KA and KT:

- If KA is zero or blank, the value for KB is used for the normal stiffness as long as KB is nonzero.
- If KA and KB are both zero and/or blank, the normal stiffness is calculated from the PENN and PENT fields on the associated BCTPARM. The automatic penalty factor calculation used for surface-to-surface contact does not apply to CGAP elements.
- If KA and KB are both zero and/or blank, and PENN and PENT are undefined, the values PENN=10 and PENT=1 will be used. PENTYP=2 should not be used in this case.
- If KT is zero or blank, the transverse stiffness is calculated from the PENT and PENTYP fields on the associated BCTPARM.

19.10 Contact Conditions in Dynamic Solutions (SOLs 103, 111 and 112)

A contact condition can be included in a normal mode solution (SOL 103), and in an optional dynamic response calculation (SOLs 111 and 112). In the normal mode solution, contact stiffness result is added from the end of the converged linear statics contact solution. The contact stiffness values in the normal mode solution represents the final contact condition of the structure around the contact interface. Thus, it will appear that the resulting contact edges or surfaces are attached during the normal mode analysis. Since the calculated normal modes include the final contact interface conditions, the response calculation (SOLs 111 and 112) which use these normal modes automatically include the same conditions.

The inputs for the normal mode solution are consistent with differential stiffness solutions which require a linear statics subcase. The difference is that the linear statics subcase should include the BCSET case control command. When defining the normal modes subcase, a STATSUB case control command must be included to reference the subcase id containing the contact definition. The contact solution in the linear statics subcase must fully converge before moving to the normal mode portion of the run.

Contact conditions can be used with the element iterative solver. However, differential stiffness conditions cannot be generated with the element iterative solver. Therefore, the default sparse solver will always be used, even when the element iterative solver is requested.

19.11 Contact Conditions in Linear Buckling (SOL 105)

Contact conditions can be included in a linear buckling solution (SOL 105), although there are important considerations. The initial linear statics solution determines the differential stiffness ($K\sigma$) and the final contact stiffness (K_{contact}). The final contact stiffness is added to the stiffness matrix for the buckling subcase:

$$([K + K_{\text{contact}}] + \lambda[K\sigma])\{D\}=0$$

The buckling load is determined by multiplying λ by the applied load. When contact conditions are included, this is valid when λ is close to "1.0", or when the contact conditions are insensitive to the applied load. Although contact conditions can be included in linear solutions, they are iterative and nonlinear conditions. Because contact conditions are nonlinear, the contact stiffness from the initial statics solution may be nonlinearly dependent on the applied load. As a result, the contact stiffness result which was valid for the original loading, may not be valid for the scaled load condition.

The buckling solution will report a warning if the lowest λ is not within 10% of 1.0 (0.9 - 1.1). When the warning is issued, you can scale your load by λ , and then rerun both the linear statics solution with contact conditions, and the buckling solution again. You will need to repeat this process until λ is close 1.0.

The inputs for a linear buckling solution with contact conditions require a subcase for the linear statics subcase and the buckling solution.

In addition, the linear statics subcase must include the BCSET case control command. If the linear statics subcase is the first subcase, then a STATSUB bulk entry is not needed. If it is not the first subcase, a STATSUB bulk entry is needed in the buckling subcase to reference the linear statics subcase ID.

19.12 Non-converged Contact Option

The contact convergence tolerance is calculated by comparing changes in contact forces between the two contacting bodies. A contact problem is considered converged when

- The calculated convergence tolerance is less than the value of CTOL (defined on the BCTPARAM bulk entry), and
- The number of contact changes from one iteration to the next is less than the value of NCHG (also on BCTPARAM).

Contact solutions iterate until these conditions have been met, or the number of iterations (MAXS field on BCTPARAM) has been exceeded. In either case, the solution continues, but produces a warning if convergence does not occur.

The system cell 476 can optionally be used to force a non-converged contact solution to end with an error.

By default, system cell 476 = 0, and a warning is reported in the *.f06 file for the non-converged contact solution, the solution continues and results creation occurs.

If system cell 476 = 1, a fatal error is reported in the *.f06 file for the non-converged contact solution, and the solution ends immediately with no results creation.

In most cases when a small percentage of contact elements fail to converge, they are typically not sensitive to the quality of the stored results away from the contact regions.

19.13 Contact Conditions with the Element Iterative Solver

Contact conditions can be included when using the element iterative solver. The element iterative solver is requested by including ITER=YES and ELEMITER=YES on the NASTRAN statement.

For example,

```
$* NASTRAN
$*
NASTRAN ELEMITER=YES, ITER=YES
$*
...
```

Preloaded bolts, glue conditions, and contact conditions can all be used simultaneously with the element iterative solver.

You can include converged linear contact conditions in consecutive solutions 103, 111, and 112. However, differential stiffness conditions cannot be generated with the element iterative solver. Therefore, the default sparse solver is always used to generate these conditions, even when you request the element iterative solver. See the section [Contact Conditions in Dynamic Solutions \(SOLs 103, 111 and 112\)](#) for more information.

There are a few other things to consider.

- The penalty factors PENN and PENT on the BCTPARM bulk entry affect the convergence of both the element iterative solver and the contact iterations. It is recommended to allow the software to calculate the appropriate penalty factors, which occurs by default. The automatic calculation is turned off if either of PENN or PENT are defined.
- Avoid using the element iterative solver when there is contact between solid and shell surfaces. These models typically require reducing the convergence tolerance with the ITSEPS parameter on the ITER bulk entry to 1.0E-10 or smaller, which usually increases the run time. As a result, the element iterative solver may be less efficient than the default, sparse matrix solver.
- Contact between shell surfaces is supported, although the element iterative solver works best with solid elements.

For the first iteration in each outer loop of the contact algorithm, the iterative solver convergence tolerance defined with the ITSEPS parameter on the ITER bulk entry is used (default=1e-8). For the remaining iterations in each outer loop, a tolerance 2 orders of magnitude lower is used since the displacements being calculated are delta displacements. In addition, a minimum of 20 iterations in the iterative solver are performed.

For more information on the Iterative Solutions, see:

- The chapter “*Iterative Solution of Systems of Linear Equations*” in the *NX Nastran Numerical Methods User’s Guide*.
- **ITER** in the *NX Nastran Quick Reference Guide*.

19.14 Contact Conditions with Inertia Relief

Inertia relief is an NX Nastran option that allows you to simulate unconstrained structures in a static analysis. For example, an aircraft in flight is unconstrained, yet it can be analyzed in a static analysis.

With inertia relief, the mass of the structure resists the applied loadings such that the structure is in a state of static equilibrium even though it is unconstrained in any or all degrees of freedom.

Contact conditions and inertia relief can be used together in a SOL 101 static solution.

The manual inertia relief is requested by including PARAM, INREL,-1 and the SUPORT bulk entry. The manual approach allows you to constrain some DOF while leaving others unconstrained.

The automatic inertia relief option is requested by including PARAM,INREL,-2 in the bulk data section. Since INREL=-2 requires that six rigid body modes exist (a free-free structure), active elements in the contact source and target regions need to be in close proximity to each other.

If any portion of the elements participating in contact is unconstrained and the initial distance between the active elements exceeds the distance allowed by small displacement theory, the applied contact stiffness will produce a grounding effect, and the rigid body modes will no longer exist. This usually results in all contact elements becoming inactive or in a solution with excessive penetration, which is a solution in which the contact condition is not enforced.

See the “Inertia Relief in Linear Static Analysis” chapter in the *NX Nastran User’s Guide* for more information.

19.15 Contact with Static Condensation

Static condensation is an optional NX Nastran method of partitioning and reformulating the stiffness matrix with the goal of reducing the solution time. This method requires the model be partitioned into the analysis set (A-set) and omitted set (O-set). In a static analysis, the results using static condensation are numerically exact. The partitioned solution merely changes the order of the operations of the unpartitioned solution.

An automatic static condensation option is available to use with models which include contact conditions. To select this option, set the parameter CNTASET to “YES”. For example,

```
PARAM,CNTASET,YES
```

If you select this option, you should not manually select degrees of freedom with the ASET bulk entry. When PARAM,CNTASET,YES is defined, NX Nastran automatically places the degrees of freedom which are part of the contact portion of the solution into the A-set. The remaining degrees of freedom are then automatically placed into the O-set. The result is that a static condensation is performed to reduce the full K_{gg} matrix to the K_{aa} matrix which contains only the contact degrees of freedom. The contact iterations are then performed using the resulting K_{aa} matrix.

This solution option shows the best improvement when the number of contact degrees of freedom is small compared to the overall number of degrees of freedom in the model. As the reduced A-set stiffness matrix becomes larger and more dense, the performance benefit decreases.

This option is only supported in a linear statics solution (SOL 101). The reduced mass at the contact degrees of freedom may not be an accurate representation of the original mass distribution. As a result, the software will issue a WARNING and continue without the A-set reduction if you attempt to use the PARAM,CNTASET,YES option in the dynamic solutions 103, 111, or 112.

Because PARAM,CNTASET,NO is the default option, automatic condensation does not occur by default.

Restrictions when PARAM,CNTASET,YES is defined

- If multiple subcases exist, the same contact set and the same constraint set must be used by all subcases. This can be achieved by including the BCSET and SPC case control commands in the global subcase. If the subcases use different contact and constraint sets, the software will continue without the A-set reduction.
- If multiple subcases exist, only a single constraint set can exist, and the SPC case control command must be in the global subcase.
- Bolt preload conditions are not supported.
- The iterative solver is not supported.
- You cannot select additional degrees of freedom with the ASET bulk entry to include in the K_{aa} matrix. The software determines all A-set and O-set degrees of freedom.
- The dynamic solutions 103, 111, or 112 are not supported.

See the “Understanding Sets and Matrix Operations” chapter in the *NX Nastran User’s Guide* for more information on static condensation.

19.16 SOL 101 Contact Algorithm

There are several steps in the development of a contact algorithm:

- Kinematic equations describe the relative motion of two contacting surfaces.
- Equilibrium equations with boundary conditions and contact constraints are the basic equations that are solved.
- To solve the contact problem for general bodies and surfaces, the kinematic and governing equations are converted into equivalent finite element matrix equations.
- The assembled matrix equations are solved.

Kinematic Equations

Consider the motion of a point on a hitting surface relative to a target surface. At the point on the hitting surface, the software constructs a set of Cartesian basis vectors, **e1**, **e2**, **e3**. The unit vectors, **e1** and **e2**, are tangent to the hitting surface and the unit vector, **e3**, is perpendicular to the hitting surface. The normal vector, **e3**, will also be designated as **n**.

The software assumes that the penetration of the hitting point into the target surface occurs at the target point and is:

$$p = p_o + (\mathbf{u}_H - \mathbf{u}_T) \cdot \mathbf{n}$$

where

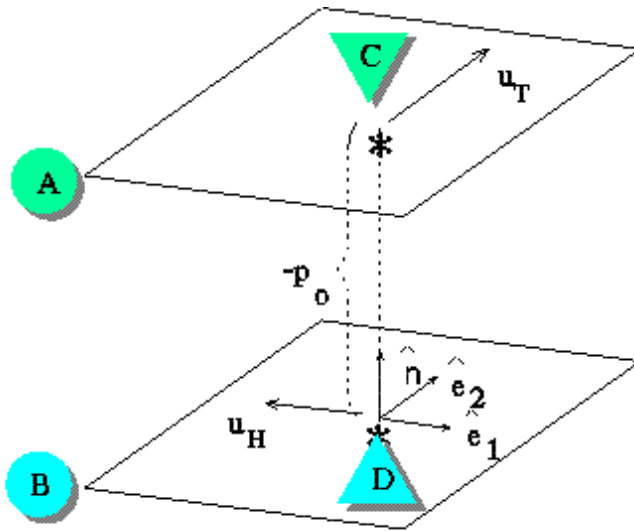
p_o is the initial penetration determined by the underlying geometry

\mathbf{u}_H is the motion of the hitting point

\mathbf{u}_T is the motion of the target point

Note that the penetration is the negative of the gap separation.

In the figure, (A) is the target surface, and (B) is the hitting surface. (C) is the target point, and (D) is the hitting point.



For calculating Coulomb friction forces, you must compute the relative tangential displacement increment, which is given by:

$$\Delta \mathbf{u}_t = (\Delta \mathbf{u}_H - \Delta \mathbf{u}_T) - [\mathbf{n} \cdot (\Delta \mathbf{u}_H - \Delta \mathbf{u}_T)] \cdot \mathbf{n}$$

This is the tangential component of the relative motion from one load case to the next. The path dependence of friction is included by applying multiple load cases in sequential order.

Normal Contact Constraints

Contact constraints are imposed at the reduced integration points on the finite element faces of the hitting contact region. These points are computed as needed during the solution.

The normal contact constraints between the two surfaces are given by:

$$p \leq 0$$

$$t_n = -\mathbf{n} \cdot \mathbf{t} \geq 0$$

$$t_n p = 0$$

- The first equation imposes the condition that the penetration of the hitting surface into the target surface can't be greater than zero. Thus, surfaces can't interpenetrate.

The contact pressure, t_n , is defined as the negative of the normal component of the surface traction.

- The second equation states that the contact pressure can't be less than zero or tensile. Or, normal tractions between surfaces can't be tensile.
- The last equation imposes the condition that:

$$p = 0 \text{ if } t_n \geq 0 \text{ and } t_n = 0 \text{ if } p \leq 0$$

Coulomb Friction Contact Constraints

For coulomb friction, additional constraint conditions are required between the hitting and target surfaces:

$$\phi = |\mathbf{t}_t| - \mu p_n \leq 0$$

$$\Delta \mathbf{u}_t = \Delta \xi \frac{\mathbf{t}_t}{|\mathbf{t}_t|}$$

where

$\Delta \xi$ is the magnitude of the relative slip increment, $\Delta \mathbf{u}_t$

$\Delta \mathbf{t}_t$ is the inplane traction

thus

$$\Delta \xi \geq 0$$

$$\phi \Delta \xi = 0$$

- The first equation imposes the constraint that the magnitude of the inplane friction traction, \mathbf{t}_t , can't exceed the coefficient of friction μ , times the contact pressure. When the magnitude of friction force reaches its maximum allowable value, the function Φ will be equal to zero.
- The second equation relates the relative tangential displacement increment between the hitting surface and the target surface, $\Delta \mathbf{u}_t$, to the magnitude of the relative slip increment, $\Delta \xi$, which must be a non-negative quantity.
- The final equation implies that if $\Delta \xi \geq 0$ (there is slipping between the surfaces) then $\Phi = 0$ and if $\Delta \xi = 0$ (the surfaces are sticking) then $\Phi \leq 0$.

Therefore, for Coulomb friction:

- The maximum possible tangential traction equals the coefficient of friction times the normal traction force.
- Contacting surfaces will "stick" if the tangential traction is less than the coefficient of friction times the normal traction force.
- Contacting surfaces will "slide" in the direction of the tangential traction if the tangential traction equals the coefficient of friction times the normal traction force.

Finite Element Kinematic Equations

With the finite element method, the contacting bodies are discretized into elements, and their surfaces are finite element faces. The contact constraint conditions are imposed at selected points on the contacting surfaces. For gap elements, the constraint conditions are enforced at nodal points.

In NX Nastran, a contact element is defined to be a hitting point, a target point, and the associated nodal degrees of freedom on the hitting and target finite element faces. Other data associated with contact elements are coefficient of friction and a user defined rigid offset, if any, at the location of the contact element. You can't directly construct the contact elements; they're formed within the solver based on your input in the bulk data file.

In terms of finite element degrees of freedom, the penetration at a particular point is given by:

$$p = p_0 + \left[\sum_{i=1}^{n_H} N_H^i u_H^i - \sum_{j=1}^{n_T} N_T^j u_T^j \right] \cdot \mathbf{n}$$

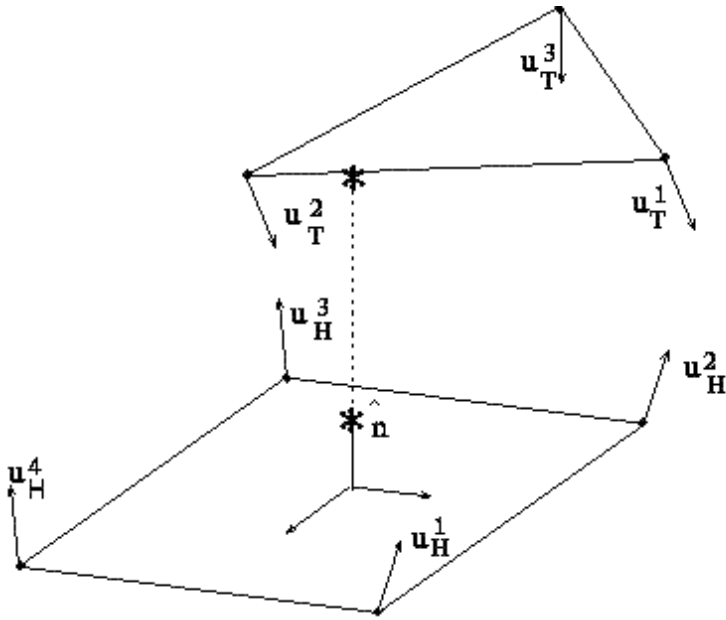
where

N_H^i are the interpolation functions for the i nodes on the hitting face

N_T^j are the interpolating functions for the j nodes on the target face

u_H^i and u_T^j are nodal displacements on the hitting and target faces

See the following figure:



The interpolation functions depend on the locations of the hitting and target points. Designating the contact element nodal displacements on the hitting and target faces as the column matrix $\{u\}$, this equation may be written as:

$$p = p_0 + [q_n] * \begin{Bmatrix} u_H \\ u_T \end{Bmatrix}$$

where the row matrix, $[q_n]$, is composed of interpolation functions times components of the surface normal. Similarly, the finite element matrix equation for tangential slip is:

$$\Delta \xi = \xi_0 + [q_t] * \begin{Bmatrix} u_H \\ u_T \end{Bmatrix}$$

For all contact elements, these two equations may be written globally as:

$$\{p\} = [Q_n]\{U\} + \{P_0\} \text{ and } \{\Delta \xi\} = [Q_t]\{U\} + \{\xi_0\}$$

Global Solution Strategy

There are several possible methods for solving this set of equations. The two methods used by NX Nastran are based on an augmented Lagrangian procedure, which offers some advantages over competing methods. A penalty stiffness is added, but unlike pure penalty methods, the satisfaction of the contact constraints may be achieved to almost any required accuracy through a series of contact traction updates. The penalty number can, therefore, be lower than in a pure penalty method, resulting in better conditioned equations. Pure Lagrange multiplier techniques require that all contacting bodies be restrained without the presence of contact constraints.

For simplification, the frictionless problem will be considered. The augmented Lagrangian potential function for the frictionless problem is:

$$\pi = 1/2 \{U\}^T [K] \{U\} - \{U\}^T \{F\} + \{T_n\}^T ([Q_n] \{U\} + \{P_o\}) + 1/2 ([Q_n] \{U\} + \{P_o\})^T [\epsilon_n] ([Q_n] \{U\} + \{P_o\})$$

where $\{T_n\}$ is the vector of unknown normal tractions (or Lagrange multipliers) at the contact points and $[\epsilon_n]$ is a diagonal matrix of normal penalty numbers. Taking the derivative of the potential function with respect to the displacements and setting it equal to zero while holding the vector of normal tractions constant produces:

$$([K] + [Q_n]^T [\epsilon_n] [Q_n]) \{U\} = \{F\} - [Q_n]^T (\{T_n\} + [\epsilon_n] \{P_o\})$$

The term $[Q_n]^T [\epsilon_n] [Q_n]$ is a penalty stiffness and $[Q_n]^T (\{T_n\} + [\epsilon_n] \{P_o\})$ is an additional force term resulting from the contact pressure and any initial non-zero penetration. Note that if penetration is zero, the penalty stiffness times displacement plus the force due to non-zero initial penetration sum to zero. So when the contact constraints are exactly satisfied, the only thing added to the global equations is the contact force. In the process of solving this set of equations, the contact pressure is iteratively updated for each contact element to enforce the zero penetration constraint condition to within a specified tolerance. The iterative update formula for the normal contact traction at a particular point is given by:

$$t_{ij}^i = t_{ij}^{i-1} + \epsilon_n p$$

where p is the current value of penetration at the point and ϵ_n is the normal penalty number.

The solution of this set of equations uses a double iteration loop. The inner force loop updates the contact tractions, $\{T_n\}$, such that a zero penetration condition is enforced for all active contact elements (to within a specified tolerance). Once the inner loop is converged, the status of all contact elements is determined in the outer loop. A contact element is set inactive and removed from the global equation if a tensile traction is required to close the contact element. If penetration occurs at an inactive contact element, it's activated and included in the global equations.

The overall solution procedure requires these steps:

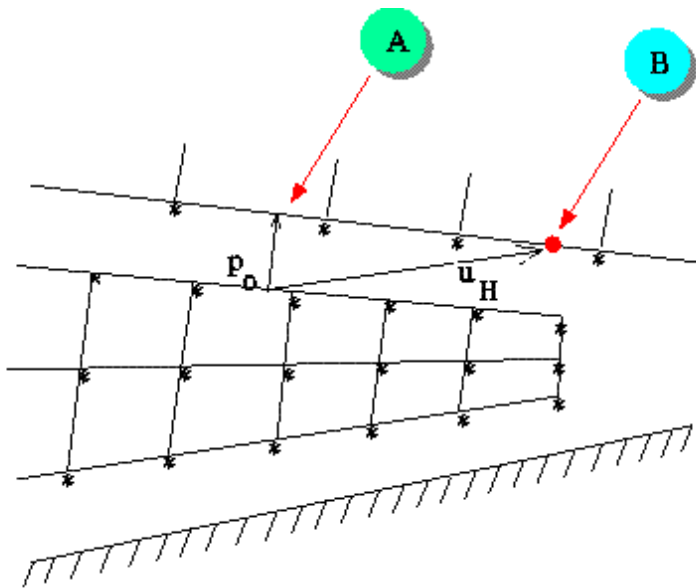
1. Form contact elements from user input of contact regions and pairs.
2. Compute stiffness for all elements including penalty stiffness for contact elements.
3. Compute loads for the current load case.
4. Begin contact outer loop iteration. Determine contact element status, depending on penetration and contact tractions.
5. Assemble contact stiffness for active contact elements into the global stiffness matrix.

6. Begin contact inner loop iteration.
7. Update contact tractions and compute contact forces.
8. Check traction convergence:
 - a. If tractions have not converged, loop back to step 6.
 - b. If tractions have converged, loop back to step 4.
 - c. If tractions have converged and the number of contact status changes is not greater than the specified allowed number of element status changes, proceed with the next load case.

Summary of Significant Aspects of Contact

1. The software assumes that penetration of the hitting point occurs along the direction of the hitting face normal. Therefore, wedge-type problems with finite sliding may be poorly simulated using linear contact analysis because contact elements are created only once using the initial geometry of the problem.

For example, in the figure, (A) is the assumed contact point; (B) is the actual contact point if the relative motion of the hitting point with respect to the target is in direction u_H .



When two surfaces are separated by an initial gap and the direction of motion is significantly different than the surface normal, then significant errors may occur (for example, wrong location for target face stresses). If the initial gap is zero (or very small), the error may be negligible.

2. The initial penetration, P_o , is computed from the FE geometry (isoparametric shape functions) which can result in kinks or bumps at element edges. These bumps usually cause local stress concentrations in a contact analysis. If possible, use the INIPENE parameter on the BCTPARM Bulk Entry to minimize these effects.
3. Gaps may be mixed with surface-surface contact, thus letting you include beams and bars in contact models. The gap element analysis has been reformulated as an augmented Lagrangian procedure so that the same solution strategy is used for both contact and gaps.

4. The behavior of the iterative solution is influenced by the penalty numbers (normal and friction). The defaults provide rapidly converging solutions for most problems. However, when the iterations don't converge, you may need to adjust the penalty numbers. Some general trends are:
 - a. The softer the hitting or target surface(s), the smaller the normal penalty number, P_n
 - b. Larger penalty numbers usually speed convergence, but if penalty numbers are too large, ill conditioned equations may result in MAXRATIO failures.
 - c. The friction penalty number is usually 10 to 100 times smaller than the normal penalty. When many elements are sliding or sticking, a smaller penalty factor may speed convergence.
 - d. If an unrestrained portion of the model exhibits rigid body motions about a contact region, reducing the penalty number may stabilize the solution.

The penalty numbers can be adjusted on the BCTPARM Bulk Entry.

5. Contact can remove rigid body motions from the structure. However, use caution because tensile contact elements are removed as the solution progresses. Therefore, if an insufficient number of active contact elements are present to prevent rigid body motion, the stiffness matrix will become singular and the solution will fail (or provide incorrect results).

Chapter 20: Gluing Elements

An option to glue elements together during a solution is available in NX Nastran. Glue is a simple and effective method to join meshes which are dissimilar. It correctly transfers displacement and loads resulting in an accurate strain and stress condition at the interface. The grid points on glued edges and surfaces do not need to be coincident.

Glue creates stiff springs or a weld like connection to prevent relative motion in all directions. For discussion purposes, the internal elements created by the solver from glue definitions will be referred to as “glue elements” in the documentation.

The following table summarizes all supported glue conditions.

Table 20-1. Glue Summary		
Type	Description	Solution Support
Edge-to-Edge	Glue between the edges of shell, axisymmetric, plane stress, and plane strain elements. Elements being glued must be in the same plane.	All solutions except 144 -146, 401, and 701. SOL 601 does not support gluing of shell edges. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.
Edge-to-Surface	Glue between shell element edges and shell or solid element faces.	All solutions except 144 -146, 401, 601, and 701. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.
Surface-to-Surface	Glue between shell or solid element faces and shell or solid element faces.	All solutions except for SOL 144–146, and 701. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They can be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.

Defining Glue Conditions

- Select the glue condition with the BGSET case control command. See [Defining and Selecting Glue Pairs](#).

- Define source and target regions. See [Glue Regions](#).
- Pair the source and target regions. See [Defining and Selecting Glue Pairs](#).
- Optionally adjust the glue algorithm using glue control parameters. See [Glue Control Parameters](#).
- Optionally request glue surface traction output. See [Glue Output Request](#).

Edge-to-Edge Summary

You can define edge-to-edge glue between the edges of the following elements:

- Shell elements CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR
- Axisymmetric elements CTRAX3, CQUADX4, CTRAX6, CQUADX8
- Plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8
- Plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8

You create edge regions with either the BEDGE or BLSEG bulk entries, then pair the regions using the source and target fields on the BGSET bulk entry.

Gluing the Edges of Axisymmetric, Plane Stress, and Plane Strain Elements

- The axisymmetric, plane stress, and plane strain elements can be defined in either the XZ plane or in the XY plane. Edge-to-edge glue is supported in either orientation.

From the source region, the software searches in the outward normal direction. In addition, the software searches a small distance in the inward normal direction in order to glue edges that may interfere due to meshing irregularities.

A glue connection occurs when the following is satisfied.

- When a projected normal hits the target side.
- The distance between the edges is equal to or less than the search distance which you specify for the glue pair on the BGSET entry.

The input files *gluedg** are included in the *installation_path\nxnr\nast\tpl* directory to demonstrate.

Gluing the Edges of Shell Elements

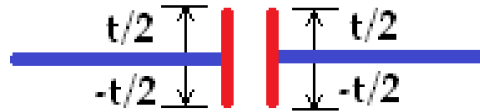
- A simplistic description of shell element edge-to-edge glue is that the software creates pseudo-faces along the edges. From the glue points on the source side pseudo-face, the software projects a normal in the outward normal direction. In addition, the software searches a small distance in the inward normal direction in order to glue edges that may interfere due to meshing irregularities.

A glue connection occurs when the following is satisfied.

- When a projected normal hits the pseudo-face on the target side.
- The distance between the two pseudo-faces is equal to or less than the search distance in which you specify for the glue pair on the BGSET entry.

The larger the angle is between neighboring shell elements, the less likely that a successful projection can occur. For example, a glue connection cannot occur if the neighboring shell elements are perpendicular. In this case, you could consider an edge-to-surface glue connection.

The image below helps to illustrate. The blue line represents the shell elements, and the red line represents an internally created pseudo-face along the edges in the source and target regions. The pseudo-face can be visualized by extruding the element edge from $-t/2$ to $+t/2$ in the parent element normal direction.



Edge-to-edge glue definitions are supported in all solutions except SOLs 144 -146, 601 and 701. In the heat transfer solutions 153 and 159, they are treated as conductivity connections. The edges of shell elements that use the plane strain option defined with MID2=-1 on the PSHELL entry are not supported in edge-to-edge glue.

The input files *gluedg** are included in the *installation_path\nxn\nast\tpl* directory to demonstrate.

Edge-to-Surface Summary

- Edge regions are edges of the shell elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR.
- Surface regions are the faces of the shell elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR and the faces of the solid elements CHEXA, CPENTA, CPYRAM, and CTETRA.

You create shell element edge regions with the BEDGE or BLSEG bulk entries, and you create shell or solid element face regions with the BSURF, BSURFS, BCPROP, or BCPROPS bulk entries. You then pair the regions using the source and target fields on the BGSET bulk entry. The source region in the pair must be the edge region. The target region consists of shell or solid element faces.

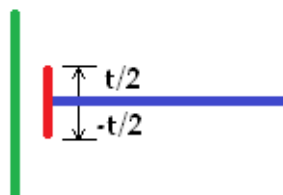
A simplistic description of edge-to-surface glue is that the software creates pseudo-faces along the edges in the source region. It then connects these pseudo-faces to the shell or solid faces in the target region with weld like connections.

From the glue points on the source side pseudo-face, the software projects a normal in the outward normal direction. In addition, the software searches a small distance in the inward normal direction in order to glue edges and surfaces that may interfere due to meshing irregularities.

A glue connection occurs when the following is satisfied.

- When a projected normal hits the face on the target side.
- The distance between the two is equal to or less than the search distance which you specify for the glue pair on the BGSET entry.

For example, the green line below represents shell or solid free faces in a target region, the blue line represents shell elements, and the red line represents an internally created pseudo-face along the edges in the source region. The pseudo-face can be visualized by extruding the element edge from $-t/2$ to $+t/2$ in the parent shell element normal direction.



Edge-to-surface glue definitions are supported in all solution sequences except solutions 144-146, 601 and 701. In the heat transfer solutions 153 and 159, they are treated as conductivity connections. They cannot be used to represent acoustic glue connections. The edges of shell elements that use the plane strain option defined with MID2=-1 on the PSHELL entry are not supported in edge-to-surface glue.

The input files *gluedg*.dat* are included in the *installation_path\nxnr\nast\tpl* directory to demonstrate.

Surface-to-Surface Glue Summary

Surface-to-surface glue can be defined on the faces of the following elements.

- Shell elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR.
- Solid elements CHEXA, CPENTA, CPYRAM, and CTETRA.

You create shell or solid element face regions with the BSURF, BSURFS, BCPROP, or BCPROPS bulk entries. You then pair the regions using the source and target fields on the BGSET bulk entry. From elements in the source region, a normal is projected.

From solid face regions, the software searches in the outward normal direction. In addition, the software searches a small distance in the inward normal direction in order to glue solid faces that may interfere due to meshing irregularities.

From shell element face regions, the software searches in both the top and bottom normal directions.

The software creates a glue element if:

- Any of the source element normals intersect with an element in the target region.
- The distance between the two faces is equal to or less than the search distance which you specify for the glue pair on the BGSET entry.

Surface-to-surface glue definitions are supported in all solution sequences except for SOL 144–146, and 701. In the heat transfer solutions 153 and 159, they are treated as conductivity connections. Surface-to-surface glue connections can also be defined between acoustic mesh faces. See [Glue Conditions in Acoustics Analysis](#)

Input files *glue*.dat* are included in the *installation_path\nxnr\nast\tpl* directory to demonstrate.

20.1 Glue Regions

Edge Glue Regions

An edge glue region is a collection of shell, axisymmetric, plane strain, and plane stress element edges in a section of the model where you expect glue to occur. These regions can be created using either the BEDGE bulk entry or the BLSEG bulk entry.

- To define a glue edge using the BEDGE entry, you enter the element ID along with the corner GRID IDs. You can define the edges on the BEDGE entry in any order.
- To define a glue edge using the BLSEG entry, you enter the GRID IDs in a continuous topological order. You must include both midside and corner GRID IDs on the BLSEG entry.

An element should not appear more than once in the regions that are part of a glue/contact pair. All region IDs defined with the BEDGE, BLSEG, BSURF, BSURFS, BCPROP, or BCPROPS entries must be unique.

Surface Glue Regions

A surface glue region is a collection of element free faces in a section of the model where you expect glue to occur. These regions can be created using the shell elements (BSURF and BCPROP) and using solid element free faces (BSURFS and BCPROPS).

- The BSURF entry is a list or range of shell element IDs to include in the region.
- The BCPROP entry is a list of shell element property IDs. Shell elements which use any of these listed property IDs will be included in the region.
- The BSURFS entry is a list of solid element IDs each followed by 3 grid points defining which face of the 3-D element to include in the glue region.
- The BCPROPS entry is a list of solid element property IDs. The free faces of the solid elements selected with a property ID are automatically determined by the software.

An element should not appear more than once in the regions that are part of a glue/contact pair. All region IDs defined with the BEDGE, BLSEG, BSURF, BSURFS, BCPROP, or BCPROPS entries must be unique.

20.2 Defining and Selecting Glue Pairs

A glue pair is a way to combine two regions, source and target, in which gluing will be analyzed during the solution. Use the BGSET bulk entry to define each glue pair. The fields on the BGSET bulk entry are described as follows:

- GSID (glue set id) will need to match the value of 'n' on the BGSET case control entry for the solution to recognize the glue definition.
- SIDi and TIDi (source and target id's) refer to regions created by the BSURF, BCPROP, BSURFS, BCPROPS, BEDGE, and BLSEG entries, and are used to define source and target regions respectively for a pair. For edge-to-surface glue pairs, the BEDGE and BLSEG IDs, which define the edge region, must be the source region ID, and a shell or solid element face region ID as the target region ID. As many pairs as desired can be included on a single BGSET bulk entry.
- SDIST (search distance) defines the distance in which the solver can initially determine if the distance between element edges or faces in a particular pair are within the threshold for creating glue elements. The default value of SDIST of 10 is large enough to handle most geometry situations, but can be adjusted as needed. This value is used once, at the beginning of the solution, to determine where glue elements need to be initially created.

Combining Glue Sets – BGADD

You can optionally define multiple BGSET/BGPARM bulk entry sets, each set with unique glue set IDs (GSID), and then combine them with a single BGADD bulk entry. The multiple BGSET/BGPARM bulk entry sets are created to adjust certain glue parameters locally. Glue parameters can also be adjusted globally with a BGPARM bulk entry having the same GSID as the BGSET case control command.

The following example demonstrates the inputs.

```

CASE CONTROL
$GSID on the BGSET case control matches GSID on BGADD
BGSET = 108
...
BULK DATA
$Local Glue Set definitions
BGSET 1 1 2
BGSET 2 3 4
5 6
...
$Local Glue Parameters
BGPARM 1 PENN 80 PENT 80
BGPARM 2 PENN 110 PENT 110
...
$Local Glue Sets are combined with BGADD
BGADD 108 1 2
...
$Global Glue Parameters
BGPARM 108 REFINE 0GLUETYPE 1

```

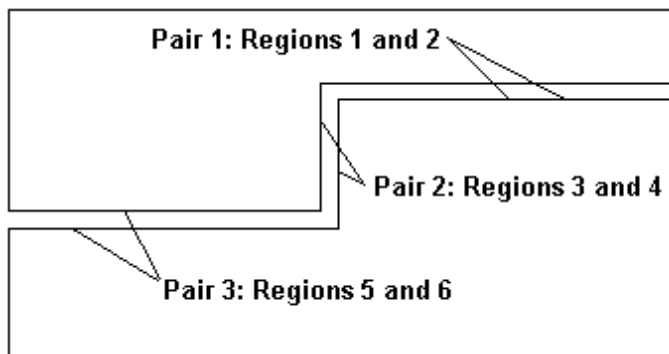
See the section **Glue Control Parameters – BGPARM Bulk Entry** in this chapter for more information on glue parameters.

Gluing Composite Solid Faces

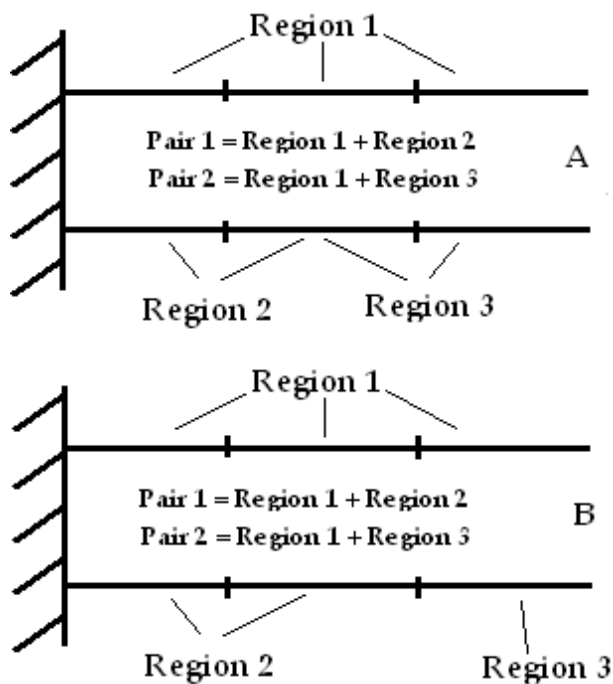
Defining glue regions and pairs on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the glue definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BGRESULTS case control command.

Additional Recommendations

When defining glue regions and pairs on geometry which are not tangent continuous, creating single glue regions which cross corner transitions can result in non-uniform stress results around the corners. It is recommended to break these areas into multiple regions and pairs as shown below.



When defining glue regions and pairs, it is recommended to not include the same element face in multiple regions. In “A” below, an element is repeated in regions 2 and 3. In “B”, the same element only exists in region 2. “B” is recommended. Repeating element faces multiple times in the same or different regions can significantly increase memory requirements and degrade performance.



20.3 Glue Control Parameters

The glue control parameters on the BGPARM bulk entry (optional) can help you adjust the glue algorithm. For most solutions, the default settings are appropriate, thus the BGPARM entry is not required.

- For edge-to-edge glue, the parameters PENTYPE, PENGLUE, and PREVIEW are supported.
- For edge-to-surface glue, the parameters PENTYPE, PENGLUE, PREVIEW, and ESOPT are supported.
- For surface-to-surface glue, all of the parameters except ESOPT are supported.

You can optionally define multiple BGSET/BGPARM bulk entry pairs, each pair with unique glue set IDs (GSID), and then combine them with a single BGADD bulk entry.

The multiple BGSET/BGPARM bulk entry pairs are created to adjust certain glue parameters locally. Glue parameters can also be adjusted globally with a BGPARM bulk entry having the same GSID as the BGSET case control command.

Global and local glue parameters have the following definition and rules:

- Global Glue Parameters

The BGPARM bulk entry, which uses the same GSID entered on the BGSET case control command, defines global parameters.

Any of the parameters on the BGPARM bulk entry can be defined globally. A parameter's default value is used if it is not defined globally or locally.

- Local Glue Parameters

The BGPARM bulk entries associated to individual BGSET bulk entries, which are then combined with a BGADD bulk entry, define local parameters.

The parameters GLUETYPE, PENN, PENT, PENTYP, and PENGLUE can be defined locally. A local parameter definition overrides a global definition.

Glue Parameter Descriptions

- Following are general definitions for the parameters GLUETYPE, PENTYP, PENN, PENT, and PENGLUE, along with details below.

GLUETYPE – Selects the glue formulation.

GLUETYPE = 1 - Normal and tangential springs will be used to define the connections.

GLUETYPE = 2 - A “weld like” connection will be used to define the connections.

PENTYP – Changes how glue element stiffness and “conductance” are calculated. (Default=1)

PENN – Penalty factor for normal direction when GLUETYPE=1. (Default=100)

PENT – Penalty factor for transverse direction when GLUETYPE=1. (Default=100)

PENGLUE – Penalty factor when GLUETYPE=2. (Default=1)

When gluing acoustic faces, GLUETYPE, PENTYPE, and PENT are all ignored. PENN is described in the section “Glue Conditions in Acoustics Analysis”.

For structural solutions and for thermal solutions 153 and 159, GLUETYPE has unique penalty factor inputs. These inputs and units are described below.

GLUETYPE=1	The glue penalty stiffness is defined by PENN and PENT. (Solutions 153 and 159 - Heat transfer analysis always uses GLUETYPE=1)	
	PENTYP=1	<p>Structural solutions: PENN and PENT have units of 1/(length), and the glue element stiffness is calculated by $K = e \cdot E \cdot dA$ where e represents PENN or PENT, E is an average modulus, and dA is area. See Average modulus computation below. A physical interpretation is that it is equivalent to the axial stiffness of a rod with area dA, modulus E, and length $1/e$.</p> <p>Solutions 153 and 159: PENT is ignored. PENN has the units of 1/(length), and “conductance” at the glue connection is calculated as $C = e \cdot k_{avg} \cdot dA$, where e represents PENN, k_{avg} is an average of the thermal conductivity (k) values for all source side elements in a pair, and dA is area. A physical interpretation is that it is equivalent to the axial “conductance” of a rod with area dA, conductivity k_{avg}, and length $1/e$.</p>
	PENTYP=2	<p>Structural solutions: PENN and PENT become a spring rate per area Force/(Length x Area), and the glue element stiffness is calculated as $K = e \cdot dA$. The spring rate input is a more explicit way of entering glue stiffness since it is not dependent on the average modulus.</p> <p>Solutions 153 and 159: PENT is ignored. PENN has the units of (thermal conductivity*length)/area, and the</p>

	"conductance" at the glue connection is calculated as $C = e \cdot dA$. Another term for e is heat flux.	
GLUETYPE=2	The glue penalty stiffness is defined by the PENGLUE penalty factor (default=1)	
Structural solutions:	PENTYP=1	PENGLUE is a unitless value (glue stiffness scale factor).
	PENTYP=2	PENGLUE has the units of F/L^2 .
Solutions 153 and 159:	Always uses GLUETYPE=1.	

For glued coincident faces, there is little flexibility between the faces with default penalty factors. Regardless of which GLUETYPE is used, the glue condition created between non-coincident faces will not usually produce a local stiffness as accurate as using a conventional finite element for the connection. The flexibility in the glue condition will depend on the GLUETYPE used, and the value of the penalty factors. If you have non-coincident faces and the glue joint flexibility is important, then it is recommended that you model this connection with conventional finite elements.

Average modulus computation

- o Generally, the modulus is averaged for the elements associated with the source side region.
- o For elements using orthotropic and anisotropic materials, the element modulus value used in the average modulus computation is computed as follows.
 - MAT2: $E = (C11 + C22)/2$ (C33 is not used)
 - MAT8: $E = (E1 + E2)/2$
 - MAT9: $E = (C11 + C22 + C33)/3$
 - MAT11: $E = (E1 + E2 + E3)/3$
- o For an element defined with a PSHELL entry, the modulus of the material selected in the MID2 field on the PSHELL entry is used in the average modulus computation. If the MID2 field is undefined, the material selected in the MID1 field is used.
- o For elements defined as a shell composite defined with a PCOMP or PCOMPG entry, the element modulus value used in the average modulus computation is computed as:
 - $E = E'(1 - PR \cdot PR)$
 - where $E' = (C11 + C22)/2$,
 - $PR = C12/C11$,
 - and C11, C22, C12 are the values defined on the software computed MAT2 entry selected in the MID2 field on the software computed equivalent PSHELL entry. If the MID2 field is undefined, the MAT2 entry selected in the MID1 field is used.
- o When a solid composite defined with the PCOMPS entry is included in a source glue region, the average modulus is computed by averaging all modulus values defined on *all* materials defined in the input file.

- INTORD and REFINE help to improve the accuracy of the glue solution. The number of locations where normals are projected (glue points) from the source region is dependent on the value assigned to the INTORD parameter, and on the element face type. The following table summarizes how the INTORD value adjusts the number of glue points for a particular element face:

Face Type	Number of Glue Points Used in Glue Element Evaluation		
	INTORD=1	INTORD=2 (default)	INTORD=3
Linear Triangle	1	3	7
Parabolic Triangle	3	7	12
Linear Quad	1	4	9
Parabolic Quad	4	9	16

REFINE will increase the number of glue points by refining the mesh on the source region. Part of the refinement process is to project element edges and grids from the associated target region back to the source region. The resulting refinement on the source region is then more consistent with the target side, which then gives a better distribution of glue elements. The refined grids and elements are only used during the solution. The glue results are transferred back to the original mesh for post processing results.

Refinement occurs when set to 2 (default). REFINE=0 turns the refinement off.

- You can use the PREVIEW parameter to request the export of a bulk data representation of the element edges and faces where glue elements are created. See [Glue Preview](#).
- You can use the ESOPT parameter to change how the edge-to-surface glue stiffness is distributed on the surface being glued. See [Edge-to-Surface Glue Stiffness](#).

20.4 Glue Conditions in Acoustics Analysis

Surface-to-surface gluing can be used between acoustic mesh faces. The acoustic mesh, which uses the solid elements CHEXA, CPENTA, CPYRAM, and CTETRA, can be a continuous 3-D mesh, or it can consist of several independent meshes with adjacent acoustic faces glued. Similar to structural glue conditions, the meshes between the acoustic glued faces can be dissimilar.

The acoustic-to-acoustic glue inputs are consistent with the structural-to-structural capability, except on the BGPARM bulk entry, GLUETYPE, PENTYPE, and PENT are all ignored. PENN is used to calculate the acoustics penalty matrix K:

$$K = e * \frac{1}{\rho} * dA$$

where e is PENN, ρ is the average density of all fluid elements in the model, and dA is the surface area. The K matrix for an acoustic element is defined by

$$K = \frac{1}{\rho} \int_V \nabla N^T \nabla N dV$$

For a small fluid column (tube of length L and cross-sectional area dA), the K matrix can be written as

$$K = \frac{1}{L} * \frac{1}{\rho} * dA$$

Therefore, the penalty factor PENN (e) can be interpreted as 1/L.

Also see “Coupled Acoustic-Structural Analysis” in the *NX Nastran User's Guide* for details on acoustics.

The input file *glueac107a.dat* is included in the *installation_path\nxnr\nast\tpl* directory to demonstrate.

20.5 Glue Output Request

An option to recover glue surface tractions for solid and shell elements is available. The BGRESULTS case control command is used to request the glue traction output. The glue traction request is supported in a linear static solution (SOL 101), in a normal modes solution (SOL 103) but only for a static preload subcase if present, and in a linear buckling solution (SOL 105). The glue results from SOL 105 are a result of the applied static loads which are not necessarily the loads at which buckling occurs.

The glue tractions, which are similar to the contact results, are calculated and stored at the grids which are on the glue surfaces. The normal component of the tractions is a scalar while the in-plane (tangential) tractions are output in the basic coordinate system. For an edge-to-edge and edge-to-surface glue pair, only forces and not tractions are recovered. Glue force recovered on axisymmetric element edges is force per unit radian, which is consistent with applied force and reaction force for these elements.

The output can be requested as PRINT, PLOT, and PUNCH for ALL grid points, or for a set (n) of grids defined with the SET command.

20.6 Glue stiffness output

You can output the glue stiffness matrix in DMIG format. In your input file which includes a glue definition, you can request the glue stiffness matrix by including the following parameter input.

```
PARAM,KGGLPCH,1
```

The solver writes the glue stiffness matrix into a PUNCH file in the format required for DMIG (Direct Matrix Input at Grids). You can request the glue stiffness matrix in any solution except 144-146, 401, 601 and 701. This option is only available with the default sparse solver. The iterative solvers do not support this option.

To include the glue stiffness in a consecutive solution:

- You include the K2GG case control command to select the DMIG entry.

```
...
Case Control
k2gg=KGGG
...
```

- You will include the DMIG in the bulk data section of your consecutive input file. You can optionally use the INCLUDE entry. For example,

```
...
begin bulk
```

```
...
include name_of_dmg_punch_file.pch
...
```

20.7 Glue preview

NX Nastran internally creates glue elements using the glue regions and search distance you define. You can optionally export a bulk data representation of the element edges and faces where glue elements are created. To do so, set the PREVIEW parameter on the BGPARM bulk entry to “1”. For example:

```
$*
BGSET      100      1      2      0.001000
BGPARM     100  PREVIEW      1
$*
```

The software writes a bulk data file containing dummy shell element entries for face locations, and dummy PLOTTEL entries for edge locations. Dummy GRID, property, and material entries are also written. You can import the file into a preprocessor to display both source and target glue locations.

To view the glue locations, glue tractions can also be requested in solutions 101, 103, and 105. In the solutions in which glue tractions are not supported, you can use the preview output.

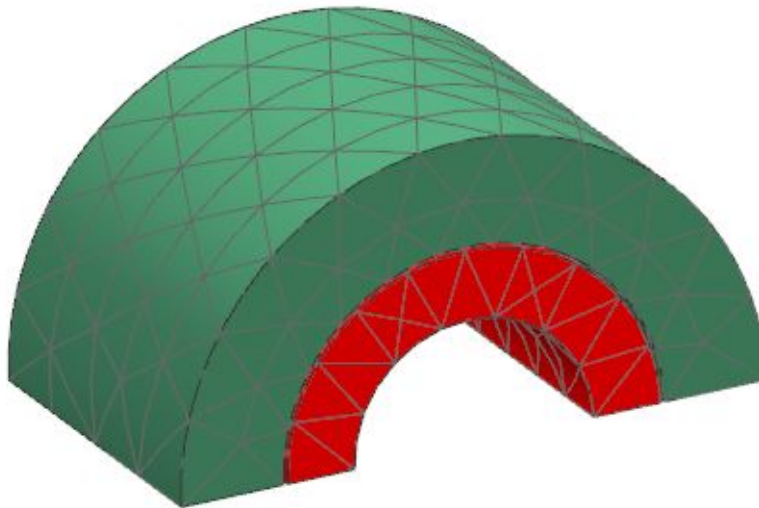
The preview file has the naming convention:

```
<input_file_name>_glue_preview_<subcaseid>_<gluesetid>.dat
```

For example, if an input file named *test.dat* includes a subcase numbered 101 and a glue set numbered 201, the resulting preview file name is:

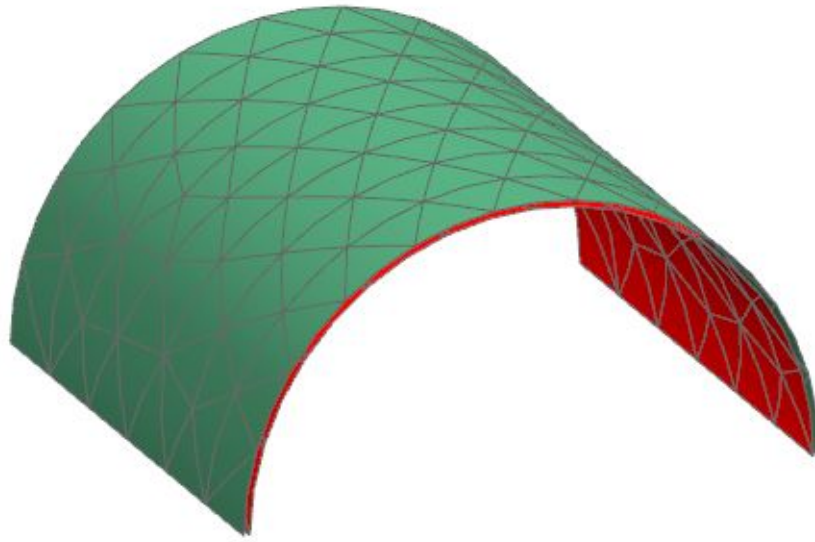
```
test_glue_preview_101_201.dat
```

In the following simple example, the red mesh is glued to the green.



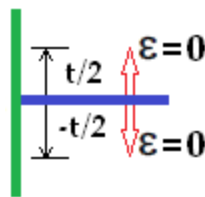
Before applying any loading, the software creates the glue elements. This is the point in the solution in which the preview output is written.

The following illustration shows how the preview appears after you import the preview file into the NX preprocessor. Note that the colors were manually modified in NX after importing the file.

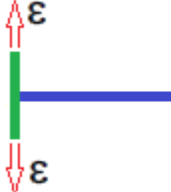


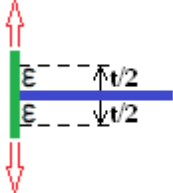
20.8 Edge-to-Surface Glue Stiffness

Shell theory does not account for changes in shell thickness or normal strains perpendicular to the plane of the shell element. The ESOPT parameter gives you a choice of how to handle the linking of the zero normal strains which exist in the shell element associated with the edge (colored blue in the example), to the surface being glued (colored green in the example).

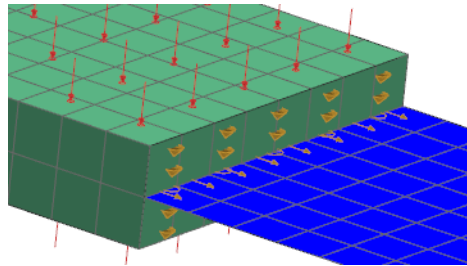


Either of the ESOPT options properly transfers all forces and moments through the glue connection. The resulting difference between these options is the local strains on the surface being glued.

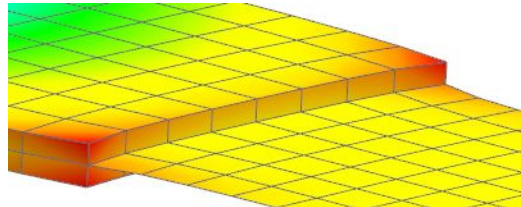
ESOPT=0 (default)	<p>The strains in the plane of the surface being glued in the direction perpendicular to the edge are not constrained by the glue stiffness.</p> 
--------------------------	---

ESOPT=1	<p>The strains in the plane of the surface being glued on the area corresponding to the shell element thickness ($t/2$) in the direction perpendicular to the edge, are constrained by the glue stiffness. This is the glue stiffness behavior which existed before NX Nastran 8.5.</p> 
----------------	---

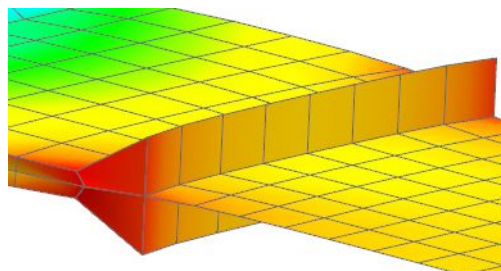
The following example illustrates the ESOPT options. It includes an equal compressive pressure load on the top and bottom of the solid element mesh. The edge of the blue shell mesh is glued to the face of the green solid mesh.



When **ESOPT=0 (default)**, the deformations in the solid mesh demonstrate that the glue stiffness did not prevent local strain at the glue interface.



When **ESOPT=1**, the deformations in the solid mesh demonstrate that the glue stiffness did prevent local strain at the glue interface.



Chapter 21: Plotting

- *Overview of Plotting in NX Nastran*
- *Superelement Plotting*
- *Post Processors*

21.1 Overview of Plotting in NX Nastran

NX Nastran can generate the following types of plots:

- X-Y graphs of transient response or frequency response.
- V-F and V-G graphs for flutter analysis.

X-Y Plots

Requests for structure plots or X-Y plots are made in the Case Control Section by submitting a structure plot request or an X-Y output request.

The optional PLOTID command is considered to be part of the plot request although it must precede any OUTPUT(PLOT), OUTPUT(XYOUT) or OUTPUT(XYPLOT) commands.

Plot requests are separated from Case Control by the OUTPUT(PLOT), OUTPUT(XYPLOT) or OUTPUT(XYOUT) commands. Data above this command (except PLOTID) will not be recognized by the plotter, even though it may have the same name (for example, the SET command).

See also

- “X-Y PLOT Commands” in the *NX Nastran Quick Reference Guide*
- “PLOTID” in the *NX Nastran Quick Reference Guide*

Flutter Analysis Plots

In NX Nastran, you can use the aeroelasticity capabilities to perform a flutter analysis to determine the stability of an aeroelastic system. You can then create graphs of your results.

See also

- “Flutter Analysis” in the *NX Nastran Aeroelastic Analysis User's Guide*

21.2 Superelement Plotting

Plotting can occur at four different places in the superelement solution sequences. There are two plot commands, SEPLOT and SEUPPLOT, used with other Case Control and PARAM commands to control the type of plot to be prepared. A flow chart of the solution process is given in [Figure 21-1](#), [Figure 21-2](#), and [Figure 21-3](#).

Undeformed structure plots are made early during Phase 0 in the structured solution sequences and during Phase I in the unstructured solution sequences. They can be made for either one superelement only, or for a superelement and all of its upstream members, as controlled by PARAM,PLOTSUP (see [Figure 21-1](#) or [Figure 21-2](#)). Plots are made for superelements selected by the SEMG command (see [Figure 21-2](#), block I), and listed in a plot request headed by SEPLOT SEID, where SEID is the superelement identification number. In the structured solution sequences, the SEMG command is not required and all superelements which appear on SEPLOT commands will be plotted (see [Figure 21-1](#), block 0). Undeformed plots can be used to check geometry and connectivity, and do not require the presence of property or material commands. In the unstructured

solution sequences, a branch to the end of the loop immediately after the plot module can be made by using the `PARAM,PLOT,-1` command.

In dynamic analysis, solution set XY-plots are requested by use of `SEPLOT 0` (see [Figure 21-3](#), block II). In the data recovery phase, XY-plots and deformed structure plots for elements in one superelement only are requested by the `SEPLOT` command (see [Figure 21-3](#), block III). Deformed structure plots for a superelement and all its upstream plots are requested with the `SEUPPLOT` command (see [Figure 21-3](#), block IV).

XY-plots and deformed structure plots are regarded as output requests and will result in automatic execution of the data recovery loop as is required to produce the plots requested. For example, the command `SEUPPLOT 0` will result in data recovery being performed on the entire model, even in the absence of any other output requests. For `SEUPPLOT` requests, if `SUBCOM` or `SYMCOM` subcases are used, each superelement must have identical `SUBCOM` and `SYMCOM` structure.

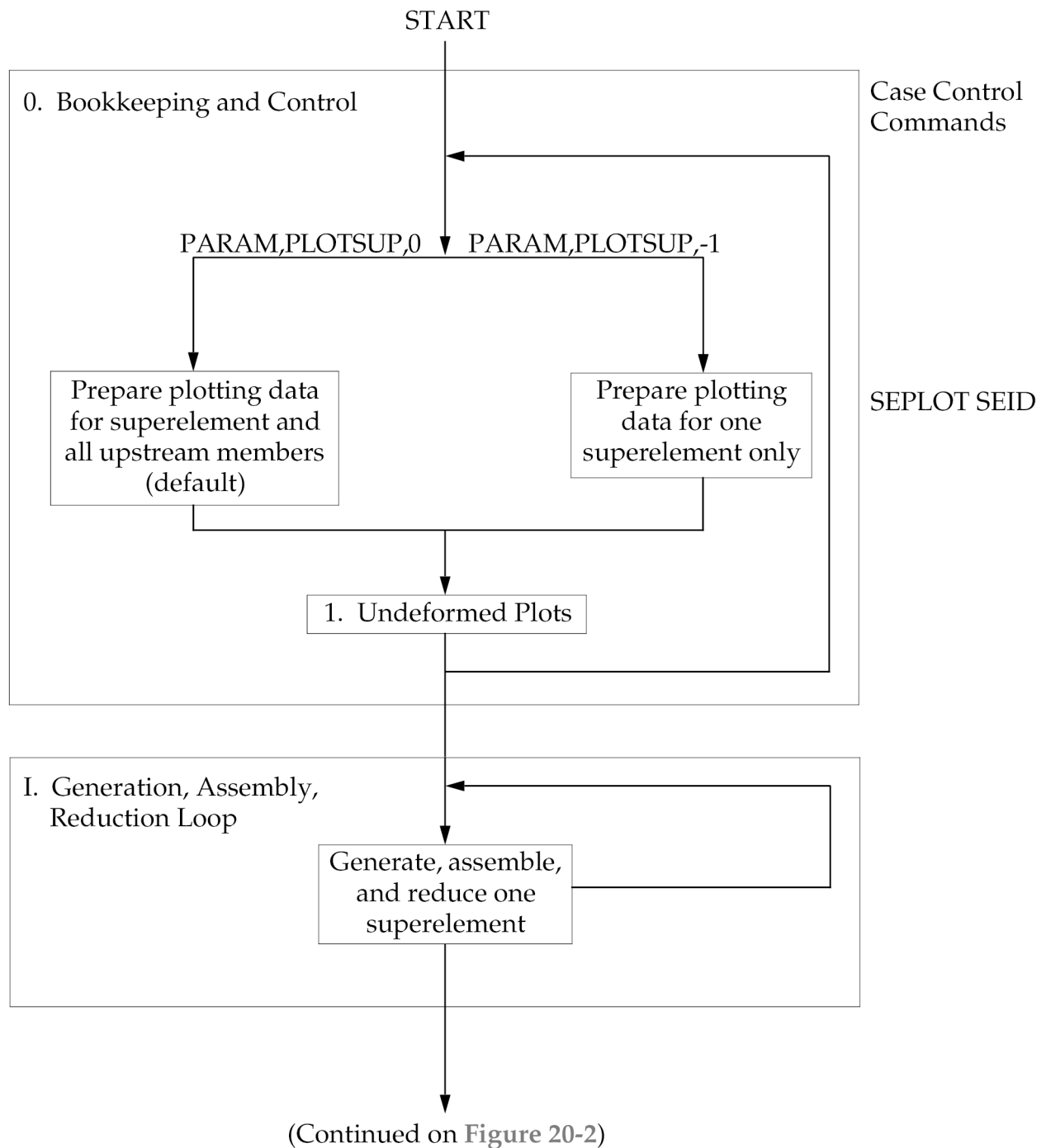
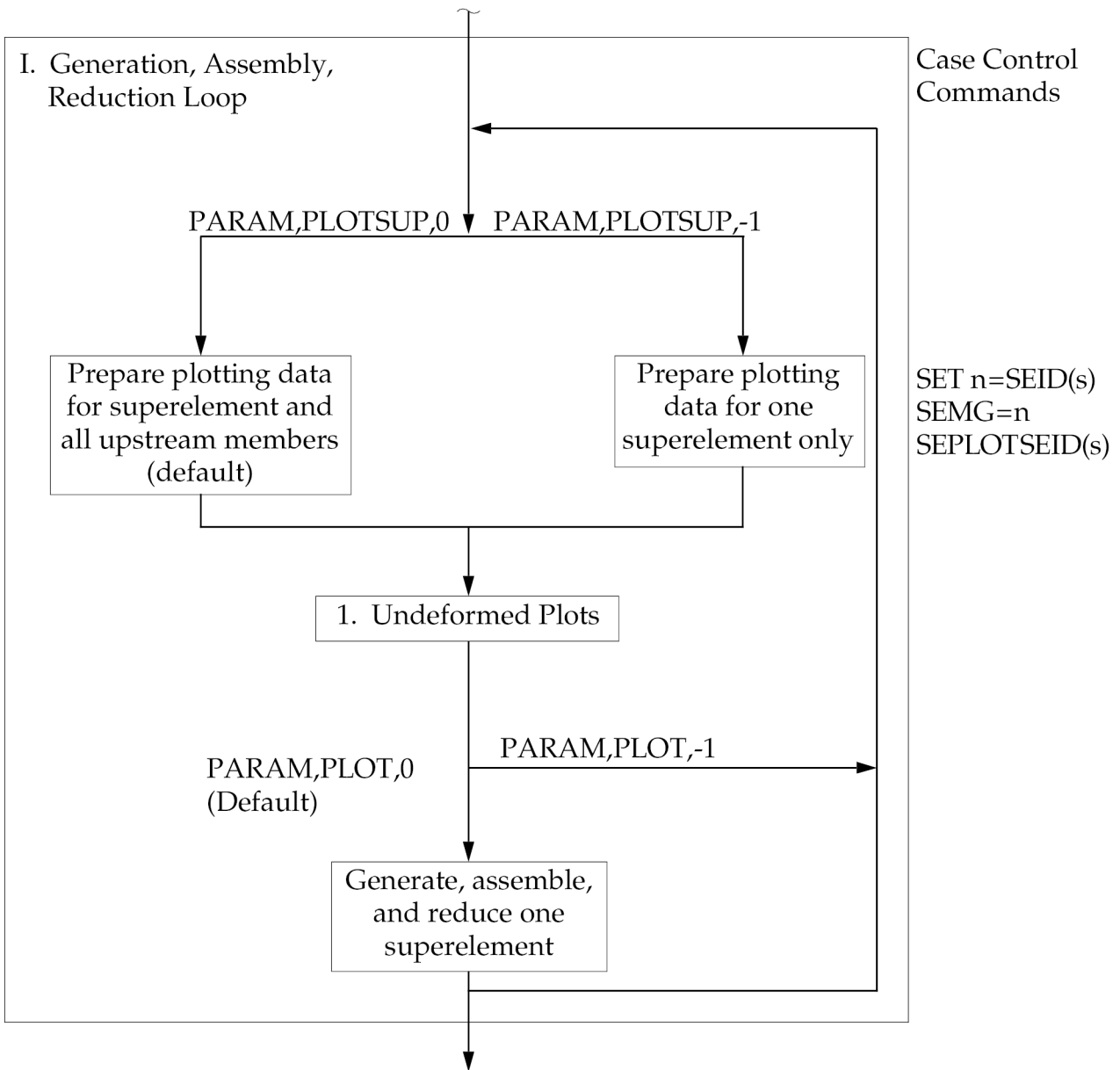


Figure 21-1. Phase 0 and I Superelement Plot Control In the Structured Solution Sequences



(Continued on Figure 20-3)

Figure 21-2. Phase I Superelement Plot Control In the Unstructured Solution Sequences

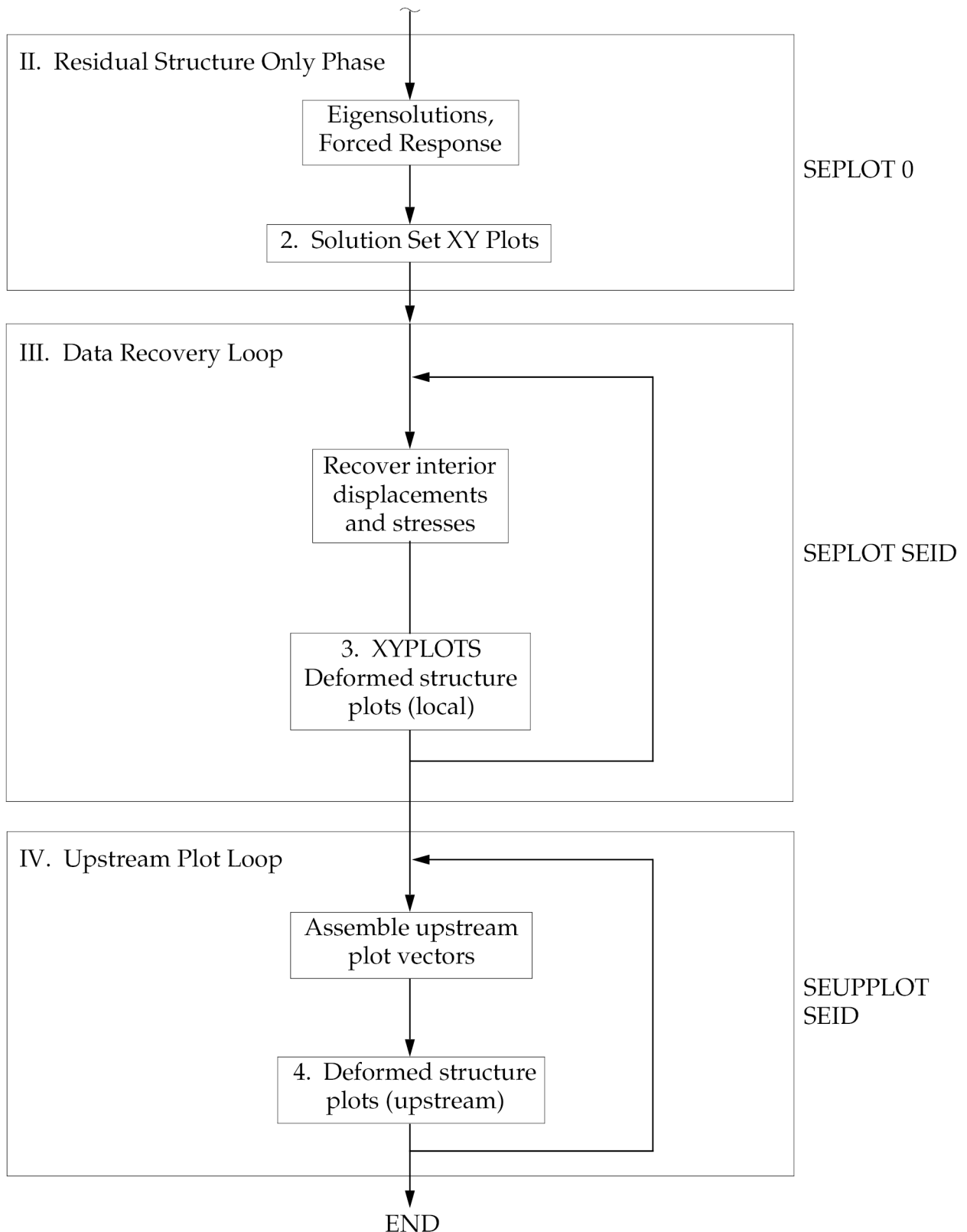


Figure 21-3. Phase II, III, and IV Superelement Plot Control in All Solution Sequences

21.3 Post Processors

The plotps post processor reads plotting commands from a single NX Nastran binary or neutral format plot file and produces a file which can be printed or viewed on a PostScript device. Each image will be oriented to best correlate the plot's page size with the printable page size (assumed to be 7.5 inches wide and 10.0 inches tall). Command line arguments can be specified in any order, command keywords can be abbreviated by truncation.

On Linux systems, you can omit the plot file type ('.plt' for binary files and '.neu' for neutral files).

Examples

- Translate a binary format plot file into PostScript:

```
plotps example.plt
```

- Translate a neutral format plot file into PostScript:

```
plotps example.neu
```

See also

- "PLOTPS" in the *NX Nastran Installation and Operations Guide*

Chapter 22: Linear Buckling

- *Introduction to Linear Buckling*
- *Finite Element Approach*
- *Eigenvalue Extraction Methods*
- *Linear Buckling Assumptions and Limitations*
- *Buckling Examples*

22.1 Introduction to Linear Buckling

In linear static analysis, a structure is normally considered to be in a state of stable equilibrium. As the applied load is removed, the structure is assumed to return to its original position. However, under certain combinations of loadings, the structure may become unstable. When this loading is reached, the structure continues to deflect without an increase in the magnitude of the loading. In this case, the structure has actually buckled or has become unstable; hence, the term “instability” is often used interchangeably with the term “buckling.”

Only linear buckling or elastic stability is considered in the discussion which follows; in other words, assume there is no yielding of the structure and the direction of the forces do not change (i.e., follower force effects are ignored). Other assumptions of elastic stability are discussed in “[Linear Buckling Assumptions and Limitations](#).” For a description of follower forces, see the chapter “Follower Stiffness” in this User's Guide.

This chapter is organized into the following sections:

- Finite element approach
- Eigenvalue extraction method
- Assumptions and limitations of linear buckling analysis
- Examples

22.2 Finite Element Approach

In finite element analysis, the problem of linear buckling is addressed by including the effect of the differential stiffness to the linear stiffness matrix. The differential stiffness results from including the higher-order terms of the strain-displacement relationships. From a physical standpoint, the differential stiffness matrix represents the linear approximation of softening (reducing) the linear stiffness matrix in the case of a compressive axial load, and stiffening (increasing) the linear stiffness matrix in the case of a tensile axial load. Since the differential stiffness either adds or subtracts from the linear stiffness, often it is also called the incremental stiffness. The element linear and differential stiffness matrices are denoted as $[k_a]_i$ and $[k_d]_i$, respectively.

The differential stiffness matrix is a function of the geometry, element type, and applied loads. A look at the differential stiffness matrix of a single planar bar element as shown in [Figure 22-1](#) shows how a linear buckling analysis is handled using a finite element approach. For clarity, only the y and θ_z degrees of freedom at each end are retained for this example.

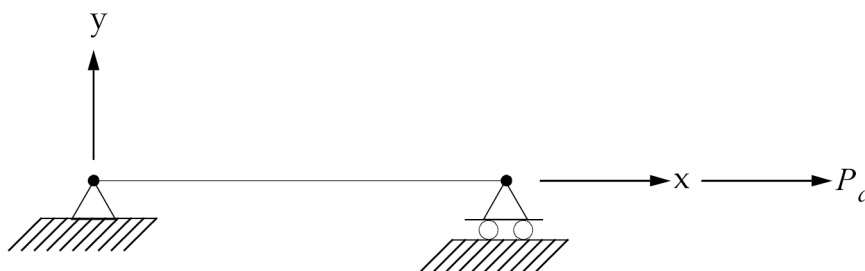


Figure 22-1. Single Planar Bar Element

From *The Nastran Theoretical Manual*, the differential stiffness for this planar bar element can be represented as

$$[k_d]_i = \begin{bmatrix} \frac{6F_{x_i}}{5l_i} & \frac{-F_{x_i}}{10} & \frac{-6F_{x_i}}{5l_i} & \frac{-F_{x_i}}{10} \\ \frac{-F_{x_i}}{10} & \frac{2l_iF_{x_i}}{15} & \frac{F_{x_i}}{10} & \frac{-l_iF_{x_i}}{30} \\ \frac{-6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} & \frac{6F_{x_i}}{5l_i} & \frac{F_{x_i}}{10} \\ \frac{-F_{x_i}}{10} & \frac{-l_iF_{x_i}}{30} & \frac{F_{x_i}}{10} & \frac{2l_iF_{x_i}}{15} \end{bmatrix}$$

Equation 22-1.

where F_{x_i} is the axial force in the CBAR element. In this case, P_a (applied load) = F_{x_i} since there is only one element in the model and the applied load is in line with the element axis. In the general case, F_{x_i} is proportional to P_a as long as the structure remains linear; in other words, if P_a is increased by scale factor a_i , then F_{x_i} also increases by the same scale factor. The value “i” stands for the i-th element. Note also that the differential stiffness matrix is dependent only on the element type, the applied forces, and the geometry of the structure. This is the reason why the differential stiffness is also often called the geometric stiffness matrix.

Eq. 22-1 can, therefore, be rewritten as follows:

$$[k_d]_i = P_a \begin{bmatrix} \frac{6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} & \frac{-6\alpha_i}{5l_i} & \frac{-\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{2l_i\alpha_i}{15} & \frac{\alpha_i}{10} & \frac{-l_i\alpha_i}{30} \\ \frac{-6\alpha_i}{5l_i} & \frac{\alpha_i}{10} & \frac{6\alpha_i}{5l_i} & \frac{\alpha_i}{10} \\ \frac{-\alpha_i}{10} & \frac{-l_i\alpha_i}{30} & \frac{\alpha_i}{10} & \frac{2l_i\alpha_i}{15} \end{bmatrix} = P_a [\bar{k}_d]_i$$

Equation 22-2.

One can view α_i as the distribution factor of the applied load to the i-th element. Each element in the structure that supports differential stiffness (see “[Linear Buckling Assumptions and Limitations](#)”) has an element differential stiffness matrix associated with it. Each of these differential stiffness

matrices has a scale factor similar to Eq. 22-2. In general, each α_i is different for each element in the structure. The value of each α_i depends on the element type, the orientation of the element relative to the overall structure, and the applied load.

The system linear stiffness matrix can then be represented as

$$[K_a] = \sum_i^n k_{a_i}$$

Similarly, the system differential stiffness matrix can be represented as

$$[K_d] = \sum_i^n k_{d_i}$$

In general, the individual $[k_d]_i$ is more complicated than Eq. 22-1; however, the concept is the same. The overall system stiffness matrix is represented by Eq. 22-3.

$$[K] = [K_a] + [K_d]$$

Equation 22-3.

The total potential energy is equal to

$$[U] = 0.5 \{u\}^T [K_a] \{u\} + 0.5 \{u\}^T [K_d] \{u\}$$

Equation 22-4.

In order for the system to achieve static equilibrium, the total potential must have a stationary value; in other words, the relationship in Eq. 22-5 must be satisfied:

$$\frac{\partial [U]}{\partial u_i} = [K_a] \{u\} + [K_d] \{u\} = \{0\}$$

Equation 22-5.

where u_i is the displacement of the i-th degree of freedom.

It is convenient to rewrite Eq. 22-5 as

$$[[K_a] + P_a[\bar{K}_d]]\{u\} = \{0\}$$

Equation 22-6.

where $[K_d] = P_a[\bar{K}_d]$ and P_a is the applied load. In order for Eq. 22-6 to have a non-trivial solution, the following relationship must be true:

$$|[K_a] + P_a[\bar{K}_d]| = \{0\}$$

Equation 22-7.

where $||$ stands for the determinant of the matrix. Eq. 22-7 is only satisfied for certain values of P_a . These values of are the critical buckling loads.

A real structure has an infinite number of degrees of freedom. The finite element model approximates the behavior of the structure with a finite number of degrees of freedom. The number of buckling loads obtainable for your finite element model is equal to the number of degrees of freedom of your model. In other words,

$$P_{cr_i} = \lambda_i \cdot P_a$$

Equation 22-8.

Eq. 22-7 can, therefore, be rewritten as

$$|[K_a] + \lambda_i[K_d]| = [0]$$

Equation 22-9.

Eq. 22-9 is in the form of an eigenvalue problem. Once you obtain the eigenvalues λ_i , the buckling loads can then be obtained using Eq. 22-8. The values λ_i are the scale factors by which the applied load P_a is multiplied to produce the critical buckling loads P_{cr_i} . As you can see from Eq. 22-8, the magnitude of the applied load P_a is arbitrary for arriving at the correct P_{cr_i} . As an example, if P_a is increased by a factor of 10, then the calculated λ_i values in Eq. 22-9 are reduced by a factor of 10; in other words, their resulting products P_{cr_i} remain the same.

In general, only the lowest buckling load is of any practical interest. The structure will fail prior to reaching any of the higher buckling loads.

Solution 105

In NX Nastran you can solve a linear buckling problem by using Solution 105 and following the procedure listed below.

1. Apply the static loads to the first n subcases (n is usually equal to one) and treat them as static analysis. The distribution of element forces due to these applied loads is generated internally. The actual magnitude of these applied loads is not critical.
2. You can perform buckling analysis on any or all of the loading conditions used in Step 1. One additional subcase is needed for each buckling analysis.
3. The $n+1$ to the $n+m$ subcases must each request an eigenvalue method from the Bulk Data Section to solve the eigenvalue problem shown in [Eq. 22-9](#). In this case, m is equal to the number of buckling analyses that you want to perform. Each buckling subcase may call out a unique eigenvalue solution.
4. The differential stiffness matrix is automatically generated for each element that supports differential stiffness. See "[Linear Buckling Assumptions and Limitations](#)" for a list of elements that support differential stiffness.
5. You must then multiply the eigenvalues obtained in Step 3 by the appropriate applied loads to obtain the buckling loads ([Eq. 22-8](#)) for each buckling analysis.
6. Each subcase may have a different boundary condition.

A typical input file used to calculate the buckling loads is shown in [Listing 22-1](#). In most applications, only one static and one buckling analysis is performed per run. Example 6 in "[Buckling Examples](#)" contains an application of multiple static and buckling analyses.

```

$
SOL 105
TIME 10
CEND
TITLE = SAMPLE INPUT FOR BUCKLING ANALYSIS
SPC = 10
DISP = ALL
$
$ STATIC SUBCASE TO GENERATE INTERNAL ELEMENT FORCES
$
SUBCASE 1
LOAD = 10
$
$ EIGENVALUE CALCULATION TO OBTAIN SCALE FACTORS BY WHICH
$ THE APPLIED LOAD(S) IS MULTIPLIED BY
$
SUBCASE 2
METHOD = 20
$
BEGIN BULK
$
$ APPLIED LOAD IN THE STATIC SUBCASE - SUBCASE ONE
$
FORCE,10,100,, -100.,1.0,0.,0.
$
$ LANCZOS EIGENVALUE METHOD REQUESTING THE LOWEST EIGENVALUE
$
EIGRL,20,,,1
$
$ BRING IN THE REST OF THE BULK DATA ENTRIES
$
include 'bulk.dat'
$
ENDDATA

```

Listing 22-1. Input File for a Typical Buckling Analysis

You should also adhere to the following guidelines when solving a linear buckling problem in NX Nastran.

- The Case Control Section must contain at least two subcases.
- A METHOD command must appear in each buckling subcase to select the appropriate eigenvalue extraction method (EIGRL or EIGB entry) from the Bulk Data Section. If there are two or more buckling subcases, then a STATSUB = x command must be placed in each buckling subcase to select the appropriate static subcase. If there is only one buckling subcase, then the STATSUB = x command is optional if x references the first static subcase ID.
- A static loading condition must be defined with a LOAD, TEMP(LOAD), or DEFORM selection unless all loading is specified by grid point displacements on SPC entries. All static subcases must be placed before the first buckling subcase.
- Correct SPC sets must be selected for all subcases.
- Output requests that apply to only a particular subcase must be placed inside that subcase.
- Output requests that apply to all subcases may be placed above the subcase level.
- Some type of output request (e.g., disp(plot) = all) must be requested for at least one subcase.

“**Buckling Examples**” contains six example problems that illustrate some of these guidelines.

22.3 Eigenvalue Extraction Methods

A number of methods of real eigenvalue extraction are available in NX Nastran:

- Householder
- Modified Householder
- Enhanced inverse power
- Lanczos

Two of these methods can be used for linear buckling analysis:

- Enhanced inverse power
- Lanczos

A brief description of each eigenvalue method is presented in the following sections. For further details, see the *NX Nastran Numerical Methods User's Guide*.

Enhanced Inverse Power Method (SINV)

This method uses Sturm sequence logic to ensure that all modes are found within the specified eigenvalue range. The Sturm sequence informs you of the number of modes below each trial eigenvalue. See the *NX Nastran Numerical Methods User's Guide* and the *NX Nastran Basic Dynamic Analysis User's Guide* for further details regarding the Sturm sequence check.

Lanczos Method

The Lanczos method overcomes the limitations and combines the best features of the other methods. It is efficient, and if an eigenvalue cannot be extracted within the range that you specify, a diagnostic message is issued. This method computes accurate eigenvalues and eigenvectors. Furthermore, it prints meaningful user diagnostics and supports parallel processing computers.

Comparison of Methods

The best method for a particular model depends on four factors:

- The size of the model (the total number of degrees of freedom)
- The number of eigenvalues desired
- The available real memory on your computer
- How well you can estimate the range of your eigenvalues

The enhanced inverse power method (SINV) can be a good choice if the model is too large to fit into memory, only a few modes are needed, and you have a reasonable idea of your eigenvalue range of interest. It is useful for models in which only the lowest few modes are desired. This method is also useful as a backup method to verify the accuracy of other methods.

For medium to large models, the Lanczos method is the recommended method. Furthermore, the Lanczos method takes full advantage of sparse matrix methods that can substantially increase

computational speed and reduce disk space usage. For overall robustness, the Lanczos method is the recommended method.

User Interface

A METHOD command is required in a subcase of the Case Control Section to select the appropriate eigenvalue extraction method in the Bulk Data Section. The Bulk Data entry is different depending on whether you are using the enhanced inverse power (SINV), or Lanczos method. The EIGRL entry is used for the Lanczos method, and the EIGB entry is used for the SINV methods.

EIGRL Format

The EIGRL entry has the following format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND					

The SID field is the set identification number, which is referenced by the METHOD command in the Case Control Section. The V1 field defines the lower eigenvalue bound λ_l , and the V2 field defines the upper eigenvalue bound λ_u . The ND field specifies the number of roots desired. [Table 22-1](#) summarizes the action NX Nastran takes depending on the values specified for V1, V2, and ND.

Table 22-1. Number and Type of Roots Found with EIGRL Entry				
Case	V1	V2	ND	Number and Type of Roots Found
1	x	x	x	Lowest ND or all in range, whichever is smaller
2	x	x		All in range
3	x		x	Lowest ND in range $[V1, +\infty]$
4	x			Lowest root in range $[V1, +\infty]$
5			x	Lowest ND roots in $[-\infty, +\infty]$
6				Lowest root
7		x	x	Lowest ND roots below V2
8		x		All below V2

EIGRL Example

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

The above example selects the Lanczos method requesting the lowest root for your model.

In order for this entry to be used, the METHOD = 10 command must be specified in the Case Control Section.

EIGB Format

The EIGB entry has the following format:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP				

The SID field is the set identification number, which is referenced by the METHOD command in the Case Control Section. The METHOD field selects the desired eigenvalue extraction method (SINV). The L1 field defines the lower eigenvalue bound λ_l , and the L2 field defines the upper eigenvalue bound λ_u .

EIGB Example

1	2	3	4	5	6	7	8	9	10
EIGB	10	SINV	0.5	1.2					

The above example selects the enhanced inverse power method. All eigenvalues between 0.5 and 1.2 are desired. In order for this entry to be used, the METHOD = 10 command must be specified in the Case Control Section.

22.4 Linear Buckling Assumptions and Limitations

The following assumptions and limitations apply to linear buckling analysis:

1. The deflections must be small.
2. The element stresses must be elastic.
3. The differential stiffness is supported for the following elements: CONROD, CROD, CTUBE, CBAR, CBEAM, CBEND, CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, CTRAX3, CTRAX6, CQUADX4, CQUADX8, CHEXA, CPENTA, CPYRAM, and CTETRA.
4. A minimum of five grid points per half sine wave (buckled shape) is recommended.
5. The distribution of the internal element forces due to the applied loads remains constant.
6. Offsets should not be used in beam, plate, or shell elements (except CQUADR/CTRIAR) for buckling analysis.
7. For 3-D buckling problems, the use of PARAM,K6ROT is recommended for CQUAD4 and CTRIA3 elements. A value of 100 is recommended.
8. For structures that exhibit nonlinear material or large deflection deformations, the linear buckling load obtained from Solution 105 may be different than the actual buckling load. For structures with significant nonlinearities, it is recommended that you perform a nonlinear buckling analysis using Solution 106.

See also

- [“Performing Nonlinear Buckling Analysis”](#) in the *NX Nastran Basic Nonlinear Analysis User's Guide*

22.5 Buckling Examples

This section contains six example problems pertaining to buckling. Each problem is chosen to illustrate either a particular type of failure mode, the effect of an element type, the effect of a structural configuration, or the method of performing multiple buckling analyses in a single run. The examples are as follows:

1. Classical Euler beam buckling.
2. Lateral buckling.
3. Planar frame buckling.
4. Buckling of a stiffened panel with transverse shear flexibility.
5. Buckling of a cylinder under uniform axial load.
6. Multiple buckling analyses in a single run.

In each example, the results produced by NX Nastran are compared with known theoretical solutions.

Example 1 – Classical Euler Beam Buckling

Consider a classical Euler buckling problem. This problem is shown in [Figure 22-2](#).

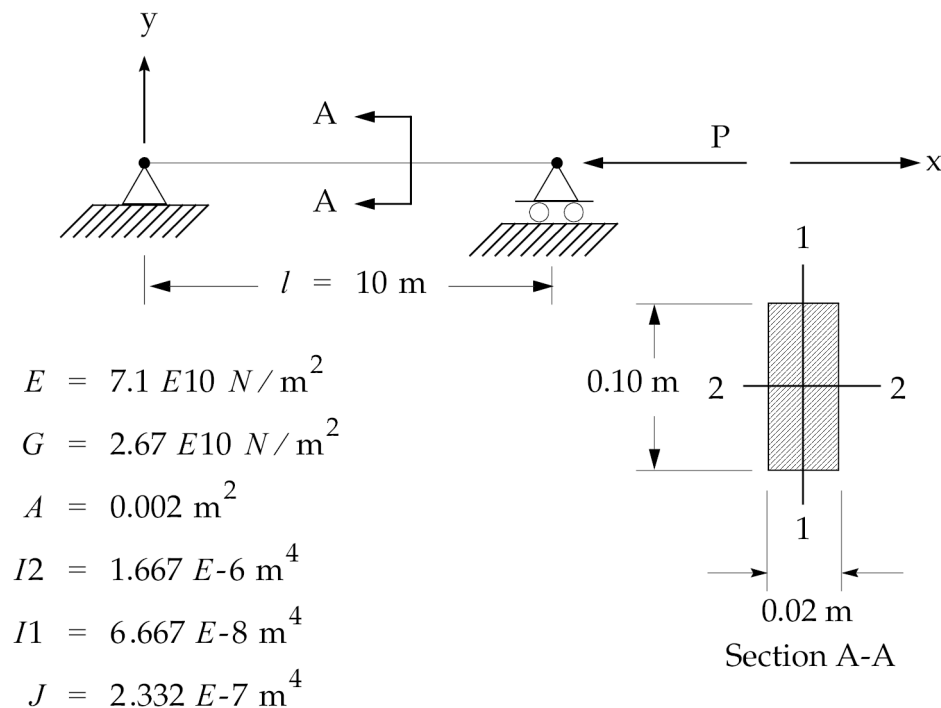


Figure 22-2. Euler Beam

The initial buckling occurs in the xz plane since $I1$ is smaller than $I2$.

NX Nastran Results for Example 1

The input file used for this problem is shown in [Listing 22-2](#). In this case, a 1000 N compressive load is applied at the roller end to generate internal loads for the structure. The Lanczos eigenvalue extraction method is used to request the first five modes. Either the CBAR or CBEAM elements can be chosen for this problem; the CBAR elements are used in this case.

```

$  FILENAME - bukeuler.dat
ID EULER BEAM
SOL 105
TIME 10
CEND
$
TITLE = EULER BEAM
SUBTITLE = METRIC UNITS
SPC = 10
DISP = ALL
$
SUBCASE 1
LOAD = 10
$
SUBCASE 2
METHOD = 10
$
BEGIN BULK
$
PARAM POST 0
EIGRL 10 5
FORCE 10 11 -1000. 1. 0. 0.
$
CBAR 1 1 1 2 100
CBAR 2 1 2 3 100
CBAR 3 1 3 4 100
CBAR 4 1 4 5 100
CBAR 5 1 5 6 100
CBAR 6 1 6 7 100
CBAR 7 1 7 8 100
CBAR 8 1 8 9 100
CBAR 9 1 9 10 100
CBAR 10 1 10 11 100
$
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
GRID 3 2. 0. 0.
GRID 4 3. 0. 0.
GRID 5 4. 0. 0.
GRID 6 5. 0. 0.
GRID 7 6. 0. 0.
GRID 8 7. 0. 0.
GRID 9 8. 0. 0.
GRID 10 9. 0. 0.
GRID 11 10. 0. 0.
GRID 100 0. 0. 10.
$
MAT1 1 7.1+10 .33 2700.
$
$ RECTANGULAR SECTION OF DIMENSION .1m x .02m
$
PBAR 1 1 .002 6.667-8 1.667-6 2.332-7
$
SPC1 10 123 1
SPC1 10 23 11
SPC 10 1 4
ENDDATA

```

Listing 22-2. Input File for Euler Beam

As indicated on the EIGRL entry, the first five modes are requested for this problem, and the eigenvalue table is shown in [Figure 22-3](#). The first eigenvalue λ_1 in this case is equal to 0.4672, while the applied load in SUBCASE 1 is equal to -1000 N. Therefore, the lowest buckling load is equal to

$$P_1 = (\lambda_1)(P_a) = (.46719)(-1000) = -467.2 \text{ N}$$

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	4.671909E-01	6.835136E-01	1.087846E-01	4.934669E+02	2.305432E+02
2	2	1.869135E+00	1.367163E+00	2.175907E-01	2.181382E+03	4.077298E+03
3	3	4.209068E+00	2.051601E+00	3.265225E-01	4.431894E+03	1.865414E+04
4	4	7.498919E+00	2.738415E+00	4.358323E-01	4.971707E+03	3.728243E+04
5	5	1.168153E+01	3.417825E+00	5.439637E-01	4.934669E+02	5.764446E+03

Figure 22-3. Eigenvalue Table for a Euler Beam

A partial output that contains the first, second, and fifth eigenvectors is shown in [Figure 22-4](#). The mode shapes for the first five modes are plotted in [Figure 22-5](#).

EIGENVALUE = 4.671909E-01

POINT ID.	TYPE	R E A L E I G E N V E C T O R N O .						
		T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	-3.141592E-01	-8.436650E-13	
2	G	2.344155E-14	-3.535500E-13	3.090170E-01	3.617327E-13	-2.987832E-01	5.605364E-13	
3	G	2.295983E-14	1.997131E-13	5.877852E-01	-5.413820E-14	-2.541602E-01	-5.748283E-13	
4	G	3.762660E-14	-8.750774E-15	8.090170E-01	-8.143186E-13	-1.846582E-01	-5.513294E-13	
5	G	5.222389E-14	1.995187E-14	9.510565E-01	2.312171E-13	-9.708054E-02	1.283237E-14	
6	G	6.208775E-14	-6.846741E-14	1.000000E+00	-2.349688E-13	6.485303E-13	-2.091279E-13	
7	G	9.229606E-14	1.712600E-13	9.510565E-01	-1.026823E-12	9.708054E-02	1.438084E-12	
8	G	8.977146E-14	-7.130678E-14	8.090170E-01	7.968065E-14	1.846582E-01	-5.453334E-13	
9	G	6.058189E-14	1.523950E-14	5.877852E-01	1.248126E-13	2.541602E-01	1.957519E-12	
10	G	8.648088E-14	-1.169763E-13	3.090170E-01	5.284149E-13	2.987832E-01	-1.513853E-12	
11	G	9.207012E-14	.0	.0	-1.129664E-12	3.141592E-01	2.010604E-12	
100	G	.0	.0	.0	.0	.0	.0	

EIGENVALUE = 1.869135E+00

POINT ID.	TYPE	R E A L E I G E N V E C T O R N O .						
		T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	-6.606488E-01	1.696698E-11	
2	G	-5.494490E-13	8.408953E-12	6.180340E-01	-8.098147E-12	-5.344761E-01	-1.038079E-11	
3	G	-5.359018E-13	-4.648322E-12	1.000000E+00	1.386206E-12	-2.041517E-01	1.248417E-11	
4	G	-8.844888E-13	-1.029370E-13	1.000000E+00	1.719914E-11	2.041517E-01	1.241602E-11	
5	G	-1.226278E-12	-3.770107E-13	6.180340E-01	-5.748548E-12	5.344761E-01	-2.901899E-12	
6	G	-1.454765E-12	1.844125E-12	-1.042989E-12	3.660258E-12	6.606488E-01	5.452428E-12	
7	G	-2.164086E-12	-4.046730E-12	-6.180340E-01	2.005433E-11	5.344761E-01	-3.277508E-11	
8	G	-2.103291E-12	1.503905E-12	-1.000000E+00	-2.061866E-12	2.041517E-01	1.278820E-11	
9	G	-1.422088E-12	-3.211627E-13	-1.000000E+00	-3.318442E-12	-2.041517E-01	-4.639094E-11	
10	G	-2.025011E-12	2.810148E-12	-6.180340E-01	-1.061518E-11	-5.344761E-01	3.454609E-11	
11	G	-2.158100E-12	.0	.0	2.222183E-11	-6.606488E-01	-4.772200E-11	
100	G	.0	.0	.0	.0	.0	.0	

EIGENVALUE = 1.168152E+01

		R E A L E I G E N V E C T O R N O .						5
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	2.954037E-12	3.141592E-01	
2	G	-1.530020E-14	3.090170E-01	-4.216155E-12	-8.584071E-11	6.817444E-12	2.987832E-01	
3	G	-3.407930E-14	5.877852E-01	-1.068567E-11	8.455609E-11	4.070235E-12	2.541602E-01	
4	G	-4.763809E-14	8.090170E-01	-7.933438E-12	-6.013640E-11	-8.832388E-12	1.846582E-01	
5	G	-5.124878E-14	9.510565E-01	3.385932E-12	-8.368475E-11	-1.058233E-11	9.708054E-02	
6	G	-2.379340E-14	1.000000E+00	1.107325E-11	8.325122E-11	-5.506725E-12	2.671122E-13	
7	G	-4.143882E-15	9.510565E-01	1.763771E-11	-4.475530E-11	-8.492263E-12	-9.708054E-02	
8	G	-2.010117E-15	8.090170E-01	2.572503E-11	-1.136649E-11	-4.849835E-12	-1.846582E-01	
9	G	3.640430E-15	5.877852E-01	2.352012E-11	-2.112232E-10	9.559306E-12	-2.541602E-01	
10	G	-1.566881E-14	3.090170E-01	1.092055E-11	5.283983E-11	1.326611E-11	-2.987832E-01	
11	G	-2.601413E-14	.0	.0	1.037907E-10	9.724306E-12	-3.141592E-01	
100	G	.0	.0	.0	.0	.0	.0	

Figure 22-4. Eigenvectors for an Euler Beam

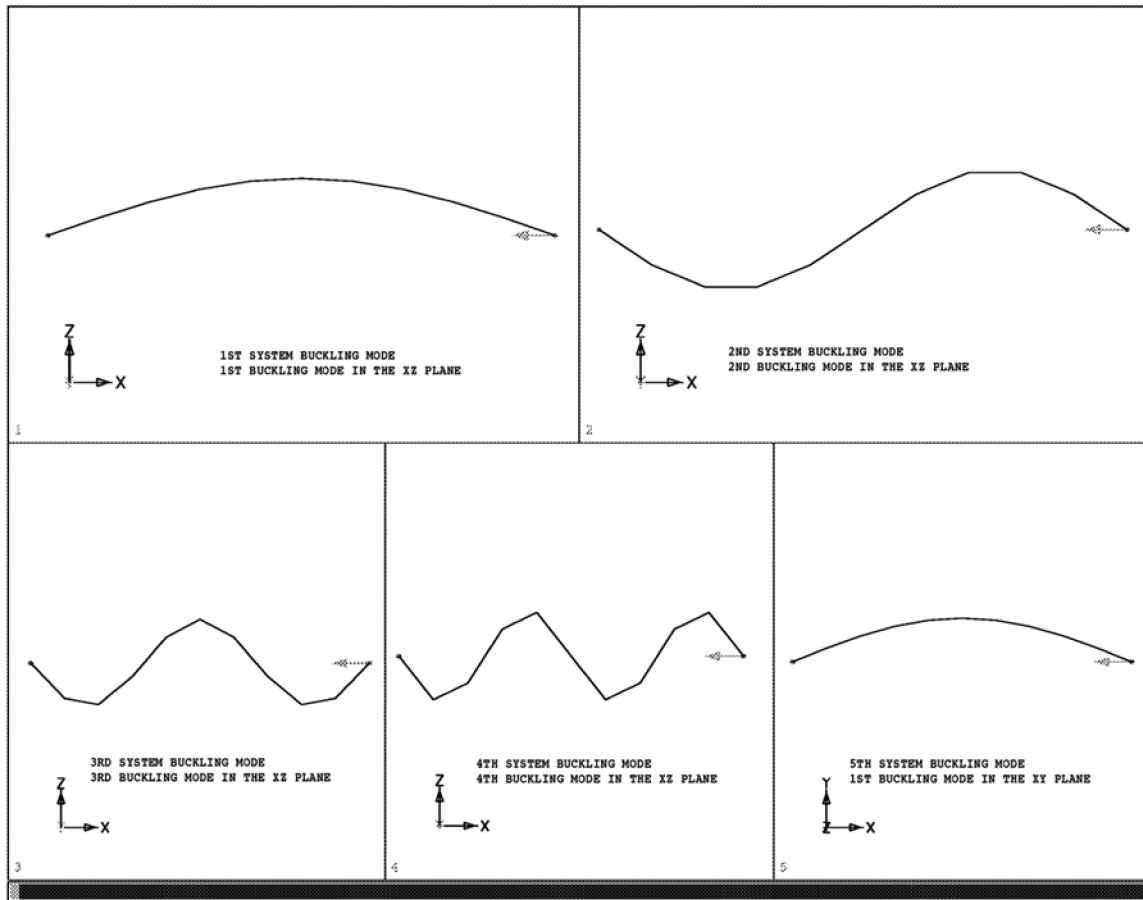


Figure 22-5. Mode Shapes for an Euler Beam

Note that the first four system buckling modes are in the xz plane and the first buckling mode in the xy plane is the fifth overall system mode. The lowest buckling load in the xy plane is, therefore, equal to

$$P_5 = (\lambda_5)(P_a) = (11.6815)(-1000) = -11681.5 \text{ N}$$

Equation 22-10.

Theoretical Results for Example 1

This Euler buckling load is calculated as follows (S.P. Timoshenko and J.M. Gere, *Theory of Elastic Stability*, Engineering Societies Monograph Series, Second Ed., 1961):

$$P_1 = \frac{\pi^2 EI_2}{l^2} = \pi^2 (7.1 \cdot 10^{10}) (6.667 \cdot 10^{-8}) / 100 = 467.2 \text{ N}$$

For this particular problem, the buckling load in the xy plane is equal to

$$P_5 = \frac{\pi^2 EI_2}{l^2} = \pi^2 (7.1 \cdot 10^{10}) (1.667 \cdot 10^{-6}) / 100 = 11681.4 \text{ N}$$

Table 22-2 compares the NX Nastran results with theoretical results.

Table 22-2. Euler Beam Results Comparison			
Mode Number	NX Nastran (N)	Theoretical (N)	% Difference
1 (XZ Plane)	467.2	467.2	0
5 (XY Plane)	11681.5	11681.4	0.001

As you approach the higher modes, the mode shapes become less smooth. This effect occurs because for the same mesh density, the higher modes have fewer grid points per buckled sine wave than the lower modes. If you are interested in these mode shapes, you should increase your model mesh density to follow the general guideline of maintaining a minimum of five grid points per half sine wave. However, these higher failure modes are seldom of interest in real world applications.

Example 2 – Lateral Buckling

The problem shown in Figure 22-6 illustrates an interesting case of lateral buckling. The model consists of a cantilever beam with a vertical tip load applied at the free end. The dimensions of the beam are identical to Example 1. In this example, the structure buckles in the lateral direction (xz plane) perpendicular to the applied load rather than in the plane of the applied load (xy plane).

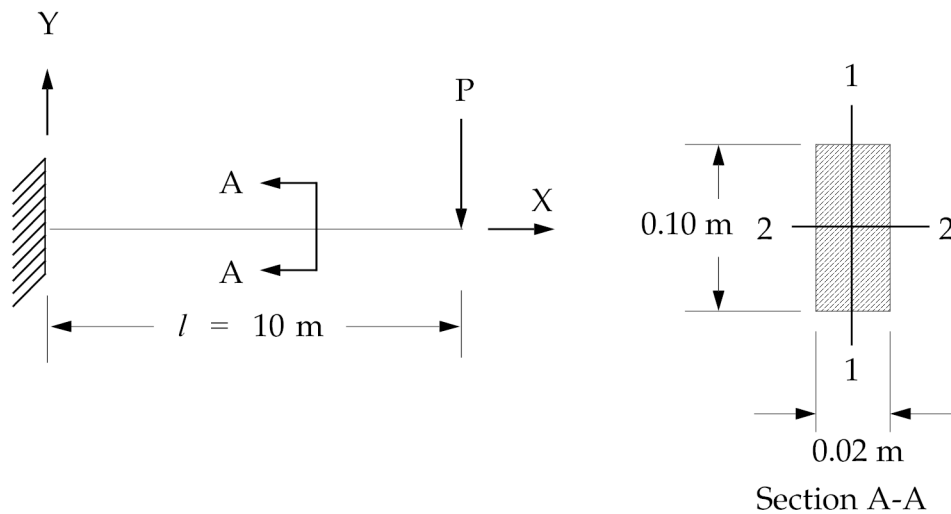


Figure 22-6. Lateral Buckling

NX Nastran Results for Example 2

Listing 22-3 contains the corresponding NX Nastran model for this lateral buckling problem. The CBEAM element is used in this case, although the CBAR element can also be used since this beam consists of a constant cross section. A vertical 1000 N load is applied in the negative y-direction to generate the element internal loads in Subcase 1. Once again, the Lanczos eigenvalue extraction method is employed requesting two modes. **Figure 22-8** contains the eigenvalue table and corresponding eigenvectors. The static deflection and buckling mode shapes are shown in **Figure 22-10**.

It is interesting to note that the static deflection is in the plane of the applied load (xy plane), while the buckling mode shape is in the plane perpendicular to the applied load (xz plane). Two modes were requested on the EIGRL entry to illustrate another subtle point for this problem. The two eigenvalues are identical in magnitude but different in sign. What is the significance of a negative eigenvalue in a buckling analysis? In buckling analysis, a negative eigenvalue implies that the load that causes the structure to buckle is opposite to the direction of the applied load.

Since both the structure and loading are symmetric, it does not matter whether the applied load is in the +y or -y direction. This fact brings up another subtle point. If you specify V2 only on the EIGRL entry, then NX Nastran attempts to extract all the roots below V2. In this case, these roots include all the negative eigenvalues. For a large model, calculating a large number of modes can be expensive especially when you are only interested in the lowest modes. Note that these negative roots are legitimate both from a physical and mathematical standpoint. In general, this situation only occurs if the applied buckling load is in the opposite direction of the buckling load. For this particular problem, there is no right or wrong direction. Therefore, in order to avoid this type of surprise, it is best to request the lowest number of desired roots (ND) on the EIGRL entry when you are not sure of the buckling load direction.


```

$  FILENAME - bucklat.dat
SOL 105
TIME 10
CEND
$
TITLE = LATERAL BUCKLING OF CANTILEVER BEAM
SUBTITLE = METRIC UNITS
SPC = 10
DISP = ALL
$
SUBCASE 1
LOAD = 10
$
SUBCASE 2
METHOD = 10
$
BEGIN BULK
$
EIGRL  10                      2
$
FORCE  10      11      -1000.  0.      1.      0.
$
CBEAM  1      1      1      2      100
CBEAM  2      1      2      3      100
CBEAM  3      1      3      4      100
CBEAM  4      1      4      5      100
CBEAM  5      1      5      6      100
CBEAM  6      1      6      7      100
CBEAM  7      1      7      8      100
CBEAM  8      1      8      9      100
CBEAM  9      1      9     10      100
CBEAM 10      1     10     11      100
$
GRID   1          0.      0.      0.
GRID   2          1.      0.      0.
GRID   3          2.      0.      0.
GRID   4          3.      0.      0.
GRID   5          4.      0.      0.
GRID   6          5.      0.      0.
GRID   7          6.      0.      0.
GRID   8          7.      0.      0.
GRID   9          8.      0.      0.
GRID  10          9.      0.      0.
GRID  11         10.      0.      0.
GRID  100         0.      0.     10.
$
MAT1   1      7.1+10      .33     2700.
$
$  RECTANGULAR SECTION OF DIMENSION .1m x .02m
PBEAM 1 1 .002 6.667-8 1.667-6 2.332-7
$
SPC1   10      123456  1
ENDDATA

```

Listing 22-3. Input File for Lateral Buckling

R E A L E I G E N V A L U E S						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	2.183456E-01	4.672747E-01	7.436907E-02	1.749877E+02	3.820780E+01
2	2	-2.183456E-01	4.672747E-01	7.436907E-02	-1.749877E+02	3.820780E+01
EIGENVALUE = 2.183456E-01						
R E A L E I G E N V E C T O R N O . 1						
POINT ID.	TYPE	T1	T2	T3	R1	R2 R3
1	G	.0	.0	.0	.0	.0 .0
2	G	2.537741E-11	-3.892257E-09	2.499774E-03	3.424457E-02	-7.371472E-03 -4.395732E-09
3	G	-1.277730E-11	-1.073776E-08	1.867549E-02	6.424069E-02	-2.656322E-02 -1.484095E-08
4	G	1.189500E-10	-2.919947E-08	5.795542E-02	8.779415E-02	-5.277112E-02 -1.207688E-08
5	G	8.478357E-11	-2.620355E-08	1.250618E-01	1.045299E-01	-8.153865E-02 1.648216E-08
6	G	1.273718E-10	-2.596014E-09	2.207153E-01	1.152518E-01	-1.093766E-01 2.570351E-08
7	G	9.853920E-11	2.034603E-08	3.427072E-01	1.213348E-01	-1.339076E-01 2.049627E-08
8	G	8.265020E-11	4.163538E-08	4.869550E-01	1.242675E-01	-1.537219E-01 2.487351E-08
9	G	1.443917E-10	6.310452E-08	6.483413E-01	1.253649E-01	-1.681112E-01 8.187754E-09
10	G	1.136984E-10	4.827729E-08	8.212749E-01	1.256189E-01	-1.767931E-01 -3.784262E-08
11	G	1.540406E-10	-4.905004E-09	1.000000E+00	1.256360E-01	-1.796903E-01 -6.074886E-08
100	G	.0	.0	.0	.0	.0 .0
EIGENVALUE = -2.183456E-01						
R E A L E I G E N V E C T O R N O . 2						
POINT ID.	TYPE	T1	T2	T3	R1	R2 R3
1	G	.0	.0	.0	.0	.0 .0
2	G	7.576565E-11	2.472828E-09	2.499775E-03	-3.424457E-02	-7.371472E-03 -3.871140E-10
3	G	-4.478584E-12	-3.649991E-09	1.867549E-02	-6.424069E-02	-2.656322E-02 -9.834048E-09
4	G	1.762648E-11	-1.338523E-08	5.795542E-02	-8.779415E-02	-5.277112E-02 -7.743097E-09
5	G	-7.876090E-11	-1.461366E-08	1.250618E-01	-1.045299E-01	-8.153865E-02 8.549041E-09
6	G	-1.393435E-10	8.698958E-10	2.207153E-01	-1.152518E-01	-1.093766E-01 1.680920E-08
7	G	-9.326392E-11	6.915506E-09	3.427072E-01	-1.213349E-01	-1.339076E-01 -1.117986E-08
8	G	-1.053673E-10	-1.124932E-08	4.869550E-01	-1.242675E-01	-1.537219E-01 -7.835656E-09
9	G	-8.121134E-11	-4.722858E-09	6.483413E-01	-1.253649E-01	-1.681112E-01 7.580319E-09
10	G	-1.725437E-10	-1.078508E-08	8.212749E-01	-1.256188E-01	-1.767931E-01 -2.140314E-08
11	G	-1.482953E-10	-4.360446E-08	1.000000E+00	-1.256359E-01	-1.796903E-01 -3.865582E-08
100	G	.0	.0	.0	.0	.0 .0

Figure 22-7. Eigenvalue Table and Eigenvectors for Lateral Beam Buckling

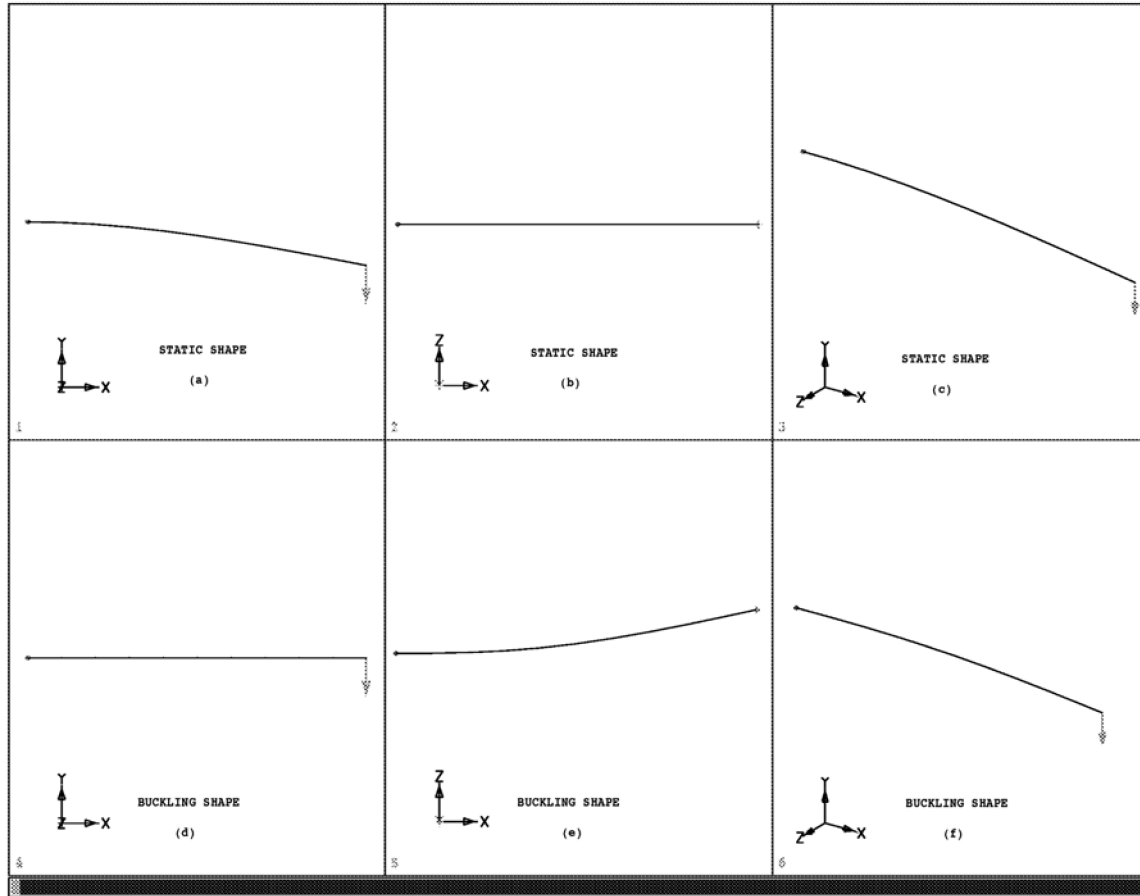


Figure 22-8. Static and Buckling Shapes

The lowest buckling load calculated by NX Nastran is

$$P_{cr_1} = \pm 218(1000) = \pm 218N$$

Theoretical Results for Example 2

The theoretical buckling load can be calculated as follows (S.P. Timoshenko and J.M. Gere, *Theory of Elastic Stability*, Engineering Societies Monograph Series, Second Ed., 1961):

$$\begin{aligned}
 P_{cr} &= \frac{4.013}{l^2} \sqrt{EIGJ} \\
 &= \frac{4.013}{100} \sqrt{7.1 \cdot 10^{10} (6.667 \cdot 10^{-8}) (2.67 \cdot 10^{10}) (2.332 \cdot 10^{-7})} \\
 &= 218 \text{ N}
 \end{aligned}$$

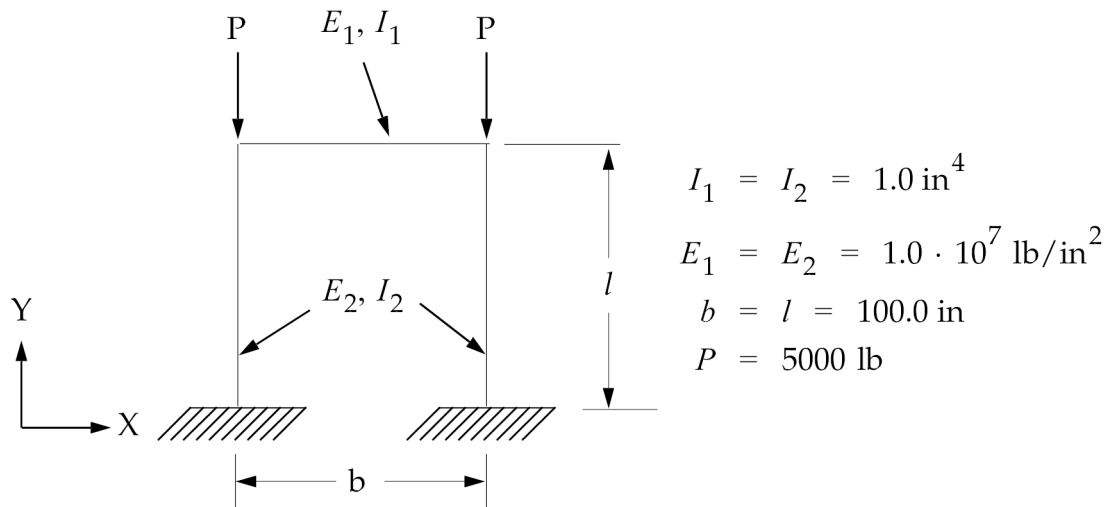
Table 22-3 compares the NX Nastran results to the theoretical results.

Table 22-3. Lateral Buckling Results Comparison

NX Nastran (N)	Theoretical (N)	% Difference
218	218	0

Example 3 – Planar Frame Buckling

This example is a planar frame subjected to a compressive load at the top as shown in [Figure 22-9](#). At first glance, since both the structure and loading are symmetrical, you may be tempted to conclude that the structure will buckle in a symmetric mode. As illustrated in [Figure 22-10\(a\)](#) (from NX Nastran) and S. P. Timoshenko and J. M. Gere, *Theory of Elastic Stability*, 1961, this isn't the case. The lowest buckling mode is actually antisymmetric. The second buckling mode is symmetric ([Figure 22-10\(b\)](#)).

**Figure 22-9. Planar Frame Buckling**

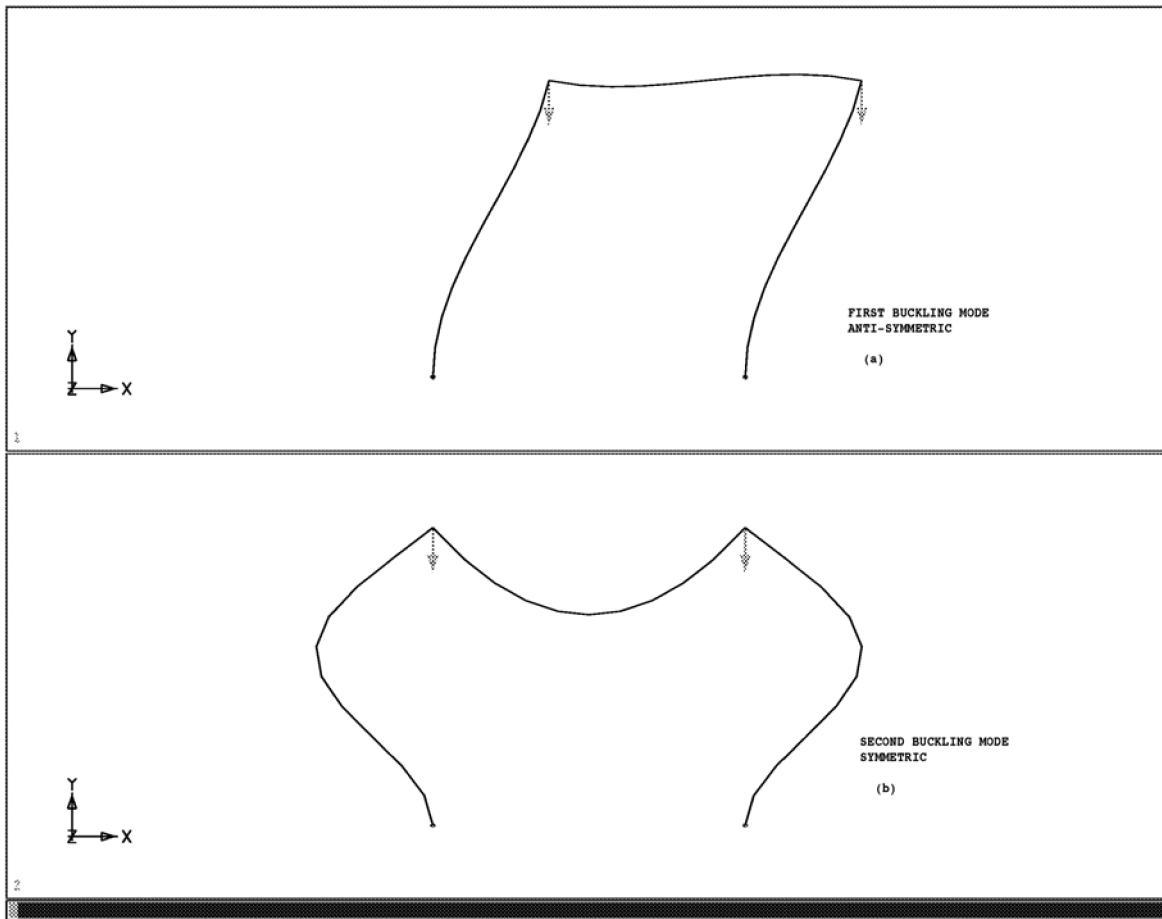


Figure 22-10. Frame Buckling

NX Nastran Results for Example 3

Listing 22-4 shows a portion of the input file for this problem.

```
$
$ INPUT FILE - bukframe.dat
$
SOL      105  $
TIME     5
CEND
TITLE = FRAME BUCKLING PROBLEM
SUBTITLE = PLANAR MODEL
DISP = ALL
SPC = 1
$
SUBCASE 1
LABEL = STATIC LOAD CASE
LOAD = 1
$
SUBCASE 2
LABEL = BUCKLING CASE
METHOD = 10
$
BEGIN BULK
$
```

```

$ BRING IN REST OF BULK DATA FILE
$
INCLUDE 'bucklat.dat'
$
$ LOADS BULK DATA ENTRIES
$
FORCE 1      11      0      5000.      -1.
FORCE 1      22      0      5000.      -1.
$
$ CONSTRAINTS BULK DATA ENTRIES
$
SPC1 1      123456 1      33
$
$ PROPERTY AND MATERIAL BULK DATA ENTRIES
$
PBAR 1      1      3.46 1.      1.
MAT1 1      1.+7      .3
$
$ FOR CALCULATING BUCKLING MODES
$
EIGRL 10      3
ENDDATA

```

Listing 22-4. Input File for Frame Buckling

The lowest buckling load calculated by NX Nastran is

$$P_{cr_1} = (\lambda_1)(P_a) = 1.4756 (-5000) = -7378 \text{ lb}$$

Theoretical Results for Example 3

From *Theory of Elastic Stability*, the buckling load P_{cr} can be calculated as

$$P_{cr} = k^2 E_2 I_2$$

Equation 22-11.

where the value of k can be obtained by solving the following transcendental equation

$$\frac{kl}{\tan kl} = \frac{-6 l I_1}{b I_2}$$

Equation 22-12.

Solving [Eq. 22-12](#), the lowest value of k is equal to

$$k_1 = 0.02716$$

Equation 22-13.

Note that all the roots from the above transcendental equation only yield the antisymmetrical modes. In this case, the lowest buckling load is

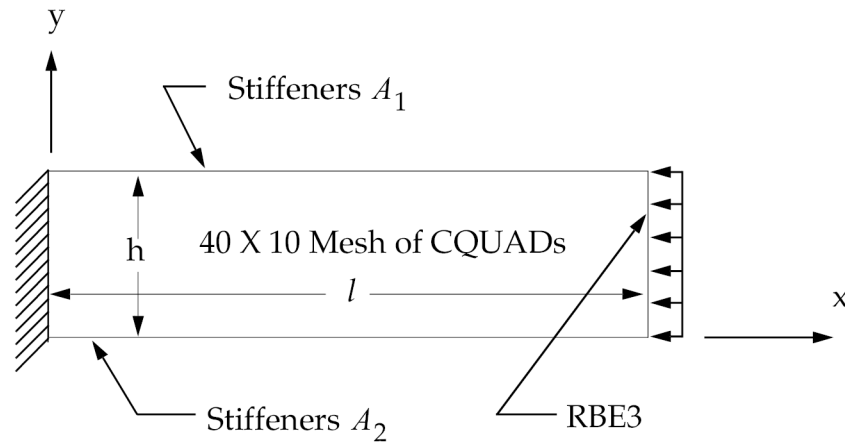
$$P_{cr_1} = (.02716)^2 (1 \cdot 10)^7 (1.0) = -7377 \text{ lb}$$

Table 22-4 compares the theoretical results to the NX Nastran results for the antisymmetric frame buckling.

Table 22-4. Frame Buckling Results Comparison		
NX Nastran (lb)	Theoretical (lb)	% Difference
7378	7377	0.01

Example 4 – Buckling of a Stiffened Panel with Transverse Shear Flexibility

This problem is selected to illustrate the effect of the transverse shear flexibility in the buckling failure of a stiffened panel. **Figure 22-11** consists of a panel reinforced with stiffeners at both the top and bottom. The panel is subjected to a distributed compressive load at the right edge. The failure mode of interest in this case is the in-plane (xy plane) buckling of the stiffened panel.



$$A_1 = A_2 = 0.6 \text{ in}^2 \quad ; h = 2.0 \text{ in}$$

$$A_t = 1.2 + 0.4 = 1.6 \text{ in}^2 \quad ; A_w = t(h) = 0.2(2.0) = 0.4 \text{ in}^2$$

$$E = 1.0 \cdot 10^7 \text{ psi} \quad ; I = 1.333 \text{ in}^4$$

$$l = 20 \text{ in} \quad ; G = 3.79 \cdot 10^6 \text{ psi}$$

$$t = 0.2 \text{ in} \quad ; n = A_t / A_w = 4$$

Figure 22-11. Buckling of a Stiffened Panel

NX Nastran Results for Example 4

The corresponding NX Nastran input file for this problem is shown in [Listing 22-5](#). The panel is modeled with CQUAD4s. Since only the in-plane buckling failure mode is of interest, only the membrane stiffness is requested for the CQUAD4s (MID1 only). The stiffeners are modeled with CRODs. Since this is a planar model, the out-of-plane motion is constrained (3, 4, 5, and 6 DOFs).

The panel is subjected to a distributed compressive load at the right end. As an alternative modeling technique, an RBE3 element is connected to the grid points at the right edge so that the load can be applied to a single grid point with the RBE3 spreading the loads to the other grid points.

The SINV method of eigenvalue extraction method is used for this problem. Therefore, the EIGB entry instead of the EIGRL entry is used. Field 3 of the EIGB entry designates the selected method. Fields 4 and 5 provide the range of the eigenvalue of interest. A reduced output showing the eigenvalue table and lowest buckling mode shape is included in [Figure 22-13](#) and [Figure 22-14](#), respectively.


```

$
$      FILENAME - stffqud4.dat
$
SOL      105
TIME     10
CEND
TITLE = PANEL WITH STIFFENERS
DISP = ALL
STRESS = ALL
SPC = 1
$
SUBCASE 1
LABEL = COMPRESSIVE LOAD
LOAD = 1
$
SUBCASE 2
LABEL = BUCKLING SUBCASE
METHOD = 10
$
BEGIN BULK
$
$EIGRL 10      1
EIGB   10      SINV   .7      .9
PARAM    POST      0
$
RBE3    1000      1000      123456 2.      12      82      123      +
+      164      205      287      328      369      410      2.      12345 +
+      246      1.0      12      41      451
$
FORCE   1      1000      -1.0      100000.
$
$ THIS SECTION CONTAINS THE LOADS, CONSTRAINTS, AND CONTROL BULK DATA ENTRIES
$
PSHELL  1      1      .2
PROD    2      1      .6
$
MAT1    1      1.+7      .32
$
$ BRING IN THE REST OF THE MODEL
$
include 'stffqud4.blk'
$
ENDDATA

```

Listing 22-5. Input File for the Buckling of Stiffened Panels

R E A L E I G E N V A L U E S						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.811782E-01	8.838429E-01	1.406680E-01	6.161843E+03	4.813498E+03
2	3	4.946603E+00	2.224096E+00	3.539759E-01	1.096963E+07	5.426240E+07
3	2	5.002625E+00	2.236655E+00	3.559746E-01	1.383715E+04	6.922209E+04

Figure 22-12. Eigenvalues for a Stiffened Panel

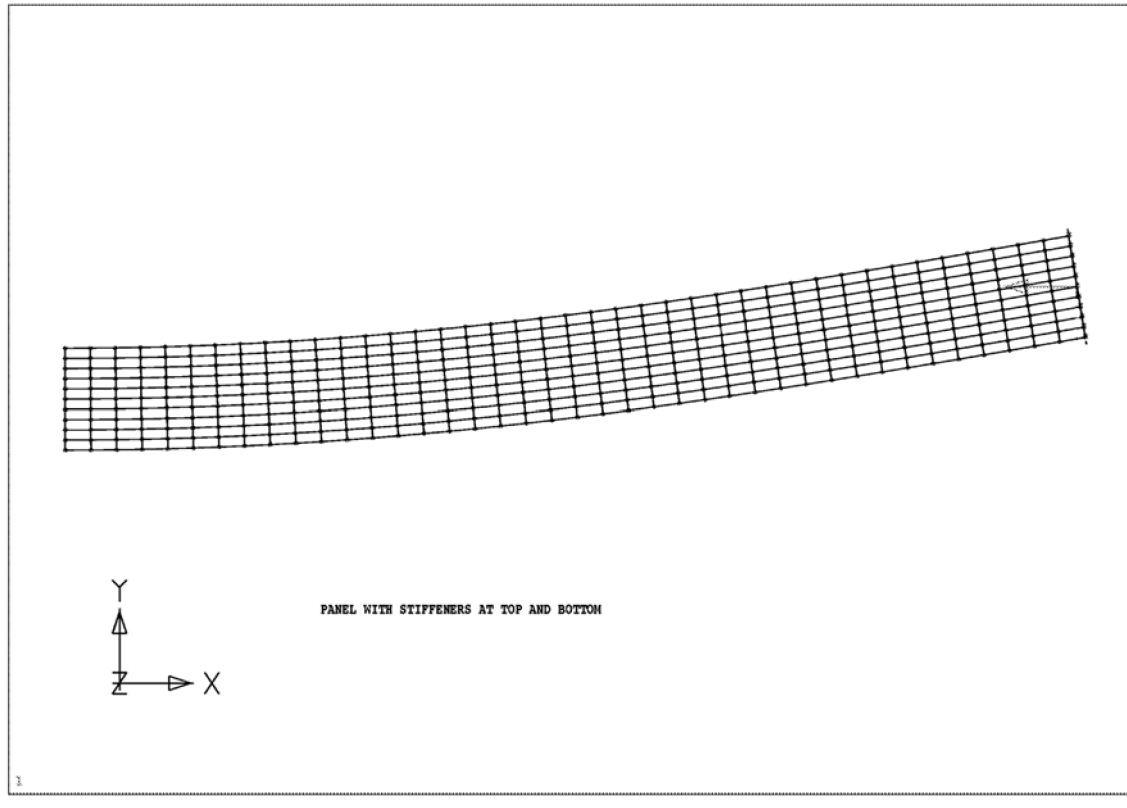


Figure 22-13. Stiffened Panel Buckling Mode

The lowest buckling load calculated by NX Nastran is

$$P_{cr_1} = (\lambda_1)(P_a) = 0.78118(-(100,000)) = -78,118 \text{ lb}$$

Theoretical Results for Example 4

If this problem is treated as an Euler beam, the critical buckling load P_e is equal to

$$P_e = \frac{\pi^2 EI}{4 l^2} = 82,245 \text{ lb}$$

Equation 22-14.

If the effect of the transverse shear flexibility is included, then it can be calculated as follows (Timoshenko and Gere, *Theory of Elastic Stability*, 1961):

$$P_{cr} = \frac{\sqrt{1 + \frac{4nPe}{A_t G}} - 1}{\frac{2n}{A_t G}} = 78,210 \text{ lb}$$

Equation 22-15.

If the transverse shear flexibility is ignored, then the buckling load deviates by 5.2%.

Table 22-5 contains a comparison between the theoretical results versus the NX Nastran results for the buckling of the stiffened panel. Note that the transverse shear flexibility of this structure is automatically included when you are performing a buckling analysis in NX Nastran. There is no additional input required on your part.

Table 22-5. Stiffened Panel Buckling Results Comparison		
NX Nastran (lb)	Theoretical (lb)	% Difference
78,118	78,210	0.1

A review of the stresses also indicates that the structure will yield prior to reaching the linear buckling load level. In other words, the critical failure mode may be due to yielding rather than to the linear buckling of the structure. The knowledge of this linear buckling load level can still be of design significance. If you are interested in detailed stresses for this problem, then a nonlinear analysis using Solution 106 may be more appropriate in this case.

Example 5 – Buckling of a Cylinder Under Uniform Axial Load

The next example is the buckling of a cylinder subjected to a distributed compressive load at one end and simply supported at the other end. The cylinder has a diameter of 20 inches and a length of 20 inches with a wall thickness of 0.03 inches as shown in **Figure 22-14**. This problem illustrates the phenomenon of the buckling of a thin curved shell structure.

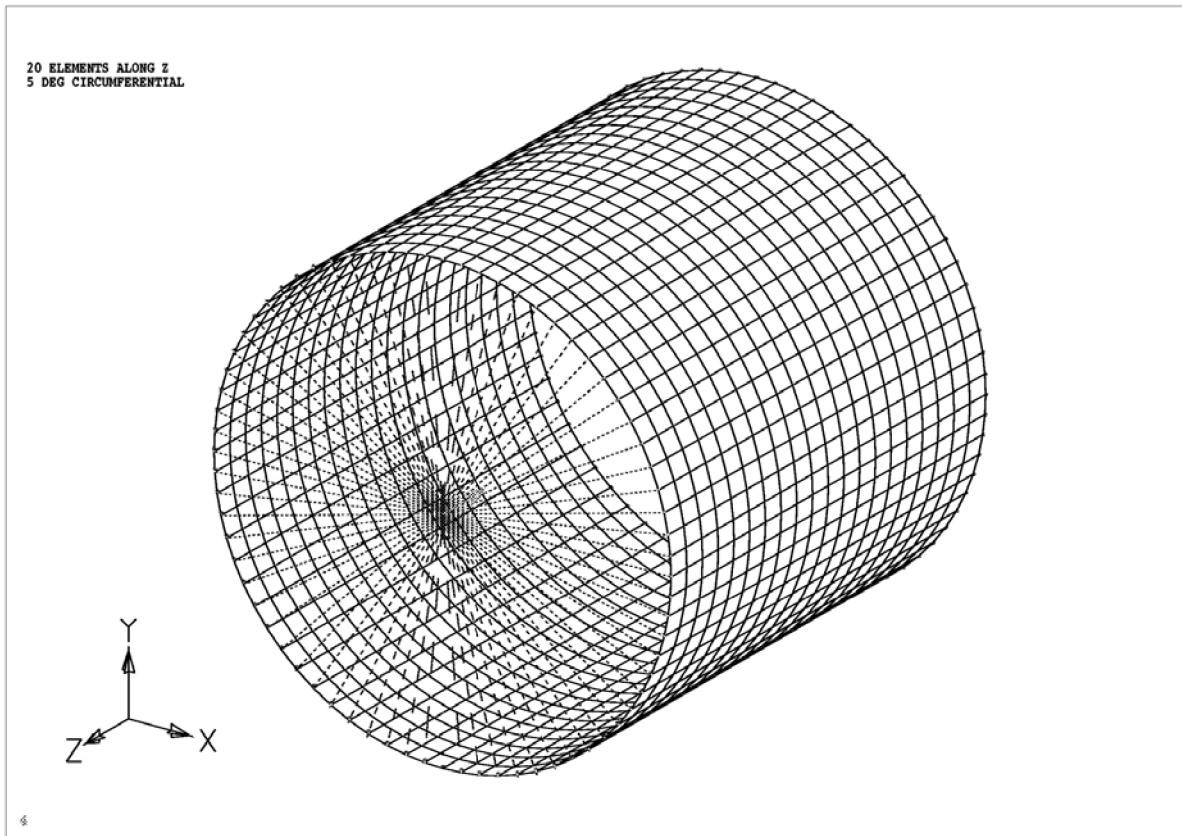


Figure 22-14. Buckling of a Cylinder Under a Compressive Load

NX Nastran Results for Example 5

The CQUAD4 elements are used for the NX Nastran model. A mesh of 20 elements along the length and 72 elements around the circumference is used for this problem. The corresponding input file containing the pertinent entries is shown in [Listing 22-6](#).

```

$  FILENAME - buckcy20r.dat
$
ID CYLIN BUCKLING
SOL      105
TIME     200
CEND
TITLE = BUCKLING OF CYLINDER - SIMPLY SUPPORTED
SUBTITLE = 20" x 20" - t=.03" - CP IN CYL COORD SYSTEM - K6ROT = 100.
DISP = ALL
SPC = 1
$
SUBCASE 1
LABEL = STATIC LOAD
LOAD = 1
SPCF = ALL
STRESS = ALL
$
SUBCASE 2
LABEL = EIGENVALUE CALCULATION
METHOD = 1
$
BEGIN BULK
$
PARAM      K6ROT      100.
PARAM      POST       -1
EIGRL      1                      2
$
PSHELL     1          1          .03      1
$
MAT1       1          1.+7          .3
$
include 'full_cp1.blk'
$
$ THIS SECTION CONTAINS ALL DEFINED COORDINATE SYSTEMS
$
CORD2C     1          0          0.0      0.0      0.0      0.0      0.0      1.      +
           1.          0.0      0.0
$
RBE3       5000          5000      123456  1.          123      381      382      +
+          383          384      385      386      387      388      389      390      +
+          391          392      393      394      395      396      397      398      +
+          399          781      782      783      784      785      786      787      +
+          788          789      790      791      792      793      794      795      +
+          796          797      798      1180     1181     1182     1183     1184      +
+          1185         1186     1187     1188     1189     1190     1191     1192      +
+          1193         1194     1195     1196     1197     1579     1580     1581      +
+          1582         1583     1584     1585     1586     1587     1588     1589      +
+          1590         1591     1592     1593     1594     1595
$
FORCE      1          5000      0          100000.          -1.
$
ENDDATA

```

Listing 22-6. Input File for a Cylindrical Buckling Problem

Once again, an RBE3 element is used to distribute the load from a single grid point to the circumference of the cylinder.

A static load of $-100,000$ lb is applied in Subcase 1 to generate the internal forces for the structure. This load is applied to the RBE3, which, in turn, distributes the loads around the circumference of the cylinder. The static deflection is shown in [Figure 22-15](#).

The Lanczos eigenvalue extraction method is used in this case to request the lowest mode. The lowest calculated eigenvalue is equal to 0.35986 . The buckling load is, therefore, equal to

$$P_{cr} = (.34262)(-100,000) = -34,262 \text{ lb}$$

The corresponding buckling mode shape is shown in [Figure 22-16](#). As you can see from the plots, there are two grid points per half sine wave (four grid points per sine wave), which is below the recommended value of a minimum of five grid points per half sine wave.

This problem is then remeshed with 40 elements along the z-direction keeping the same mesh density in the circumferential direction. The new eigenvalue calculated in this case is equal to 0.34914 . The revised buckling load is, therefore, equal to

$$P_{cr} = (.34234)(-100,000) = -34,234 \text{ lb}$$

The corresponding mode shape is shown in [Figure 22-17](#). In this case, there are three grid points per half sine wave (six grid points per sine wave), which is still below the minimum requirement. To obtain better accuracy, you can certainly further refine this model until you meet the minimum number of grid points requirement. However, this mesh is sufficient for demonstrating the basic linear buckling features of NX Nastran.

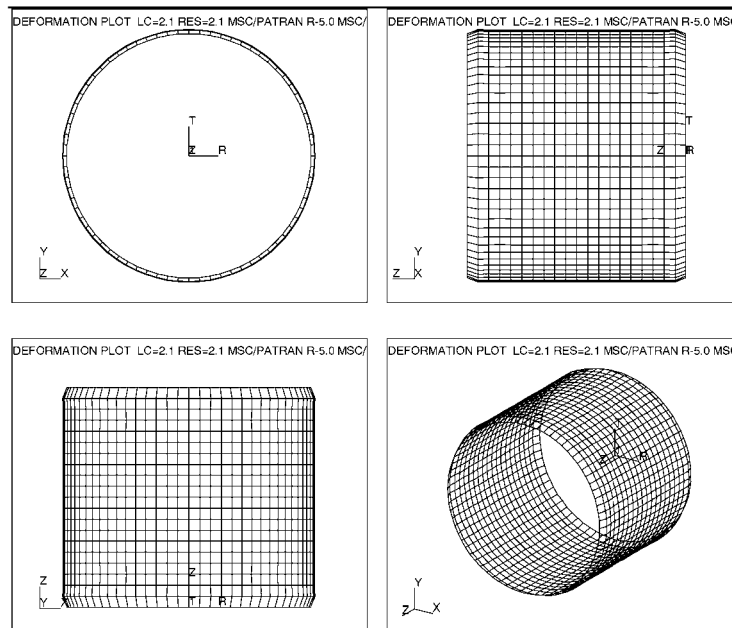


Figure 22-15. Static Deflection of a Cylinder Due to a Compressive Load

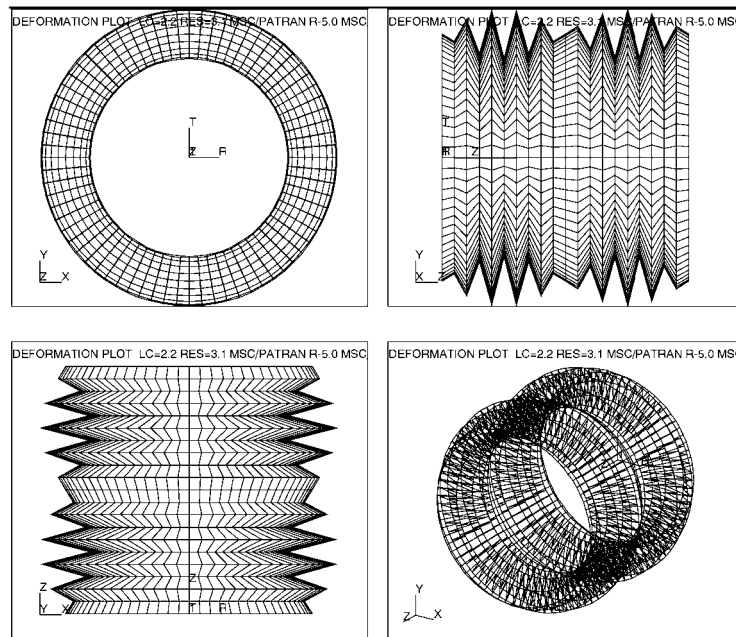


Figure 22-16. Buckling Shapes of a Cylinder with 20 Elements Along the z-Direction

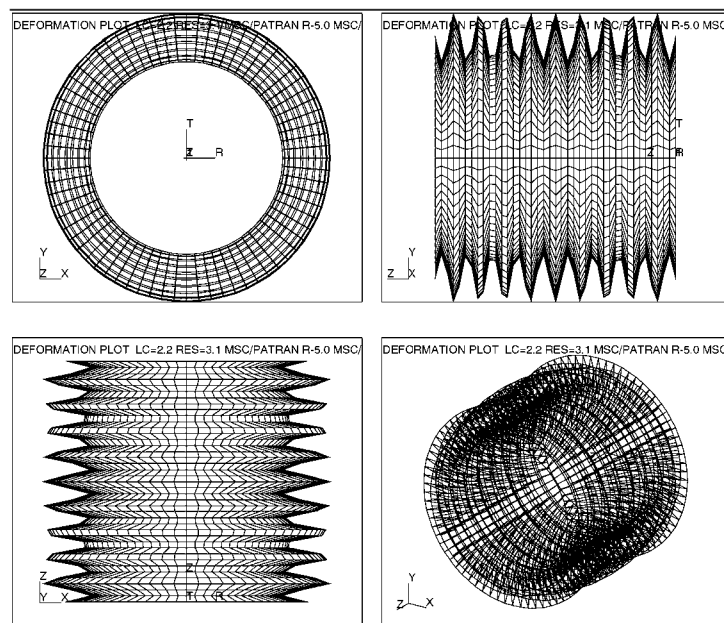


Figure 22-17. Buckling Shape of a Cylinder with 40 Elements Along the z-Direction

Theoretical Results for Example 5

The first buckling load (Timoshenko and Gere, *Theory of Elastic Stability*, 1961) can be calculated as follows:

$$N_x = \frac{E(h)^2}{r\sqrt{3(1-\nu^2)}} = \frac{1 \times 10^7 (.03)^2}{10\sqrt{3(1-.3^2)}} = 544.7 \text{ lb/in}$$

$$P_{cr} = N_x(2)(\pi)r = 544.7(2)(\pi)(10) = 34,225 \text{ lb}$$

Table 22-6 contains a comparison between the results obtained with NX Nastran versus the theoretical results.

Table 22-6. Thin Cylinder Buckling Results Comparison			
Mesh Density	NX Nastran (N)	Theoretical (N)	% Difference
72 x 20	34,262	34,225	.11
72 x 40	34,234	34,225	.03

Example 6 – Multiple Buckling Analyses in a Single Run

So far, a single static analysis was considered followed by a single buckling analysis. Example 6 shows you how to run multiple static and buckling analyses in a single run. In fact, what is done in this example is to combine the Euler beam buckling (Example 1) and lateral buckling (Example 2) problem into a single run. The corresponding input file is shown in **Listing 22-7**.

The model used is the same geometric beam model used in Examples 1 and 2. The first subcase (Subcase 2) is a static subcase consisting of a cantilever beam with a vertical tip load applied to the free end (see **Figure 22-6**). The second subcase (Subcase 5) is a static subcase consisting of a simply supported beam with an axial load applied to the roller end (see **Figure 22-2**). The third subcase (Subcase 11) is for a lateral buckling analysis. The Case Control command (STATSUB = 2 in Subcase 11) tells NX Nastran that you want to generate the differential stiffness matrix from the first static subcase (Subcase 2). The fourth subcase (Subcase 21) is for a Euler beam buckling analysis. The Case Control command (STATSUB = 5 in Subcase 21) tells NX Nastran that you want to generate the differential stiffness matrix from the second static subcase (Subcase 5). The Bulk Data entries are similar to Examples 1 and 2. Note that different boundary conditions are allowed for different subcases.

```

$   FILENAME - MULTBUCK.DAT
$
SOL 105
TIME 10
CEND
$
TITLE = LATERAL BUCKLING OF CANTILEVER BEAM
SUBTITLE = METRIC UNITS
DISP = ALL
$
SUBCASE 2
LABEL = CANTILEVER BEAM
LOAD = 10
SPC = 10
$
SUBCASE 5
LABEL = SIMPLY SUPPORTED BEAM

```



```
LOAD = 20
SPC = 20
$
SUBCASE 11
LABEL = LATERAL BUCKLING OF CANTILEVER BEAM
METHOD = 10
SPC = 10
STATSUB = 2
$
SUBCASE 21
LABEL = EULER BUCKLING OF SIMPLY SUPPORTED BEAM
METHOD = 10
SPC = 20
STATSUB = 5
$
```

Listing 22-7. Input File for Multiple Buckling Analyses (Continued)

```

BEGIN BULK
$
EIGRL 10 2
$
FORCE 10 11 -1000. 0. 1. 0.
FORCE 20 11 -1000. 1. 0. 0.
$
CBEAM 1 1 1 2 100
CBEAM 2 1 2 3 100
CBEAM 3 1 3 4 100
CBEAM 4 1 4 5 100
CBEAM 5 1 5 6 100
CBEAM 6 1 6 7 100
CBEAM 7 1 7 8 100
CBEAM 8 1 8 9 100
CBEAM 9 1 9 10 100
CBEAM 10 1 10 11 100
$
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
GRID 3 2. 0. 0.
GRID 4 3. 0. 0.
GRID 5 4. 0. 0.
GRID 6 5. 0. 0.
GRID 7 6. 0. 0.
GRID 8 7. 0. 0.
GRID 9 8. 0. 0.
GRID 10 9. 0. 0.
GRID 11 10. 0. 0.
GRID 100 0. 0. 10.
$
MAT1 1 7.1+10 .33 2700.
$
$ RECTANGULAR SECTION OF DIMENSION .1m x .02m
$
PBEAM 1 1 .002 6.667-8 1.667-6 2.332-7
$
SPC1 10 123456 1
$
SPC1 20 123 1
SPC1 20 23 11
SPC 20 1 4
$
ENDDATA

```

Listing 22-7. Input File for Multiple Buckling Analyses

Chapter 23: Follower Stiffness

- *Introduction*
- *Benefits*
- *Input*
- *Output*
- *Linear Dynamic Analysis with Follower (SOLs 108, 109, 111, and 112)*

23.1 Introduction

A follower stiffness is an adjustment to the standard linear stiffness matrix to account for a load's dependency on the orientation. A common example to demonstrate is a bar in compression as a result of an axial force, and the force direction remains tangential to the deformed axis:

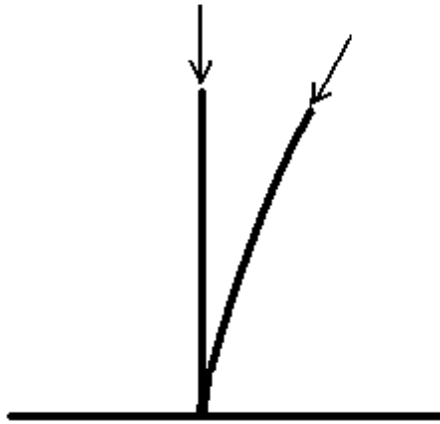


Figure 23-1. Follower Force Example

Assuming that differential stiffness is also added in this example, the new stiffness matrix would become:

$$K_{\text{total}} = K_{\text{linear}} + K_{\text{differential}} + K_{\text{follower}}$$

Follower stiffness due to pressure loads (PLOAD, PLOAD2, PLOAD4) and point loads (FORCE1, FORCE2, MOMENT1, MOMENT2) were first introduced for modal analysis (SOLs 103 and 107) and buckling analysis (SOL 105) and are applicable for both quadratic elements (QUAD8, TRIA6, HEXA, PENTA, TETRA) and linear elements (QUAD4, TRIA3, HEXA, PENTA, TETRA). Follower stiffness was then extended to dynamic responses for direct response analyses (SOLs 107, 108 and 109) and modal response analyses (SOLs 110, 111 and 112). Centrifugal load (RFORCE) was implemented for buckling, dynamic analyses, and cyclic symmetry analysis for modal and buckling (SOLs 115 and 116). Follower stiffness has been further extended to include nonlinear analysis (SOL 106) for nonlinear static, nonlinear modal and nonlinear buckling analyses.

23.2 Benefits

The follower stiffness is generated if the structure is loaded by follower forces. Effects of follower stiffness are prominent in dynamic and buckling analyses. For example, for a cylinder under external pressure critical buckling load may be over-estimated and the natural frequencies in vibration may be under-estimated in the absence of follower stiffness. To the contrary, this observations are reversed in case of centrifugal loads.

For moderately geometric nonlinear analysis, exclusion of follower stiffness affects the rate of convergence, but the converged solution is correct. For severely geometric nonlinear analysis, it may not be possible to obtain a converged solution without including follower stiffness. As the geometric

nonlinearity intensifies, so is the effect of follower stiffness. Therefore, inclusion of follower stiffness greatly enhances the convergence if the deformation involves severe geometric nonlinearity.

23.3 Input

The follower stiffness is primarily controlled by PARAM, FOLLOWK. By default, the follower stiffness is included wherever applicable in all the solution sequences mentioned above. The effect of follower stiffness may be ignored by specifying PARAM, FOLLOWK, NO. This parameter may be changed from Subcase to Subcase.

You may also perform multiple dynamic analyses in a single run to study the effects of preload. For this, you will need to specify multiple static Subcases (with static loads) in a dynamic analysis solution sequence. For example, to study the effect of preloads on normal modes, you will need to:

1. Specify a static load subcase for each of the loading conditions, and
2. Specify normal modes subcases by specifying the METHOD command and selecting the pre-loading condition by the STATSUB command.

In case of buckling analysis, two different STATSUB conditions may be selected: one for sustaining preload by STATSUB(PRELOAD) and another for modulated preload by STATSUB(BUCKLE) for the critical load in search.

In nonlinear analysis (SOL 106), the follower stiffness is included if PARAM, LGDISP, 1 is specified. Although it is not recommended under normal circumstances, follower stiffness may be ignored in SOL 106 by PARAM, FOLLOWK, NO. When PARAM LGDISP is greater than 0, the differential stiffness of the superelements are also computed and added to the nonlinear stiffness. This will result in somewhat different solution than Version 70.5 if superelements are present. The follower stiffness in nonlinear static solution improves convergence rate, but it does not alter the solution except for a small difference due to round-off error.

23.4 Output

When a static subcase is specified for linear transient response analysis (SOLs 109 and 112), the data recovery is controlled by PARAM, ADSTAT. By default (YES) the static solution will be superimposed on the dynamic response solution (displacement, stress and SPCForce). The relative solution can be obtained in reference to the static solution point by PARAM, ADSTAT, NO. No provision is made for frequency response analysis, because the static responses contribute only to the zero frequency response.

23.5 Linear Dynamic Analysis with Follower Stiffness (SOLs 108, 109, 111, and 112)

If the external load changes as function of displacements, it is termed follower force, which creates follower stiffness. The derivatives of the external load constitute a follower stiffness, i.e.,

$$[K_{ij}]^f = -\frac{\partial}{\partial u_j}\{P\}_i = -\frac{\partial}{\partial X_j}\{P\}_i$$

This follower stiffness is included as a part of differential stiffness in MSC.Nastran. In direct transient response analysis (SOL 109), the following equation is solved

$$M\ddot{u} + B^* \dot{u} + [K + K^d]u = P(t)$$

Equation 23-1.

with

$$B^* = B + \frac{g}{W_3} K^1 + \frac{1}{W_4} \sum_e g_e K_e$$

In direct frequency response analysis (SOL 108) the following equation is solved:

$$M\ddot{u} + B\dot{u} + [(1 + ig)K + K^d]u = P(t)$$

or in the frequency domain, the complex equation

$$[-\omega^2 M + i(\omega B + gK) + (K + K^d)]u(\omega) = P(\omega)$$

Equation 23-2.

is solved by Gauss elimination in complex domain.

The modal transient response analysis (in SOL 112) reduces the system by normal modes before integration, i.e.,

$$\phi^T M \phi \ddot{q} + \phi^T B \phi \dot{q} + \phi^T [K + K^d] \phi q = \phi^T P(t)$$

Equation 23-3.

If the damping is proportional, the above equation can be decoupled by

$$\ddot{q}_n + 2\zeta_n \omega_n \dot{q}_n + \omega_n^2 q_n = \frac{1}{M_n} \phi_n^T P(t)$$

The modal frequency response analysis (in SOL 111) reduces the system by normal modes before Gauss elimination in the complex domain, i.e.,

$$[-\omega^2 \phi^T M \phi + i(\omega \phi^T B \phi + g \phi^T K \phi) + \phi^T (K + K^d) \phi] q(\omega) = \phi^T P(\omega)$$

Equation 23-4.

In presence of follower forces, the differential stiffness in the above equations includes follower stiffness. At this point, the solution is the one perturbed from the static solution under preload.

The follower stiffness also affects the data recovery in dynamic response analysis, controlled by PARAM, ADSTAT. For linear dynamic response, the static solution can be superimposed after the dynamic solution procedure. The preload effect is reflected only in the stiffness and the actual static load is omitted in the dynamic response analysis. The total displacement can be obtained by superposing the static solution to the transient response analysis:

$$\{u\}_{total} = \{u\}_{dynamic} + \{u\}_{static}$$

Equation 23-5.

and

$$\{u\}_{total} = [\Phi]\{q\} + \{u\}_{static}$$

Equation 23-6.

The stress output can be obtained by

$$\sigma = [S]\{u_{static} + u_{dynamic}\}$$

Equation 23-7.

In case of Matrix method (PARAM, DDRMM, 0) for modal approach,

$$\sigma = [S]\left\{u_{static} + \sum_n^H \Phi q_n\right\}$$

Equation 23-8.

is processed by the module DDRMM using augmented matrices:

$$[\Phi^*] = [u_{static} | \Phi]$$

and

$$[q^*] = \begin{bmatrix} 1 & 1 & 1 \dots \\ q_1 & q_2 & q_3 \dots \end{bmatrix}$$

which results in

$$\sigma = [S][\Phi^*][q^*]$$

with a static deformed shape as the first mode. The mode acceleration method also requires differential stiffness in the recovery process, i.e.,

$$[\phi^T(K + K^d)\phi]q = \phi^T P(t) - \phi^T M \phi \ddot{q} - \phi^T B \phi \dot{q}$$

Then the mode acceleration is expressed as

$$[(K + K^d + K_{gg}^2)]u = P(t) - K_{pp}^2 \phi q - [M + M_{gg}^2 + M_{pp}^2] \phi \ddot{q} - [B + B_{gg}^2 + B_{pp}^2] \phi \dot{q}$$

Equation 23-9.

where K2GG is included in the modal analysis but not K2PP.

Chapter 24: Laminates

- *Overview of Laminated Composite Materials*
- *Understanding Classical Lamination Theory*
- *Understanding Laminate Failure Indices*
- *Calculating Interlaminar Shear Stress and Strains*
- *Examples: Defining Laminate Material Properties*

24.1 Overview of Laminated Composite Materials

The term composite refers to an engineering material that is made up of more than one material. Classical lamination theory takes the term composite another step to mean a material that is composed of stacks of ply layers or lamina, with each lamina having its own properties. A laminate is a stack of these individual lamina arranged with the individual lamina having different orientations of the principal material directions. The laminae are bonded together with a thin layer of bonding material that is considered to be of zero thickness.

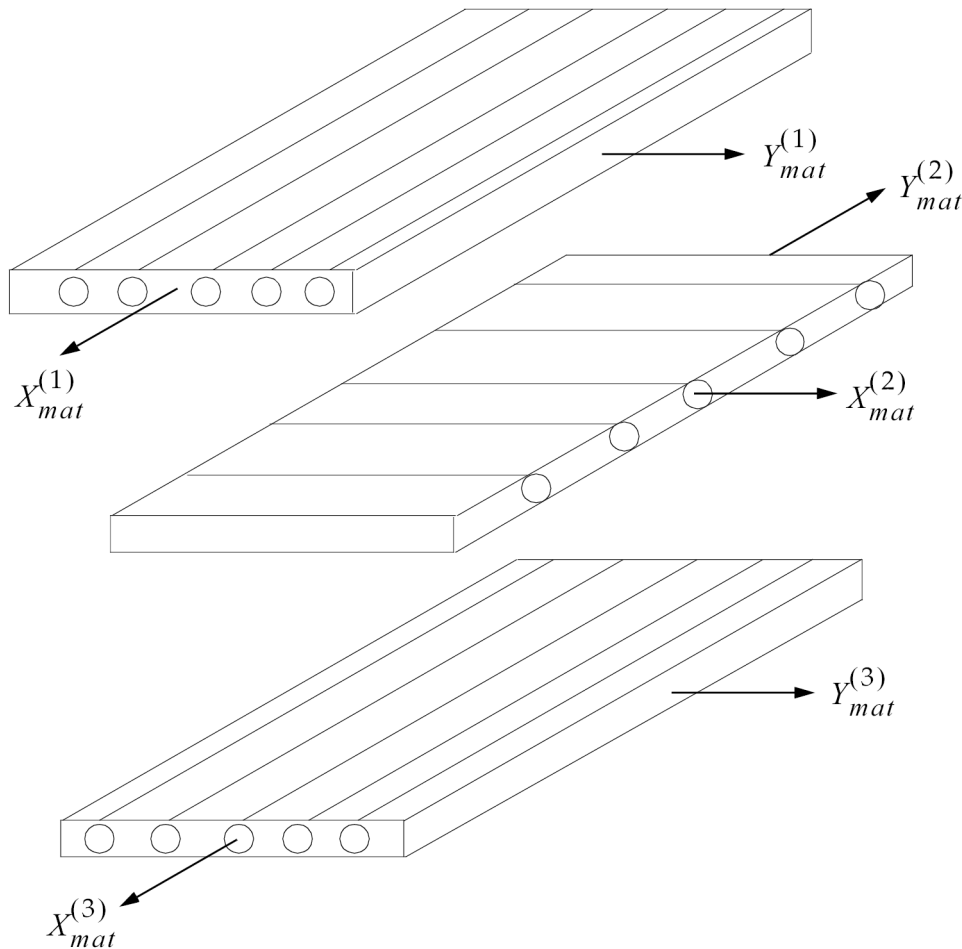


Figure 24-1. Laminae Arranged to Form a Laminate

Each ply or lamina may be considered as a group of unidirectional fibers. The ability to orient the fibers in particular directions lets you tailor the mechanical properties of a composite material to match the loading environment. The principal material directions for the lamina are parallel and perpendicular to the fiber direction. The principal material directions are referred to as:

- “Longitudinal” or “1-direction”, which corresponds to the fiber direction.
- “Transverse” or “2-direction”, which corresponds to the direction perpendicular to the fibers.

Figure 24-2 shows an exploded view of three cross-ply laminated plates. The n -laminae ($n = 1, 2, 3, 4$) of each of the three configurations are normal to the z -axis of the indicated coordinate system and the

1- and 2-axes appended to the individual lamina denote principal material directions. The directions of the principal material axes of each lamina alternate as implied by the use of the word "cross-ply" to describe the configuration. The xy-plane of the coordinate axes is defined in the geometric middle plane of the laminae.

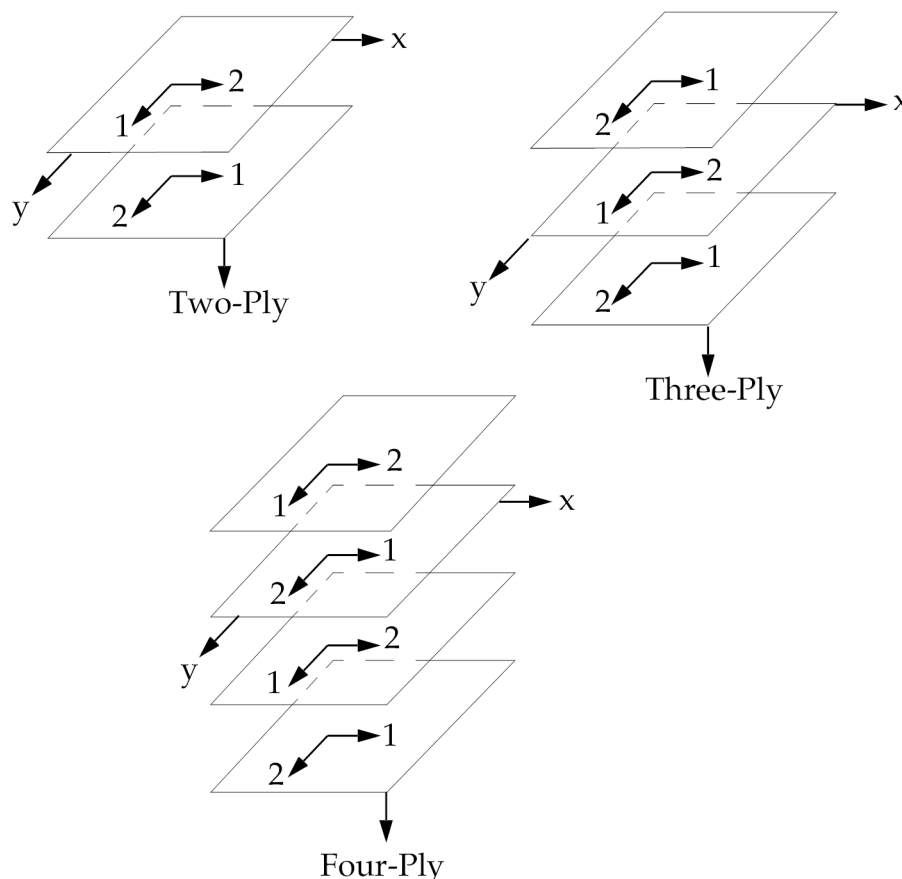


Figure 24-2. Exploded View of Three Cross-Ply Laminated Plates

Plies are typically made of fibers bonded together by a matrix. If the ply is a tape, all the fibers are oriented in the same direction. Cloth plies have fibers woven in two directions. Many different materials can be used as fibers or matrices. Examples of fibers include graphite, glass, boron, silicon carbide, and tungsten. Examples of matrices include epoxy and aluminum.

Modeling Laminates in NX Nastran

Three methods are available for you to model composite laminates. The PSHELL and PCOMP methods are used to model laminates with plate and shell elements like the CQUAD4, CQUAD8, CTRIA3, and CTRIA6 elements. The PSHELL and PCOMP methods are based upon classical lamination theory.

- You can use the PSHELL method to directly input membrane, bending, membrane-bending coupling, and transverse shear constitutive relationships.

- You can use the PCOMP method to define the composite laminate on a ply-by-ply basis. When you use the PCOMP method, the software computes the equivalent PSHELL and MAT2 entries for you.

The third method is the PCOMPS method. The PCOMPS method is used to model laminates with CHEXA and CPENTA solid elements. Unlike the other two methods, the PCOMPS method is not based on classical lamination theory. It is useful for modeling very thick laminates where interlaminar normal stresses may be important.

Each method allows you to model an entire stack of laminae with a single element because the material properties of the stack are completely reflected in the matrices of elastic moduli for the element. With the PCOMPS method, you can also optionally model the entire stack of laminae with multiple solid elements. In all three methods, the software automatically calculates these matrices from user-supplied definitions of the thickness, the material properties, and the relative orientation of these properties for the individual lamina. Once the software calculates the matrices of elastic moduli, the analysis proceeds.

Because the material properties of the laminate are completely contained in the matrices of elastic moduli, you can use standard data recovery methods to have NX Nastran calculate:

- Stresses, strains, and appropriate failure indices in individual laminae.
- Interlaminar stresses and strains and a bonding failure index.

If the loading on a structure is sufficient to exceed the elastic limit of the material or if the material behaves nonlinear elastically, then nonlinear methods are required. SOL 601 supports material nonlinear analysis of laminated structures defined using the PCOMP method.

The PSHELL, PCOMP, and PCOMPS methods are described further in the following sections.

Note: For three-dimensional composite analyses, you must develop the anisotropic material matrix yourself. Typically, you would use the MAT9 bulk entry to define this material matrix.

See also

- “**MAT9**” in the *NX Nastran Quick Reference Guide*.
- “**Understanding MAT9**” in the *NX Nastran User’s Guide*.
- “Composite shell elements” in the *NX Nastran Advanced Nonlinear Theory and Modeling Guide* for SOL 601 information.

Using the PSHELL Method

With the PSHELL method, you use the PSHELL entry with the appropriate material definitions and geometric parameters to directly define a laminate. For example, you can use PSHELL to define properties such as membrane thickness, material properties, and bending and transverse shear parameters.

On the PSHELL entry, there are four different material ID (MID) fields as shown in [Table 24-1](#).

Table 24-1. Material Properties on the PSHELL Entry

ID	PSHELL Field	Purpose
MID1	3	Membrane material property.
MID2	5	Bending material property.
MID3	7	Transverse shear stiffness material property.
MID4	4 (cont.)	Material property used to define coupling between the membrane and the bending deformation.

You can specify MAT1, MAT2, or MAT8 as the material type for any of the material (MIDi) fields. For isotropic materials, use the same MAT1 identification for MID1 and MID2, and leave the MID3 and MID4 fields blank.

- Use MID1 if you want to include membranes only.
- Use MID2 (and optionally, MID3) if you want to include bending only.
- Use MID3 to have the software use thick plate theory, which includes the transverse shear flexibility, to develop the element stiffness matrix. In general, for thin or curved surfaces, you should not use MID3.
- Use MID4 to input a coupling relationship between the in-plane forces and bending moments. This coupling only occurs in plates that are non-symmetric about the neutral plane or where the neutral plane is offset from the grid points. Typical applications include reinforced skins and aluminum bonded to fiberglass. You should leave the MID4 field blank if the element cross section is symmetric.

While the PSHELL method is sufficient for defining simple composites, determining the appropriate material definitions can be difficult for more complex composites. Additionally, when you use the PSHELL method, you can only recover smeared element data (running loads and moments, and equivalent strains and curvatures).

See also

- “PSHELL” in the *NX Nastran Quick Reference Guide*.

Using the PCOMP Method

The PCOMP entry provides a convenient way of entering appropriate material properties for a plate consisting of layers of unidirectional fibers. The input on the PCOMP entry consists of the layer-by-layer material definition. You can use PCOMP to define the thickness, orientation, and the material identification number of each of the individual lamina.

When you solve your model, NX Nastran calculates the membrane, bending, transverse shear, and coupled membrane-bending material properties of the laminate as a whole from the data you specified on the PCOMP entry. The software outputs these calculated properties in the form of one equivalent PSHELL and four MAT2 entries, as shown in [Figure 24-3](#). If you specify ECHO = PUNCH in the case control section, the software prints the generated PSHELL and MAT2 entries in the PUNCH file after the sorted bulk data echo.

Because the MIDi fields use more than eight digits, the software uses the large field format for the generated PSHELL and MAT2 entries. You can then modify your input file with these new entries and proceed with your analysis.

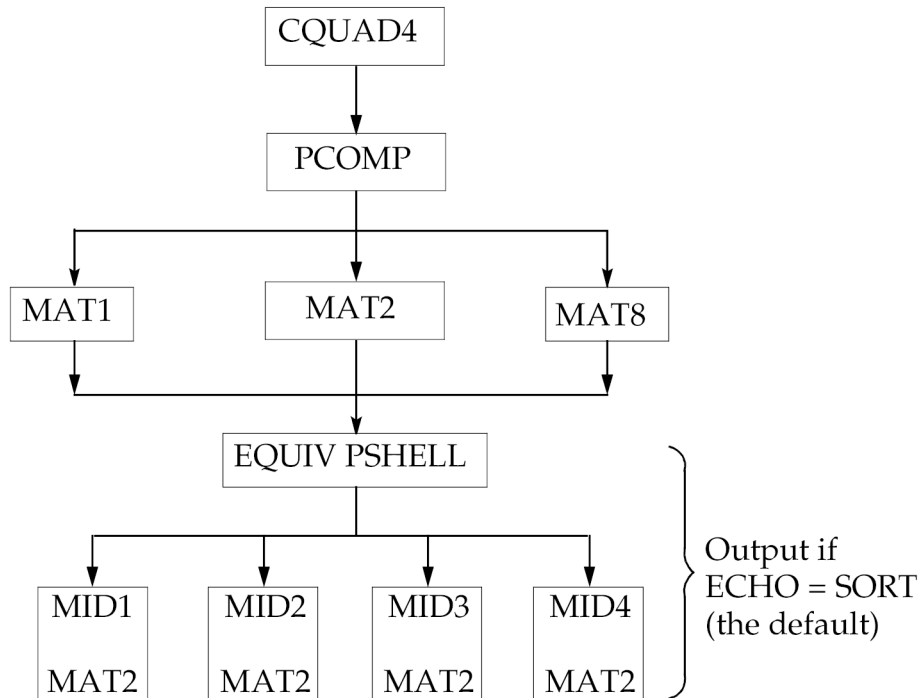


Figure 24-3. Equivalent PSHELL and MAT2 Entries Are Generated

The ID numbers of the generated MAT2 entries are important.

- The MID1 material has an ID in the range of 100000000 to 199999999.
- The MID2 material has an ID in the range of 200000000 to 299999999, etc.

These MIDi ranges alert NX Nastran that the material is part of a composite analysis. Therefore, if you are using the equivalent properties in a future analysis instead of using the PCOMP entry, and you are entering a thermal coefficient of expansion, don't change the ID numbers.

See also

- **“PCOMP”** in the *NX Nastran Quick Reference Guide*.

Using the PCOMPS Method

The PCOMPS method is used to model laminates with CHEXA and CPENTA solid elements. Unlike the other methods, the PCOMPS method is not based on classical lamination theory and is useful for modeling very thick laminates where interlaminar normal stresses may be important.

Like the PCOMP method, the PCOMPS method allows you to define the laminate on a layer-by-layer basis. Using the PCOMPS bulk entry, you can:

- Define the orientation, stacking, and thickness of each ply

- Assign MAT1, MAT9, or MAT11 material property entries to each ply
- Define reference temperature and damping coefficient for the laminate
- Specify ply and interlaminar failure theories and define allowable interlaminar shear and normal stresses

Because the solid elements used to model a laminated structure have potentially numerous material coordinate systems, but otherwise identical element properties, consider using MATCID bulk entries to map the solid elements to a single PCOMPS property entry. By minimizing the number of PCOMPS entries, the size of the input file is reduced and the model data is handled more efficiently.

See also

- “PCOMPS” in the *NX Nastran Quick Reference Guide*.
- “MATCID” in the *NX Nastran Quick Reference Guide*.

Selecting Output Options

Stress and strain output for the individual lamina is supported in solutions 101, 103, 105, 106, 109, 112, 114, 129, 144, 200, and 601. In all other solutions except for heat transfer SOLs 153, 159 and SOL 701, stress and strain can only be recovered for the equivalent laminate. That is, output on the equivalent PSHELL created by the software.

The output you may request for PCOMP or PCOMPG entries in solutions 101, 103, 105, 106, 109, 112, 114, 129, 144, and 200 includes:

- Stresses and strains for the equivalent laminate (that is, output on the equivalent PSHELL created by the software)
- Stress resultants, which are requested with the FORCE case control
- Stresses and strains in the individual lamina including approximate interlaminar shear stress in the bonding material output
- A failure index table

See the remarks on the PCOMP and PCOMPG bulk entry listings in the *Quick Reference Guide* for information on these output requests.

See the remarks related to SOL 601 on the PCOMP bulk entry listing in the *Quick Reference Guide* for information on requesting laminate results for solution 601. Also see “Composite shell elements” in the *NX Nastran Advanced Nonlinear Theory and Modeling Guide* for solution 601 information.

See also

- “STRESS” in the *NX Nastran Quick Reference Guide*.
- “NOCOMPS” in the *NX Nastran Quick Reference Guide*.

24.2 Understanding Classical Lamination Theory for Shell Composites (PCOMP,PCOMPG)

Classical lamination theory makes the following assumptions regarding the behavior of the laminae:

- The laminate consists of perfectly bonded laminae.
- The bonds are infinitesimally thin and nonshear-deformable; i.e., displacements are continuous across laminae boundaries so that no lamina can slip relative to another.
- There is linear variation of strain through the laminate thickness.

Deformation in the X-Y plan of the plate at any point C at a distance z in the normal direction to plate middle surface is

$$U = U_0 + z\theta_y$$

Equation 24-1.

$$V = V_0 + z\theta_x$$

Equation 24-2.

where U , V , and W are the displacements along the X, Y, and Z directions in the element coordinate system, and θ_x , θ_y are the rotations.

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

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial U_0}{\partial x} \\ \frac{\partial V_0}{\partial y} \\ \frac{\partial U_0}{\partial y} + \frac{\partial V_0}{\partial x} \end{Bmatrix} + z \begin{Bmatrix} \frac{\partial \theta_y}{\partial x} \\ -\frac{\partial \theta_x}{\partial y} \\ \frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x} \end{Bmatrix} = \begin{Bmatrix} 0 \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - z \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}$$

Equation 24-3.

where the 0 's and χ 's are the middle surface strains and curvatures, respectively.

The stress resultants for an N-layer laminate are obtained by integration of the stresses in each lamina through the laminate thickness as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz = \sum_{k=1}^N \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k dz$$

Equation 24-4.

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = - \int_{-\frac{t}{2}}^{\frac{t}{2}} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} z dz = - \sum_{k=1}^N \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k z dz$$

Equation 24-5.

The stress resultant to strain relationship is:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \sum_{k=1}^N [G]_k \left\{ \int_{z_{k-1}}^{z_k} \begin{Bmatrix} 0 \\ \varepsilon_x \\ 0 \\ \varepsilon_y \\ 0 \\ \gamma_{xy} \end{Bmatrix}_k dz - \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}_k z dz \right\}$$

Equation 24-6.

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \sum_{k=1}^N [G]_k \left\{ - \int_{z_{k-1}}^{z_k} \begin{Bmatrix} 0 \\ \varepsilon_x \\ 0 \\ \varepsilon_y \\ 0 \\ \gamma_{xy} \end{Bmatrix}_k z dz + \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}_k z^2 dz \right\}$$

Equation 24-7.

where $[G]_k$ is the material matrix transformed from the laminate coordinate system into the lamina coordinate system.

These relations can be written in the following form used to describe composite elements:

$$\begin{Bmatrix} F \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{Bmatrix} \varepsilon^0 \\ \chi \end{Bmatrix}$$

Equation 24-8.

where:

$[A]$	$=$	$\sum_{k=1}^N [G]_k (z_k - z_{k-1})$
$[B]$	$=$	$\frac{1}{2} \sum_{k=1}^N [G]_k (z_k^2 - z_{k-1}^2)$
$[D]$	$=$	$\frac{1}{3} \sum_{k=1}^N [G]_k (z_k^3 - z_{k-1}^3)$

are named in composite element literature as the membrane, membrane-coupling, and bending matrices, respectively.

In the shell element formulation in NX Nastran (CQUAD4, CQUAD8, CTRIA3, and CTRIA6), these relationships take the following form:

$$\begin{Bmatrix} N \\ M \\ Q \end{Bmatrix} = \begin{bmatrix} TG_1 & T^2G_4 & 0 \\ T^2G_4 & \frac{T^3}{12}G_2 & 0 \\ 0 & 0 & T_sG_3 \end{bmatrix} \begin{Bmatrix} \varepsilon^0 \\ \chi \\ \gamma \end{Bmatrix}$$

where:

Equation 24-9.

$[A]$	=	TG_1 (3x3 matrix)
$[B]$	=	$-T^2G_4$ (3x3 matrix)
$[D]$	=	$\frac{T^3}{12}G_2$ (2x2 matrix)
$\{Q\}$	=	$\begin{Bmatrix} Q_x \\ Q_y \end{Bmatrix}$ = transverse shear resultants
$\{Y\}$	=	$\begin{Bmatrix} \gamma_x \\ \gamma_y \end{Bmatrix}$ = transverse shear strains
T		nominal plate thickness
T_s	=	effective transverse shear material thickness
$[G_3]$	=	effective transverse shear material matrix

If you use the PSHELL entry, you can directly input G_1 , G_2 , G_4 , T , G_3 , and T_s (G_1 =MID1, G_2 =MID2, G_3 =MID3, G_4 =MID4 on the PSHELL entry) If you use the PCOMP entry, you can have NX Nastran calculate the composite equivalent material matrices from the data you supply.

Equivalent thermal properties are determined as follows:

$$\begin{Bmatrix} \alpha_{\varepsilon_0} \\ \alpha_{\chi} \\ \alpha_{\chi\varepsilon_0} \end{Bmatrix} = \begin{Bmatrix} T\alpha_1 \\ \frac{T^3}{12}\alpha_2 \\ T^2\alpha_3 \end{Bmatrix} = \begin{bmatrix} A & B & 0 \\ B & D & 0 \\ 0 & 0 & T_sG_3 \end{bmatrix}^{-1} \begin{Bmatrix} G\alpha_{\varepsilon_0} \\ G\alpha_{\chi} \\ G\alpha_{\chi\varepsilon_0} \end{Bmatrix}$$

Equation 24-10.

The following coefficients are used to determine equivalent thermal properties:

$$\left\{ G\alpha_{\varepsilon_0} \right\} = \sum_{k=1}^N [G]_k \{ \alpha \}_k (z_k - z_{k-1})$$

$$\{ G\alpha_{\chi} \} = \frac{1}{3} \sum_{k=1}^N [G]_k \{ \alpha \}_k (z_k^3 - z_{k-1}^3)$$

$$\left\{ G\alpha_{\chi\varepsilon_0} \right\} = \frac{1}{2} \sum_{k=1}^N [G]_k \{ \alpha \}_k (z_k^2 - z_{k-1}^2)$$

These coefficients are used to calculate the equivalent thermal properties as follows:

$$\left\{ \alpha_{\varepsilon_0} \right\} = [A]^{-1} \left\{ G\alpha_{\varepsilon_0} \right\}$$

and

$$\{ \alpha_{\chi} \} = [D]^{-1} \{ G\alpha_{\chi} \}$$

where $\{\alpha_{\varepsilon_0}\}$ and $\{\alpha_{\chi}\}$ are the membrane and bending equivalent thermal properties. Note that $\{\alpha_0\}$ is not directly calculated, but is determined from $\{G\alpha_0\}$ when the PCOMP input is used when the MID4 field on the PSHELL is > 400,000,000. Note that $\{G\alpha_{\chi\varepsilon_0}\}$ cannot be input directly using PSHELL and $\{\alpha_0\}$ can be input only if $[B]$ is invertible (which is generally not true).

The thermal expansion relationships in the shell element formulation take the following form:

$$\alpha_{\varepsilon_0} = T\alpha_1$$

$$\alpha_{\chi} = \frac{T^3}{12}\alpha_2$$

$$\alpha_{\chi\varepsilon_0} = T^2\alpha_3$$

where α_1 , α_2 , and α_3 are the thermal expansion inputs on the materials referenced by the MID1, MID2, and MID4 fields on the PSHELL entry. If you use PCOMP, NX Nastran automatically calculates these relationships.

The terms G_1 , G_2 , and G_4 are defined by the following integrals:

$$\begin{aligned} G_1 &= \frac{1}{T} \int [G_e] dz \\ G_2 &= \frac{1}{I} \int z^2 [G_e] dz \\ G_4 &= \frac{1}{T^2} \int (-z) [G_e] dz \end{aligned}$$

Equation 24-11.

The limits on the integration are from the bottom surface to the top surface of the laminated composite. The matrix of material moduli, $[G_e]$, has the following form for isotropic materials:

$$[G_e]_I = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{\nu E}{1-\nu^2} & 0 \\ \frac{\nu E}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{bmatrix}$$

Equation 24-12.

For orthotropic materials, the matrix, $[G_e]$, is written as follows:

$$[G_e]_0 = \begin{bmatrix} \frac{E_1}{1-\nu_1\nu_2} & \frac{\nu_1 E_2}{1-\nu_1\nu_2} & 0 \\ \frac{\nu_2 E_1}{1-\nu_1\nu_2} & \frac{E_2}{1-\nu_1\nu_2} & 0 \\ 0 & 0 & G_{12} \end{bmatrix}$$

Equation 24-13.

Here, $\nu_1 E_2 = \nu_2 E_1$ to satisfy the requirement that the matrix of elastic moduli be symmetric. In general, the analyst may supply element properties with respect to a particular orientation which does not necessarily correspond to the principal material axes. In this case, the analyst must also supply the value of the angle, θ or material coordinate system that orients the element material axis relative to the side G1-G2 of the element. The material elastic modulus matrix is then transformed by the program into the element modulus matrix through the relation

$$[G_e] = [U]^t [G_m] [U]$$

Equation 24-14.

where:

$$[U] = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & \cos \theta \sin \theta \\ \sin^2 \theta & \cos^2 \theta & -\cos \theta \sin \theta \\ -2 \cos \theta \sin \theta & 2 \cos \theta \sin \theta & \cos^2 \theta - \sin^2 \theta \end{bmatrix}$$

Equation 24-15.

The finite element model for a structure composed of composite materials requires the evaluation of the matrix of elastic moduli for each plate element of the model. The characteristics of the composite media are totally contained in these matrices.

To illustrate evaluation of these matrices, consider the cross-ply laminates shown in **Figure 24-2**. Here, all three configurations are represented by a single quadrilateral plate element. The coordinate axes are coincident with the element coordinate axes. Then, if we assume that each lamina of the n-ply laminates is of thickness T/n , where T is the total thickness of each of three configurations, the matrices of elastic moduli may be evaluated from the following relations:

$$[G_1] = \frac{1}{T} \left\{ \begin{array}{ccc} -\frac{T}{2} + \frac{T}{n} & -\frac{T}{2} + \frac{2T}{n} & \frac{T}{2} \\ \int_{-\frac{T}{2}}^{\frac{T}{2}} [G_e]_1 z^2 dz + \int_{-\frac{T}{2} + \frac{T}{n}}^{\frac{T}{2} + \frac{T}{n}} [G_e]_2 dz + \dots + \int_{-\frac{T}{2} + \frac{(n-1)T}{n}}^{\frac{T}{2} + \frac{(n-1)T}{n}} [G_e]_n dz \end{array} \right\}$$

Equation 24-16.

$$[G_2] = \frac{1}{T} \left\{ \begin{array}{ccc} -\frac{T}{2} + \frac{T}{n} & -\frac{T}{2} + \frac{2T}{n} & \frac{T}{2} \\ \int_{-\frac{T}{2}}^{\frac{T}{2}} [G_e]_1 dz + \int_{-\frac{T}{2} + \frac{T}{n}}^{\frac{T}{2} + \frac{T}{n}} [G_e]_2 z^2 dz + \dots + \int_{-\frac{T}{2} + \frac{(n-1)T}{n}}^{\frac{T}{2} + \frac{(n-1)T}{n}} [G_e]_n z^2 dz \end{array} \right\}$$

Equation 24-17.

$$[G_4] = \frac{1}{T^2} \left\{ \begin{array}{ccc} -\frac{T}{2} + \frac{T}{n} & -\frac{T}{2} + \frac{2T}{n} & \\ \int_{-\frac{T}{2}}^{\frac{T}{2}} [G_e]_1 (-z) dz + \int_{-\frac{T}{2} + \frac{T}{n}}^{\frac{T}{2} + \frac{T}{n}} [G_e]_2 (-z) dz & & \\ & & \frac{T}{2} \\ + \dots + \int_{-\frac{T}{2} + \frac{(n-1)T}{n}}^{\frac{T}{2} + \frac{(n-1)T}{n}} [G_e]_n (-z) dz & & \end{array} \right\}$$

Equation 24-18.

These relations reflect the assumption that the xy-plane of the element coordinate system is coincident with the geometric middle plane of the laminate. The xy-plane of the element coordinate system is defined in the mean plane of the element so that any offset between the mean plane of the connected grid points and the geometric middle plane of the laminate would be reflected in the integration limits of the preceding relations.

Note that $I = T^3/12$ in the evaluation of $[G_2]$, i.e., the value $(12I)/T^2$, will be assigned the default value of 1.0 on the plate element property entry.

The matrix of elastic moduli for transverse shear, $[G_3]_m$ is defined as a two-by-two matrix of the form

$$[G_3]_m = \begin{bmatrix} G_{11} & G_{12} \\ G_{21} & G_{22} \end{bmatrix}$$

Equation 24-19.

and the corresponding matrix transformed into an element coordinate system is given by

$$[G_3]_e = [W]^T [G_3]_m [W]$$

Equation 24-20.

$$[W] = \begin{bmatrix} \cos\theta & \sin\theta \\ -\sin\theta & \cos\theta \end{bmatrix}$$

where

The mean value of the transverse shear modulus \bar{G} for the laminated composite is defined in terms of the transverse shear strain energy, U , through the depth

$$U = \frac{1}{2} \frac{V^2}{\bar{G}T} = \frac{1}{2} \int \frac{(\tau(z))^2}{G(z)} dz$$

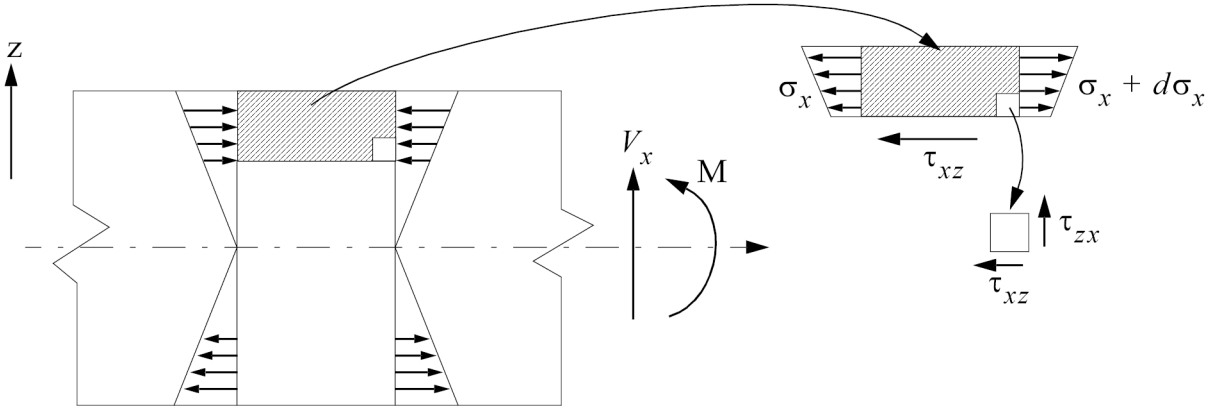
Equation 24-21.

A unique mean value of the transverse shear strain is assumed to exist for both the x and y components of the element coordinate system, but for ease of discussion, only the evaluation of an uncoupled x component of the shear moduli will be illustrated here. From [Eq. 24-21](#) the mean value of transverse shear modulus may be written in the following form

$$\frac{1}{G_x} = \frac{T}{V_x^2} \sum_{i=1}^N \int_{z_{i-1}}^{z_i} \frac{(\tau_{zx}(z))^2}{(G_x)_i} dz$$

Equation 24-22.

where G is an “average” transverse shear coefficient used by the element code and $(G_x)_i$ is the local shear coefficient for layer i . To evaluate [Eq. 24-13](#), it is necessary to obtain an expression for $(\tau_{zx}(z))$. This can be accomplished by assuming that the x- and y-components of stress are decoupled from one another. This assumption allows the desired equation to be deduced through an examination of a beam unit cross-sectional width.



The equilibrium conditions in the horizontal direction and for total moment are

$$\frac{\partial \tau_{xz}}{\partial z} + \frac{\partial \sigma_x}{\partial x} = 0$$

Equation 24-23.

$$V_x + \frac{\partial M_x}{\partial x} = 0$$

Equation 24-24.

Now, if the location of the neutral surface is denoted by \bar{z}_x and ρ is the radius of curvature of the beam, the axial stress E_x may be expressed in the form

$$\sigma_x + \frac{E_x(\bar{z}_x - z)}{(\bar{EI})_x} = 0$$

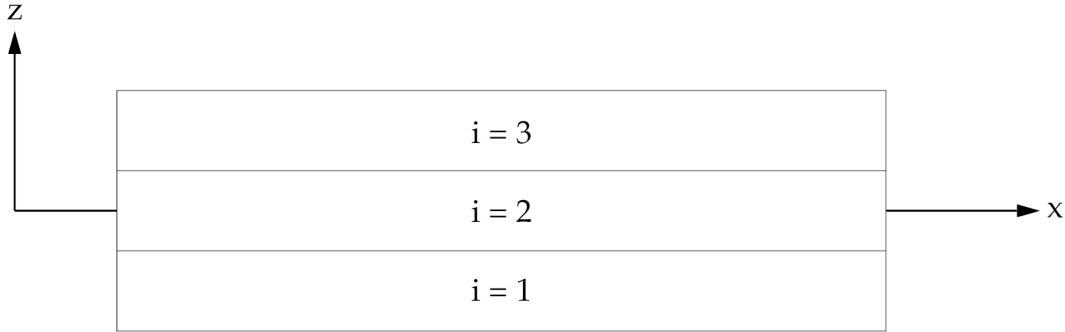
Equation 24-25.

Relation 20 may be differentiated with respect to x combined with [Eq. 24-23](#) and [Eq. 24-24](#). In a region of constant E_x the result may be integrated to yield the following expression

$$\tau_{xz} = C_i + \frac{V_x}{(\bar{EI})_x} \left(\bar{z}_x z - \frac{z^2}{2} \right) E_{xi} \quad z_{i-1} < z < z_i$$

Equation 24-26.

Eq. 24-30 is particularly convenient to use in the analysis of n-ply laminates because sufficient conditions exist to determine the constants C_i ($i = 1, 2, \dots, n$) and the “directional bending center” \bar{z}_x . For example, consider the following laminated configuration



At the bottom surface ($i = 1$, $z = z_0$, and $\tau_{xz} = 0$)

$$C_1 = \frac{-V_x}{(\bar{EI})_x} \left(\bar{z}_x z_0 - \frac{z_0^2}{2} \right) E_{x1}$$

Equation 24-27.

and for the first ply at the interface between plies $i = 1$ and $i = 2$ ($z = z_1$)

$$(\tau_{xz})_1 = + \frac{V_x}{(\bar{EI})_x} \left[\bar{z}_x (z_1 - z_0) \frac{1}{2} [z_1^2 - z_0^2] \right] E_{x1}$$

Equation 24-28.

At this interface between plies $i = 1$ and $i = 2$,

$$(\tau_{xz})_2 = C_2 + \frac{V_x}{(\bar{EI})_x} \left(\bar{z}_x z_1 - \frac{z_1^2}{2} \right) E_{x2}$$

Equation 24-29.

and as $(\tau_{xz})_2 = (\tau_{xz})_1$ at $z = z_1$,

$$C_2 = (\tau_{xz})_1 - \frac{V_x E_{x2}}{(\bar{EI})_x} \left[\bar{z}_x z_1 - \frac{1}{2} z_1^2 \right]$$

Equation 24-30.

Then, in the ply $z_1 < z < z_2$ the shear is

$$\tau_{xz}(z) = (\tau_{xz})_1 \frac{V_x E_{x2}}{(\overline{EI})_x} \left[\dot{z}_x (z - z_1) - \frac{1}{2} (z^2 - z_1^2) \right]$$

Equation 24-31.

In general, for any ply, $z_{i-1} < z < z_i$, the shear is

$$\tau_{xz}(z) = (\tau_{xz})_{i-1} \frac{V_x E_{xi}}{(\overline{EI})_x} \left[\dot{z}_x (z - z_{i-1}) - \frac{1}{2} (z^2 - (z_{i-1})^2) \right]$$

Equation 24-32.

At any ply interface, z_i , the shear is therefore

$$(\tau_{xz})_i = \frac{V_x}{(\overline{EI})_{x_j=1}} \sum_{j=1}^i E_{xj} T_j \left[\dot{z} - \frac{1}{2} (z_j + z_{j-1}) \right]$$

Equation 24-33.

where $T_j = z_j - (z_{j-1})$.

Note that the shear at the top face, $(\tau_{xz})_n$, is zero and therefore

$$(\tau_{xz})_n = \frac{V_x}{(\overline{EI})} \left[\dot{z}_x \sum_{j=1}^n E_{xj} T_j - \sum_{j=1}^n E_{xj} T_j (z_j + z_{j-1}) / 2 \right] = 0$$

Equation 24-34.

Eq. 24-34 proves that if \bar{z}_x is the bending center, the shear at the top surface must be zero.

Eq. 24-32 could be substituted into Eq. 24-23 and integrated. A better form of Eq. 24-34, for this purpose is

$$(\tau_{xz}(z))_i = \frac{V_x E_{xi}}{(\overline{EI})_x} \left[f_{xi} + \dot{z}_x (z - z_{i-1}) \frac{1}{2} (z^2 - (z_{i-1})^2) \right]$$

Equation 24-35.

where

$$f_{xi} = \frac{1}{E_{xi}} \sum_{j=1}^{i-1} E_{xj} T_j \left[\dot{z}_x - \frac{1}{2} (z_j + z_{j-1}) \right]$$

Equation 24-36.

Substituting Eq. 24-35 into Eq. 24-26 and integrating the results, we obtain

$$\frac{1}{G_x} = \frac{T}{(\overline{EI})_x^2} \sum_{i=1}^N \frac{1}{G_{xi}} R_{xi}$$

Equation 24-37.

where

$$R_{xi} = (E_{xi})^2 T_i \left[\left\{ f_{xi} + (\dot{z}_x - z_{i-1}) T_i - \frac{1}{3} T_i^2 \right\} f_{xi} \right. \\ \left. + \left\{ \frac{1}{3} (\dot{z}_x - 2 z_{i-1}) - \frac{1}{4} T_i \right\} \dot{z}_x T_i^2 + \left\{ \frac{1}{3} z_{i-1}^2 + \frac{1}{4} z_{i-1} T_i + \frac{1}{20} T_i^2 \right\} T_i^2 \right]$$

Equation 24-38.

This expression for the inverse shear modulus for the x-direction may be generalized to provide for the calculation of each term in the two-by-two matrix of shear moduli.

$$[\bar{G}_{kl}] = \left[\frac{T}{(\bar{EI})_{kk}} \sum_{i=1}^n [G_{kl}^i]^{-1} R_{ki} \right]^{-1}$$

Equation 24-39.

where:

$$\begin{aligned} k &= 1,2 \\ l &= 1,2 \end{aligned}$$

Note that if no shear is given, $[G]^{-1}$, and also that in [Eq. 24-32](#)

where $[G_2]^*$ is calculated in the same manner as $[G_2]$ except that Poisson's Ratio is set to zero.

The moduli for individual plies are provided through user input because, in general, $G_{12} \neq G_{21}$ will be used for the coupling terms. Finally,

$$[G_3] = \begin{bmatrix} \bar{G}_{11} & (\bar{G}_{12})_{\text{avg}} \\ (\bar{G}_{12})_{\text{avg}} & \bar{G}_{22} \end{bmatrix}$$

Equation 24-40.

As an example, consider a single layer element. For this case let $z_{i-1} = -T/2$, $\bar{z} = 0$, $f_0 = 0$, and $EI = ET^3/12$. Evaluating [Eq. 24-39](#) we obtain

$$R_i = E^2 T^5 \left[\frac{1}{12} - \frac{1}{8} + \frac{1}{20} \right] = \frac{E^2 T^5}{120}$$

Equation 24-41.

and

$$\frac{1}{G} = \left(\frac{(12)^2 T}{E^2 T^6} \right) \cdot \left(\frac{E^2 T^5}{120 G_1} \right) = \frac{6}{5 G_1}$$

Equation 24-42.

which provides the correct factor for a nonuniform shear distribution in a plate.

The coefficients of thermal expansion derived for membrane-bending coupling, which appear in the A1, A2, and A12 fields of MAT2 entry and correspond to the MID4 Field on the PSHELL, require special interpretation. They are given by:

$$\{\alpha_{MAT2}\} = [G_{ij_{MAT2}}]\{\alpha_{ACTUAL}\}$$

Equation 24-43.

To obtain the actual values of A1, A2, and A12 $\{\alpha_{MAT2}\}$, you must solve [Eq. 24-43](#).

24.3 Laminate Failure Indices and Strength Ratios

When you use the shell composites (PCOMP/PCOMPG) or solid composites (PCOMPS), you can optionally output a failure index and strength ratio for individual laminae and for the laminate bonding material.

Failure indices assume a value of one on the periphery of a failure surface in stress space.

- If the failure index is < 1 , the lamina stress is interior to the periphery of the failure surface and the lamina is predicted to be safe.
- If the failure index is > 1 , the lamina stress is exterior to the periphery of the failure surface and the lamina is predicted to have failed.

The failure indices represent a phenomenological failure criterion in that only an occurrence of a failure is indicated and not the mode of failure.

Strength ratio is a more direct indicator of failure than the failure index since it demonstrates the percentage of applied load to the failure criteria. Strength ratio is defined as:

Strength Ratio (SR) = Allowable Stress / Calculated Stress

For example, a SR = 0.75 not only indicates that a failure has occurred, but also indicates that the applied load is 25% beyond the allowable. A FI = 1.25 on the other hand does not represent a percentage of failure; only that a failure condition exists.

See the remarks on the PCOMP, PCOMPG, and PCOMPS bulk entries in the *NX Nastran Quick Reference Guide* for the input requirements needed to request failure indices and strength ratio output.

Failure Theories for Shell Composites (PCOMP, PCOMPG)

You can use the FT field on the PCOMP or PCOMPG entries to request a failure theory which applies to all plies. The allowable entries in the FT field are listed below.

“HILL” for the Hill failure theory.

“HOFF” for the Hoffman failure theory.

“TSAI” for the Tsai-Wu failure theory.

“STRN” for the Maximum Strain failure theory.

Note: If you specify a failure theory in the FT field on the PCOMP or PCOMPG entries, you must also specify the allowable shear stress of the bonding material in the SB field.

Hill Theory (HILL) for Shell Composites

The Hill failure theory for orthotropic materials that have the same strength in tension and compression, i.e., $x_t = x_c$ and $y_t = y_c$ can be expressed as:

$$\text{Failure Index} = \frac{\sigma_1^2}{X^2} - \frac{\sigma_1 \sigma_2}{X^2} + \frac{\sigma_2^2}{Y^2} + \frac{\tau_{12}^2}{S^2}$$

Equation 24-44.

X is allowable stress in 1-direction

Y is allowable stress in 2-direction

S is allowable stress in shear

and $X = X_t$ if σ_1 is positive or $X = X_c$ if σ_1 is negative and similarly for Y and σ_2 . For the interaction term $\sigma_1 \sigma_2 / X^2$, $X = X_t$ if σ_1 is positive or $X = X_c$ if σ_1 is negative.

A plot of the above equation obtained by setting the failure index to 1 on the σ_1 - σ_2 plane yields an ellipse and is the anisotropic yield criterion of Hill (modified later by Tsai, and hence also sometimes known as the Tsai-Hill theory). Therefore, if the failure index so calculated is less than 1, the ply stresses are inside the yield ellipse and the ply is said to be "safe"; conversely, if the failure index is greater than 1, the ply stresses are outside the yield ellipse and the ply has failed.

Replacing the applied stress with (SR · applied stress), the Hill Failure Criteria can be rewritten in terms of a strength ratio:

$$1.0 = \left[(SR^2 \sigma_1^2) \left(\frac{1}{X^2} \right) \right] + \left[(SR^2 \sigma_2^2) \left(\frac{1}{Y^2} \right) \right] - \left[\frac{(SR^2 \sigma_1 \sigma_2)}{(X^2)} \right] + \left[\frac{(SR^2 \tau_{12}^2)}{(S^2)} \right]$$

The above equation can be rearranged into quadratic equation format:

$$aSR^2 + bSR + c = 0$$

giving:

$$\left[\left(\frac{\sigma_1^2}{X^2} \right) + \left(\frac{\sigma_2^2}{Y^2} \right) - \left(\frac{\sigma_1 \sigma_2}{X^2} \right) + \left(\frac{\tau_{12}^2}{S^2} \right) \right] SR^2 - 1 = 0$$

The value of a in the quadratic equation format matches the value for the Hill failure index (FI).

Substituting the values of $a = FI$, $b = 0$, and $c = -1$ into the general quadratic equation solution:

$$SR = \frac{-b + \sqrt{b^2 - 4ac}}{2a}$$

gives the Hill strength ratio:

$$SR = \frac{1}{\sqrt{FI}}$$

Hoffman Failure Theory (HOFF) for Shell Composites

The Hoffman theory for an orthotropic lamina in a general state of plane stress with unequal tensile and compressive strengths is given by

$$\left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_2 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\sigma_{12}^2}{S^2} - \frac{\sigma_1 \sigma_2}{X_t X_c} = 1$$

Equation 24-45.

The failure index is obtained by evaluating the left-hand side of the above equation.

Note that this theory takes into account the difference in tensile and compressive allowable stresses by using linear terms in the equation.

To calculate the strength ratio, the following terms are defined:

$$\begin{aligned} F_1 &= \frac{1}{X_t} - \frac{1}{X_c} \\ F_2 &= \frac{1}{Y_t} - \frac{1}{Y_c} \\ F_{11} &= \frac{1}{X_t X_c} \\ F_{22} &= \frac{1}{Y_t Y_c} \\ F_{66} &= \frac{1}{S^2} \end{aligned}$$

Substituting above terms into Hoffman FI equation and setting FI = 1:

$$F_1 \sigma_1 + F_2 \sigma_2 + F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + F_{66} \sigma_{12}^2 - F_{11} \sigma_1 \sigma_2 = 1$$

Replacing the applied stress with (SR · applied stress), the Hoffman Failure Criteria can be rewritten in terms of a strength ratio:

$$F_1 \text{SR} \sigma_1 + F_2 \text{SR} \sigma_2 + F_{11} \text{SR}^2 \sigma_1^2 + F_{22} \text{SR}^2 \sigma_2^2 + F_{66} \text{SR}^2 \sigma_{12}^2 - F_{11} \text{SR}^2 \sigma_1 \sigma_2 = 1$$

Rearranging into quadratic equation format:

$$[F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + F_{66} \sigma_{12}^2 - F_{11} \sigma_1 \sigma_2] \text{SR}^2 + [F_1 \sigma_1 + F_2 \sigma_2] \text{SR} - 1 = 0$$

Using the general quadratic equation solution:

$$\text{SR} = \frac{-b + \sqrt{b^2 - 4ac}}{2a}$$

The Hoffman strength ratio is calculated using the following roots:

$$a = [F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + F_{66} \sigma_{12}^2 - F_{11} \sigma_1 \sigma_2]$$

$$\begin{aligned}b &= [F_1\sigma_1 + F_2\sigma_2] \\c &= -1\end{aligned}$$

Tensor Polynomial Theory of Tsai-Wu (TSAI) for Shell Composites

The theory of strength for anisotropic materials proposed by Tsai and Wu specialized to the case of an orthotropic lamina in a general state of plane stress with unequal tensile and compressive strengths is

$$F_1\sigma_1 + F_2\sigma_2 + F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + 2F_{12}\sigma_1\sigma_2 + F_{66}\sigma_{12}^2 = 1$$

Equation 24-46.

where:

$$\begin{aligned}F_1 &= \frac{1}{X_t} - \frac{1}{X_c} \\F_2 &= \frac{1}{Y_t} - \frac{1}{Y_c} \\F_{11} &= + \frac{1}{X_t X_c} \\F_{22} &= + \frac{1}{Y_t Y_c} \\F_{66} &= + \frac{1}{S^2}\end{aligned}$$

and F_{12} is to be evaluated experimentally.

The magnitude of F_{12} is, however, constrained by the following inequality called a “stability criterion”:

$$F_{11}F_{22} - F_{12}^2 > 0$$

Equation 24-47.

The necessity of satisfying the stability criterion, together with the requirement that F_{12} be determined experimentally from a combined stress state, poses difficulties in the use of this theory. Narayanaswami and Adelman¹ have suggested that F_{12} be set to zero and that use of Hoffman's theory or Tensor Polynomial theory with $F_{12} = 0$ is a preferred alternative to the experimental determination of F_{12} . If you have a value for use with F_{12} in the theory, you can input that value in the MAT8 Bulk Data entry; otherwise, with this failure theory, NX Nastran sets F_{12} to 0.0. The left-hand side of the above equation will be evaluated as the failure index by this theory.

1. Narayanaswami, R., and H. M. Adelman, “Evaluation of the Tensor Polynomial and Hoffman Strength Theories for Composite Materials,” *Journal of Composite Materials*, Vol. II, 1977, p. 366.

Replacing the applied stress with (SR · applied stress), the Hoffman Failure Criteria can be rewritten in terms of a strength ratio:

$$F_1 SR \sigma_1 + F_2 SR \sigma_2 + F_{11} SR^2 \sigma_1^2 + F_{22} SR^2 \sigma_2^2 + 2F_{12} SR^2 \sigma_1 \sigma_2 + F_{66} SR^2 \sigma_{12}^2 = 1$$

Rearranging into quadratic equation format:

$$[F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + 2F_{12} \sigma_1 \sigma_2 + F_{66} \sigma_{12}^2] SR^2 + [F_1 \sigma_1 + F_2 \sigma_2] SR - 1 = 0$$

Using the general quadratic equation solution:

$$SR = \frac{-b + \sqrt{b^2 - 4ac}}{2a}$$

The TSAI-Wu strength ratio is calculated using the following roots:

$$\begin{aligned} a &= [F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + 2F_{12} \sigma_1 \sigma_2 + F_{66} \sigma_{12}^2] \\ b &= [F_1 \sigma_1 + F_2 \sigma_2] \\ c &= -1 \end{aligned}$$

Maximum Strain Theory (STRN) for Shell Composites

The midplane strains and curvatures are available in the element coordinate system. From these, the stresses and strains in each individual lamina along the fiber direction and transverse direction can be easily calculated. You can use the STRAIN case control command to request the output of lamina strains.

The maximum strain criteria has no strain interaction terms. The strain allowables specified on the MAT8 entry for each lamina include

X_t, X_c	Allowable strains in tension and compression, respectively, in the longitudinal direction.
Y_t, Y_c	Allowable strains in tension and compression, respectively, in the transverse direction.
S	Allowable strain for inplane shear.

The failure index is calculated using

$$\begin{aligned} \frac{\varepsilon_1}{X_t} \text{ (or } \frac{\varepsilon_1}{X_c} \text{ if } \varepsilon_1 \text{ compressive)} \\ \frac{\varepsilon_2}{X_t} \text{ (or } \frac{\varepsilon_2}{Y_c} \text{ if } \varepsilon_2 \text{ compressive)} \end{aligned}$$

and $\left| \frac{\gamma_{12}}{S} \right|$ i.e.,

the Failure Index = $\text{MAX} \left[\left(\frac{\varepsilon_1}{X} \right), \left(\frac{\varepsilon_2}{Y} \right), \left(\frac{|\gamma_{12}|}{S} \right) \right]$ where $\varepsilon_1, \varepsilon_2, \gamma_{12}$ are the elastic strains (total strains minus thermal strains).

Because you must understand which mode of failure index is critical; i.e., longitudinal (1), transverse (2), or shear (12), NX Nastran prints the mnemonic 1, 2 or 12 alongside the FP value to indicate the critical direction.

Note: There is no change in the way the failure index is calculated for interlaminar shear stresses.

When you use the maximum strain theory, you may want to specify lamina stress allowables instead of strain allowables on the MAT8. To do this, leave the STRN field on the MAT8 entry blank.

For this case, NX Nastran calculates the failure indices using

$$\frac{\varepsilon_1 \cdot E_{11}}{X_t} \text{ (or } \frac{\varepsilon_1 \cdot E_{11}}{X_c} \text{ if } \varepsilon_1 \text{ compressive)}$$

$$\frac{\varepsilon_2 \cdot E_{22}}{Y_c} \text{ (or } \frac{\varepsilon_2 \cdot E_{22}}{Y_c} \text{ if } \varepsilon_2 \text{ compressive)}$$

and

$$\frac{|\gamma_{12}| \cdot G_{12}}{S}$$

that is,

$$\text{the Failure Index} = \text{MAX} \left[\left(\frac{\varepsilon_1 \cdot E_{11}}{X} \right), \left(\frac{\varepsilon_2 \cdot E_{22}}{Y} \right), \left(\frac{|\gamma_{12}| \cdot G_{12}}{S} \right) \right]$$

To calculate the Maximum Stress (Strain) strength ratio, the failure index which is defined as FI = Calculated Stress / Allowable Stress is set to unity, and the applied stress is replaced with (SR · applied stress). The result is:

$$\text{SR} = \frac{1}{\text{FI}}$$

Failure Theories for Solid Composites (PCOMPS)

To request that the software compute a ply and/or bonding failure index for solid composites, the STRESS case control command must be present, SOUTi on the PCOMPS bulk entry must be set to "YES", and the following must be defined.

- For a stress or strain ply failure index, you must define the FTi field on the PCOMPS entry to request a specific failure theory for each ply. In addition, the stress or strain allowables must be defined on the referenced MATFT bulk entry. The allowable entries in the FTi field are as follows.

"HILL" for the Hill failure theory.

"HOFF" for the Hoffman failure theory.

"TSAI" for the Tsai-Wu failure theory.

“STRN” for the Maximum Strain failure theory.

“STRS” for the Maximum Stress failure theory.

“TS” for the Maximum Transverse Shear Stress failure theory.

The failure theories are applied at each position that stress output is requested using the CPLYMID, CPLYBT, or CPLYBMT descriptors on the STRESS case control command. Thus, for the solid composite, you can evaluate the potential for failure at the top, middle, and bottom of each ply. By default, the failure index output prints in the f06 file even when using the PLOT or PUNCH descriptors on the STRESS and STRAIN case control commands. The parameter entry PARAM,NOFISR,1 can be used to turn off the printing of the failure index output.

Hill Theory for Solid Composites

$$F = \left[\frac{\sigma_1^2}{X^2} + \frac{\sigma_2^2}{Y^2} + \frac{\sigma_3^2}{Z^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2} \right) \sigma_1 \sigma_2 - \left(-\frac{1}{X^2} + \frac{1}{Y^2} + \frac{1}{Z^2} \right) \sigma_2 \sigma_3 - \left(\frac{1}{X^2} - \frac{1}{Y^2} + \frac{1}{Z^2} \right) \sigma_1 \sigma_3 + \frac{\sigma_{12}^2}{S_{12}^2} + \frac{\sigma_{13}^2}{S_{13}^2} + \frac{\sigma_{23}^2}{S_{23}^2} \right]$$

where,

X is the allowable stress in the 1-direction,

Y is the allowable stress in the 2-direction,

Z is the allowable stress in the 3-direction,

S12 is the maximum allowable in-plane shear stress,

S13 is the maximum allowable 23 shear stress, and

S13 is the maximum allowable 13 shear stress.

Hoffman Failure Theory for Solid Composites

$$F = C_1 (\sigma_2 - \sigma_3)^2 + C_2 (\sigma_3 - \sigma_1)^2 + C_3 (\sigma_1 - \sigma_2)^2 + C_4 \sigma_1 + C_5 \sigma_2 + C_6 \sigma_3 + C_7 \tau_{23}^2 + C_8 \tau_{13}^2 + C_9 \tau_{12}^2$$

$$C_1 = \frac{1}{2} \left(\frac{1}{Z_t Z_c} + \frac{1}{Y_t Y_c} - \frac{1}{X_t X_c} \right)$$

$$C_2 = \frac{1}{2} \left(\frac{1}{X_t X_c} + \frac{1}{Z_t Z_c} - \frac{1}{Y_t Y_c} \right)$$

$$C_3 = \frac{1}{2} \left(\frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c} \right)$$

$$C_4 = \frac{1}{X_t} - \frac{1}{X_c}$$

$$C_5 = \frac{1}{Y_t} - \frac{1}{Y_c}$$

$$C_6 = \frac{1}{Z_t} - \frac{1}{Z_c}$$

$$C_7 = \frac{1}{s_{23}^2}$$

$$C_8 = \frac{1}{s_{13}^2}$$

$$C_9 = \frac{1}{s_{12}^2}$$

where,

Xt, Xc are the maximum allowable stresses in the 1-direction in tension and compression,

Yt, Yc are the maximum allowable stresses in the 2-direction in tension and compression,

Zt, Zc are the maximum allowable stresses in the 3-direction in tension and compression,

S12 is the maximum allowable in-plane shear stress,

S23 is the maximum allowable 23 shear stress, and

S13 is the maximum allowable 13 shear stress.

Tsai-Wu Failure Theory for Solid Composites

$$F = \left(\frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_2 + \left(\frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_3 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\sigma_3^2}{Z_t Z_c} + \frac{\tau_{12}^2}{S_{12}^2} + \frac{\tau_{23}^2}{S_{23}^2} + \frac{\tau_{13}^2}{S_{13}^2} + 2F_{12}\sigma_1\sigma_2 + 2F_{23}\sigma_2\sigma_3 + 2F_{13}\sigma_1\sigma_3$$

where,

Xt, Xc are the maximum allowable stresses in the 1-direction in tension and compression,

Yt, Yc are the maximum allowable stresses in the 2-direction in tension and compression,

Zt, Zc are the maximum allowable stresses in the 3-direction in tension and compression,

S12 is the maximum allowable in-plane shear stress,

S23 is the maximum allowable 23 shear stress S13 is the maximum allowable 13 shear stress,

F12 is the interactive strength constant for the 12 plane,

F23 is the interactive strength constant for the 23 plane, and

F13 is the interactive strength constant for the 13 plane.

Maximum Strain Theory for Solid Composites

The software reports the maximum from the following six computations.

$$1. \quad \text{If } \varepsilon_1 > 0, \quad F = \left(\frac{\varepsilon_1}{X_t} \right)$$

$$\text{If } \varepsilon_1 < 0, \quad F = \left(\frac{\varepsilon_1}{X_c} \right)$$

$$2. \quad \text{If } \varepsilon_2 > 0, \quad F = \left(\frac{\varepsilon_2}{Y_t} \right)$$

$$\text{If } \varepsilon_2 < 0, \quad F = \left(\frac{\varepsilon_2}{Y_c} \right)$$

$$3. \quad \text{If } \varepsilon_3 > 0, \quad F = \left(\frac{\varepsilon_3}{Z_t} \right)$$

$$\text{If } \varepsilon_3 < 0, \quad F = \left(\frac{\varepsilon_3}{Z_c} \right)$$

$$4. \quad F = \left(\left| \frac{\gamma_{12}}{S_{12}} \right| \right)$$

$$5. \quad F = \left(\left| \frac{\gamma_{23}}{S_{23}} \right| \right)$$

$$6. \quad F = \left(\left| \frac{\gamma_{13}}{S_{13}} \right| \right)$$

where,

ε and γ are the elastic strains (total strains minus thermal strains),

X_t , X_c are the maximum allowable strains in the 1-direction in tension and compression,

Y_t , Y_c are the maximum allowable strains in the 2-direction in tension and compression,

Z_t , Z_c are the maximum allowable strains in the 3-direction in tension and compression,

S_{12} is the maximum allowable in-plane shear strain,

S_{23} is the maximum allowable 23 shear strain, and

S_{13} is the maximum allowable 13 shear strain.

Maximum Stress Theory for Solid Composites

The software reports the maximum from the following six computations.

$$1. \quad \text{If } \sigma_1 > 0, \quad F = \left(\frac{\sigma_1}{X_t} \right)$$

$$\text{If } \sigma_1 < 0, \quad F = \left(\frac{\sigma_1}{X_c} \right)$$

$$2. \quad \text{If } \sigma_2 > 0, \quad F = \left(\frac{\sigma_2}{Y_t} \right)$$

$$\text{If } \sigma_2 < 0, \quad F = \left(\frac{\sigma_2}{Y_c} \right)$$

$$3. \quad \text{If } \sigma_3 > 0, \quad F = \left(\frac{\sigma_3}{Z_t} \right)$$

$$\text{If } \sigma_3 < 0, \quad F = \left(\frac{\sigma_3}{Z_c} \right)$$

$$4. \quad F = \left(\left| \frac{\tau_{12}}{S_{12}} \right| \right)$$

$$5. \quad F = \left(\left| \frac{\tau_{23}}{S_{23}} \right| \right)$$

$$6. \quad F = \left(\left| \frac{\tau_{13}}{S_{13}} \right| \right)$$

where,

Xt, Xc are the maximum allowable stresses in the 1-direction in tension and compression,

Yt, Yc are the maximum allowable stresses in the 2-direction in tension and compression,

Zt, Zc are the maximum allowable stresses in the 3-direction in tension and compression,

S12 is the maximum allowable in-plane shear stress,

S23 is the maximum allowable 23 shear stress, and

S13 is the maximum allowable 13 shear stress.

Maximum Transverse Shear Failure Theory for Solid Composites

$$F_{13} = \frac{\tau_{13}}{S_{13}} \quad F_{23} = \frac{\tau_{23}}{S_{23}}$$

where,

S23 is the maximum allowable 23 shear stress, and

S13 is the maximum allowable 13 shear stress.

Bonding Material Failure Indices

To request the bonding failure index for shell composites (PCOMP,PCOMPG), you must define the allowable bonding stress (SB field on the PCOMP or PCOMPG entry). NX Nastran calculates the failure index of the bonding material (F_{SB}) as the maximum interlaminar shear stress divided by the allowable bonding stress.

$$F_{SB} = \frac{(\tau_{13}, \tau_{23})_{\max}}{SB}$$

As a result, the bonding failure index for the element is the largest value of the failure indices for all plies of the element. See [Calculating Interlaminar Shear Stress and Strains on Shell Composites \(PCOMP,PCOMPG\)](#).

To request the bonding failure index for solid composites (PCOMPS):

- You must define the ILFTi field on the PCOMPS entry to request the shear (SB) and/or normal (NB) inter-laminar failure index.
- You must define the allowable shear and/or normal bonding stress (SB and NB fields on the PCOMPS entry).
- You must include the CPLYBT or the CPLYBMT descriptors on the STRESS case control command to request stress output for the top and bottom of the laminate plies.

NX Nastran calculates the shear (F_{SB}) and normal (F_{NB}) failure indices of the bonding material as follows.

$$F_{SB} = \frac{(\sqrt{\tau_{13}^2 + \tau_{23}^2})_{\max}}{SB} \quad F_{NB} = \frac{(\sigma_{33})_{\max}}{NB}$$

Bonding Material Strength Ratios

To request the bonding material strength ratio (SR) for shell and solid composites, you must request the failure indices as described above, and you must define PARAM,SRCOMPS,YES. The bonding material strength ratio is computed as follows.

$$SR = \frac{1}{Failure\ Index}$$

24.4 Ply Stress and Strain Calculation for Shell Composites (PCOMP,PCOMPG)

The parameter NOCOMPS determines if stress and/or strain recovery is at the composite ply layers (default), on the equivalent PSHELL, or both. The STRESS and/or STRAIN case control commands are required for any of these recovery options. When ply results are requested, stress and/or strain are computed on every ply. To print the computed ply stress and/or strain results, the case control command request must include the "PRINT" option, and SOUTi=YES must be defined on the ply definition.

When ply results are requested, the element strains are transformed to ply strains in the element coordinate system by:

$$\{\varepsilon_{pxy}\} = \{\varepsilon_e\} - z\{\kappa_e\}$$

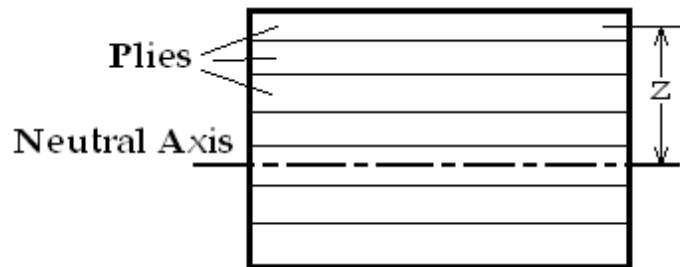
where:

$$\{\varepsilon_{pxy}\} = \text{ply strains } \{\varepsilon_{px}, \varepsilon_{py}, \gamma_{pxy}\}$$

$$\{\varepsilon_e\} = \text{element strains } \{\varepsilon_x, \varepsilon_y, \gamma_{xy}\}$$

z = distance from the neutral axis to the center of the ply

$$\{\kappa_e\} = \text{element curvatures } \{\kappa_x, \kappa_y, \kappa_{xy}\}$$



Note that the ply strains need to be rotated to the ply coordinate system using the following relation:

$$\{\varepsilon_p\} = [T]\varepsilon_{pxy}$$

where:

$$[T] = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & 2 \sin \theta \cos \theta \\ \sin^2 \theta & \cos^2 \theta & -2 \sin \theta \cos \theta \\ -\sin^2 \theta & \sin \theta \cos \theta & \cos^2 \theta - \sin^2 \theta \end{bmatrix}$$

$$\{\varepsilon_{pxy}\} = \text{ply strains } \{\varepsilon_1, \varepsilon_2, \gamma_{12}\}$$

The ply stresses are computed from the ply strains using Hooke's Law and the ply material constitutive matrix:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix}$$

The resulting ply stresses and strains are used in the composite failure theories.

24.5 Calculating Interlaminar Shear Stress and Strains on Shell Composites (PCOMP,PCOMPG)

The two-dimensional plate theory used in the CQUAD4, CQUAD8, CTRIA3 or CTRIA6 elements doesn't allow for the exact calculation of interlaminar stresses. High values of these interlaminar stresses can lead to failures that are unique to composite materials.

NX Nastran uses an approximate technique to determine the interlaminar shear stresses. The basic assumption in this approximate technique is that the x- and y-components of stress are decoupled from one another. The approximation consists of neglecting the effects of twisting moments in the computation of interlaminar shear stresses. For the majority of composite material structural analyses, this approximation generally yields satisfactory results for all elements in the model except those at the edges of the structure.

Eq. 24-33 is used to calculate $(\tau_{xz})_i$ for the i-th lamina and \bar{z}_x is calculated from Eq. 24-34. Similar expressions are used to calculate (τ_{yz}) and \bar{z}_y .

The interlaminar shear strains are calculated by

$$\gamma_{1z} = \frac{\tau_{1z}}{G_{1z}}$$

and

$$\gamma_{2z} = \frac{\tau_{2z}}{G_{2z}}$$

The interlaminar shear stress and strain is reported in the .f06 output according to the ply number. In the .f06 output, the interlaminar results reported for ply 1 are associated with the interface between plies 1 and 2, the results reported on ply 2 are associated with the interface between plies 2 and 3, and so on. The results reported on the final, or top ply will be zero since there is no interface.

24.6 Examples: Defining Laminate Material Properties

The following examples demonstrate how to use PSHELL and PCOMP to model a composite honeycomb section.

Example: Using PCOMP to Model a Honeycomb Plate

As an example of the PCOMP entry, consider the cantilevered honeycomb plate shown in Figure 24-4. Although the honeycomb structure is not considered a composite layup, it can be analyzed effectively using the PCOMP entry.

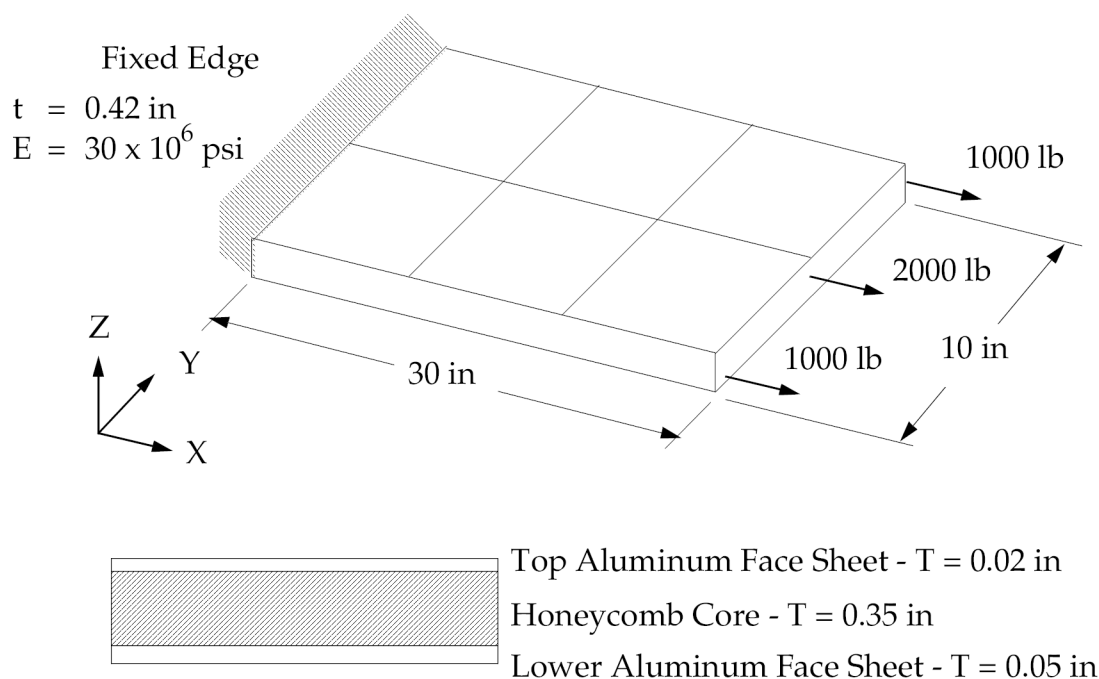


Figure 24-4. Honeycomb Cantilever Plate

The material properties of the honeycomb section are listed in [Table 24-2](#). Note that the face sheets are not of the same thickness. The result is a nonsymmetric plate ([Figure 24-5](#)), which can be noted by the MAT2 entry for the MID4 field of the equivalent PSHELL.

Table 24-2. Honeycomb Material Properties				
Material	Modulus of Elasticity (10⁶ psi)	Tensile Limit(10³ psi)	Compression Limit (10³ psi)	Shear Limit (10³ psi)
Aluminum Face Sheets	10.0	35	35	23
Core	0.0001	0.05	0.3	0.2
Bonding Material	—	—	—	0.1

Part of the input file for the honeycomb plate is shown in [Listing 24-1](#). The load on the plate is a 4000-pound uniform load acting in the plane of the plate. The output requests include the element force, laminae stress, laminae strain, and failure index table.

A selected portion of the output is shown in [Figure 24-5](#).

```

$
$ FILENAME - PCOMP1.DAT
$
CQUAD4  1      100      1      2      6      5
CQUAD4  2      100      2      3      7      6
CQUAD4  3      100      3      4      8      7
CQUAD4  4      100      5      6     10      9
CQUAD4  5      100      6      7     11     10
CQUAD4  6      100      7      8     12     11
$
FORCE   1      12      0      1.     1000.0  0.0    0.0
FORCE   1      4       0      1.     1000.0  0.0    0.0
FORCE   1      8       0      1.     2000.0  0.0    0.0
$
SPC1    1      123456  1      5      9
$
PCOMP   100
        120     0.05   0.     YES     STRN     130     0.35   0.0    YES
        120     0.02   0.     YES
$
MAT1    120     10.+6
        35.E3    35.E3    23.E3
MAT1    130     100.
        50.     300.     200.

```

Listing 24-1. Honeycomb Plate

```

*** USER INFORMATION MESSAGE 4379, THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING SHELL AND MAT2 CARDS.
PSHELL      100      100000100      4.2000E-01      200000100      1.0000E+00      300000100      1.0000E+00      0.0000E+00
      -2.1000E-01      2.1000E-01      400000100
MAT2      100000100      1.8316E+06      5.4948E+05      0.0000E+00      1.8316E+06      0.0000E+00      6.4106E+05      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      200000100      4.4895E+06      1.3469E+06      0.0000E+00      4.4895E+06      0.0000E+00      1.5713E+06      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      300000100      3.9189E+01      0.0000E+00      0.0000E+00      3.9189E+01      0.0000E+00      0.0000E+00      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0
MAT2      400000100      3.2705E+05      9.8115E+04      0.0000E+00      3.2705E+05      0.0000E+00      1.1447E+05      0.0000E+00
      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
      0

DISPLACEMENT VECTOR

POINT ID. TYPE T1 T2 T3 R1 R2 R3
      8 G 1.004209E-02 2.155610E-17 -3.155122E-01 -3.609077E-17 2.089842E-02 .0
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/04 PAGE 9

FORCES IN QUADRILATERAL ELEMENTS (QUAD4)

ELEMENT - MEMBRANE FORCES - - BENDING MOMENTS - - TRANSVERSE SHEAR FORCES -
ID FX FY FXY MX MY MXY QX QY
      4 2.000000E+02 2.784165E+01 -9.330303E+00 -4.593548E-15 -3.536165E-03 4.504795E-03 -7.892992E-17 -9.747001E-04
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/04 PAGE 10

HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/04 PAGE 11

STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRESSES IN FIBRE AND MATRIX DIRECTIONS INTER-LAMINAR STRESSES PRINCIPAL STRESSES (ZERO SHEAR) MAX
ID ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
      4 1 2.08187E+03 2.89631E+02 -9.68898E+01 -2.03948E-16 -2.51853E-03 -3.09 2.08710E+03 2.84409E+02 9.01344E+02
      4 2 3.49114E-02 4.86145E-03 -1.63057E-03 -2.03936E-16 -2.51839E-03 -3.10 3.49996E-02 4.77323E-03 1.51132E-02
      4 3 4.79470E+03 6.67919E+02 -2.24262E+02 -2.82614E-24 -3.48998E-11 -3.10 4.80685E+03 6.55768E+02 2.07554E+03
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/96 PAGE 12

FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT FAILURE PLY FP=FAILURE INDEX FOR PLY FB=FAILURE INDEX FOR BONDING FAILURE INDEX FOR ELEMENT FLAG
ID THEORY ID (DIRECT STRESSES/STRAINS) (INTER-LAMINAR STRESSES) MAX OF FP,FB FOR ALL PLIES
      4 STRAIN 1 .0570 -1 .0000
      2 .0007 -1 .0000
      3 .1313 -1 .1313
HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/04 PAGE 13

HONEYCOMB COMPOSITE CANTILEVER PLATE JULY 1, 2004 NX NASTRAN 6/28/04 PAGE 14

STRAINS IN LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT PLY STRAINS IN FIBRE AND MATRIX DIRECTIONS INTER-LAMINAR STRAINS PRINCIPAL STRAINS (ZERO SHEAR) MAX
ID ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
      4 1 1.99499E-04 -3.34931E-05 -2.51914E-05 -5.30264E-23 -6.54819E-10 -3.09 2.00177E-04 -3.41721E-05 2.34350E-04
      4 2 3.34529E-04 -5.61196E-05 -4.23949E-05 -5.30233E-18 -6.54781E-05 -3.10 3.35676E-04 -5.72665E-05 3.92943E-04
      4 3 4.59433E-04 -7.70492E-05 -5.83081E-05 -7.34797E-31 -9.07396E-18 -3.10 4.61012E-04 -7.86288E-05 5.39641E-04

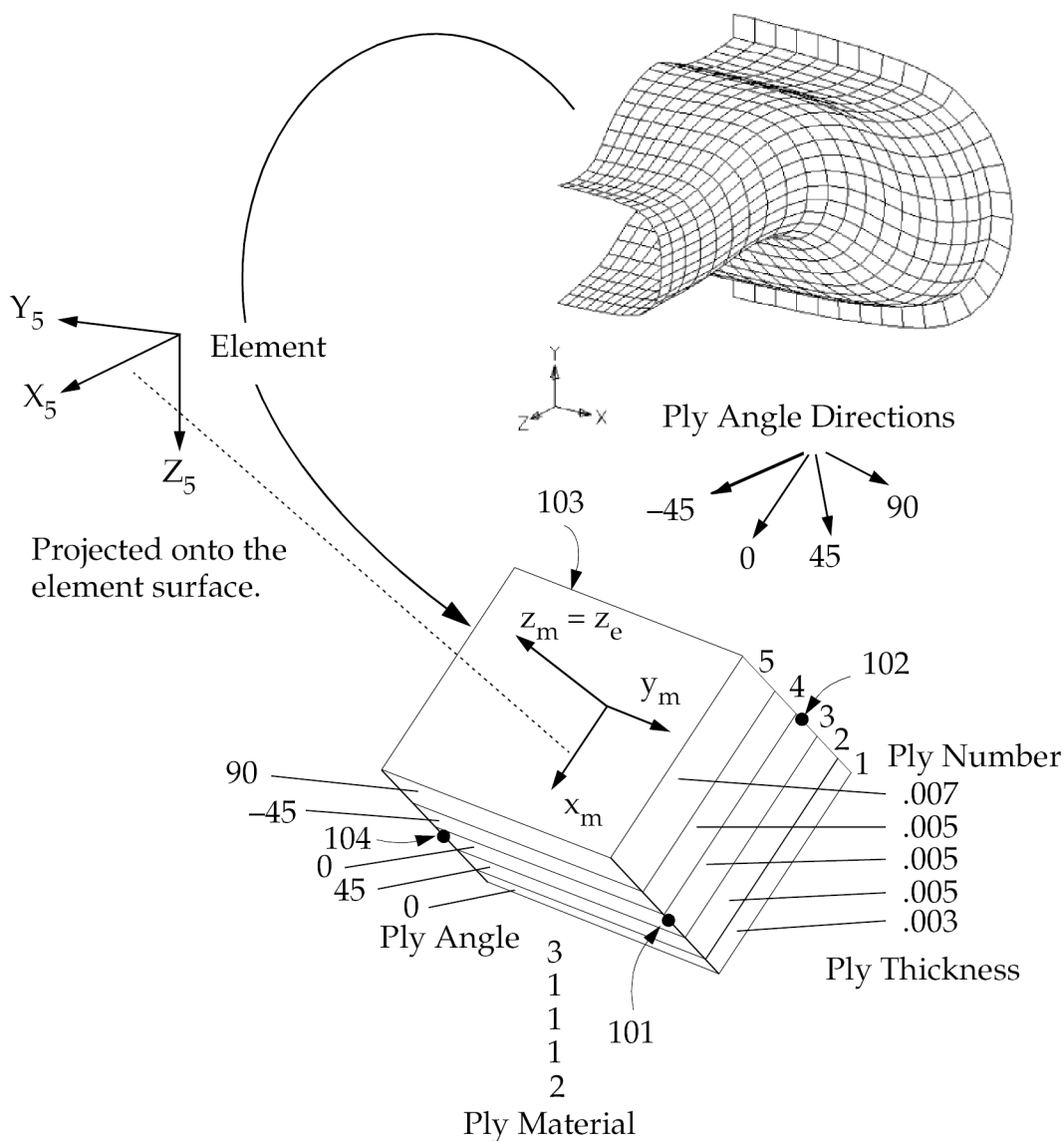
```

Figure 24-5. Composite Output of the Honeycomb Plate

The plate results indicate that the plates are bending downward due to the axial load, which is expected from a nonsymmetric cross section. The laminae stresses show that the face sheets are highly stressed while the core has virtually no stress. The failure index table shows the failure for each laminae and the bonding material to be less than 1, indicating that the structure will not fail. Note, however, that since each lamina is assumed to be in a state of plane stress, the failure index doesn't take into account the crushing of the core.

As a final example showing the use of the PCOMP entry, consider the model shown in [Figure 24-6](#).

Figure 24-6 shows an example of the data entries required to define a five layer composite material. A typical element (with grid points 101, 102, 103, and 104) in the model is magnified to illustrate the individual layers.



```

CQUAD4, 101, 1, 101, 102, 103, 104, 5
PCOMP, 1, , , 10000.0, STRN,
+, 2, .003, 0.0, YES, 1, .005, 45.0, YES,
+, 1, .005, 0.0, YES, 1, .005, -45.0, YES,
+, 3, .007, 90.0, YES
MAT8, 1, 2.0E+7, 2.0E+6, 0.35, 1.0E+6, 1.0E+6, 1.0E+6, 0.0
+, 0.0, 0.0, 0.0, .007, .006, .007, .006, 15000.,
+, , , 1.0
MAT8, 2, 1.0E+7, 1.0E+7, 0.05, 1.0E+6, 1.0E+6, 1.5E+6, 0.0
+, 0.0, 0.0, 0.0, .007, .006, .007, .006, 15000.,
+, , , 1.0
MAT8, 3, 0.8E+7, 0.8E+7, 0.05, 0.7E+6, 0.7E+6, 1.3E+6, 0.0
+, 0.0, 0.0, 0.0, .006, .005, .006, .005, 12000.,
+, , , 1.0
CORD2R, 5, , 0., 0., 0., 0., -1., 0.,
+, 0., 0., 1.

```

Figure 24-6. Typical Composite Layup

In the example above, the direction of the x-axis of the material coordinate system (x_m) is defined by the projection of the x-axis of coordinate system 5 onto the element. Using this technique, the direction of material coordinate system is independent of the shape of the element.

The material direction of all the elements in the model can be defined by referencing a single coordinate system. The MCID field on the CQUAD4 Bulk Data entry defines the ID of this coordinate system. The coordinate system is shown in the figure as the x_5 -, y_5 -, z_5 -axis and is defined on the CORD2R Bulk Data entry.

- The x-axis of the material coordinate system can also be defined with an angle. If the THETA or MCID field on the CQUAD4 Bulk Data entry is an integer, then the value is taken as a coordinate system. If it is a real number, then the value is taken as an angle.
- The z-axis (z_m) of the material coordinate system is in the direction of the z-axis of the element coordinate system (z_e).
- The direction of the y-axis (y_m) follows the right-hand rule.

The CQUAD4 entry references the PCOMP entry, which defines the following layup:

Ply	Material	Thickness	Angle
1	2	0.003	0.0
2	1	0.005	45.0
3	1	0.005	0.0
4	1	0.005	-45.0
5	3	0.007	90.0

The composite material is made of 5 plies. Ply number 1 is at the most negative z_m . Note that since z_m is in the direction of z_e , the element grid point connectivity order determines which is the top and bottom surface of the composite and thus, the stacking direction. Ply angle 0 is in the direction of x_m . The positive sense of the ply angle is in the direction from x_m to y_m . The PCOMP entry references the MAT8 data entries by the MID field.

The example uses three different materials. Note that strain allowables are specified on the MAT8 entry. To let NX Nastran know that these values are strain allowable, the STRN field must be set equal to 1.0.

- Material 1 is typical of a graphite epoxy tape. A tape has continuous fibers that all point in the same direction. Note that E1 is ten times the value of E2.
- Material 2 is also a tape.
- Material 3 is a cloth since E1 equals E2. A cloth made of woven fibers has approximately the same number of fibers going in the 1 direction as in the 2 direction.

Directions 1 and 2 refer to the ply coordinate system. The ply material properties on the MAT8 entry are in the ply coordinate system.

- o The 1-axis of the ply coordinate system is in the direction of the ply angle.
- o The 3-axis is in the direction of z_m (or z_e).
- o The 2-axis follows the right-hand rule.

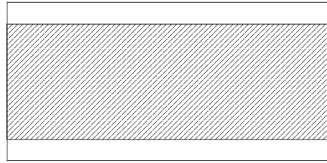
Ply results are output in the ply coordinate system.

Example: Using PSHELL to Model a Honeycomb Section

If you're only interested in the overall behavior of the honeycomb composite section, then you can use an equivalent PSHELL/MAT2 or PSHELL/MAT1 combination to model the PCOMP/MAT8 effect. This representation yields the correct equivalent stiffness and, consequently, the correct displacements. However, the stresses in general are meaningless since the information regarding the individual layers are not available to NX Nastran with the PSHELL/MATi combination. If you want to obtain the stresses, you should use the PCOMP/MAT8 combination.

This example shows how to model the honeycomb section shown in [Figure 24-7](#) using PSHELL/MAT1. The assumptions are that the membrane and bending loads are carried by the facesheets and the transverse shear loads are carried by the honeycomb core. The cross section consists of two 0.01 in thick aluminum facesheets sandwiching a 0.75 in thick honeycomb core. The properties of the section are shown in [Figure 24-7](#).

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



Aluminum Facesheet — $T=0.01''$

Aluminum Honeycomb Core — $T=.75''$

Aluminum Facesheet — $T=0.01''$

Figure 24-7. Honeycomb Cantilever Plate Represented by PSHELL/MAT1

The calculation of the equivalent properties is shown below:

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

$$T = 0.02$$

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

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

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

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

The transverse shear moduli ($G13$ and $G23$) are different in the two transverse directions, which is typical of honeycomb core. Since the MAT1 entry only allows a single shear modulus, an average G_{core} is used.

A more accurate representation of $G13$ and $G23$ is to use the MAT2 entry instead of the MAT1 entry. Listing 24-2 and Listing 24-3 show the input files using the PCOMP/MAT8 and PSHELL/MAT1 entries, respectively. As shown in the corresponding abridged output files in Figure 24-8 and Figure 24-9, the displacements calculated by using the PSHELL/MAT1 combination are comparable with the displacements obtained using the PCOMP/MAT8 combination.

```

$ filename - pcomp2.dat
ID      LINEAR,PCOMP2
SOL      101
TIME     5
CEND
TITLE = HONEYCOMB COMPOSITE CANTLEVER PLATE
SUBTITLE = SYMMETRIC - .01/.75/.01
$
SPC = 1
DISP = ALL
$
SUBCASE 1
LABEL = AXIAL LOAD
LOAD = 1
$
SUBCASE 2
LABEL = VERTICAL SHEAR LOAD
LOAD = 2
$
SUBCASE 3
LABEL = VERTICAL TWIST LOAD
LOAD = 3
$
BEGIN BULK
$
PARAM    POST    -1
$
GRDSET
GRID     1          0.0      -10.      0.0
GRID     2          10.      -10.      0.0
GRID     3          20.      -10.      0.0
GRID     4          30.      -10.      0.0
GRID     5          0.0       0.0       0.0
GRID     6          10.       0.0       0.0
GRID     7          20.       0.0       0.0
GRID     8          30.       0.0       0.0
GRID     9          0.0       10.       0.0
GRID    10          10.       10.       0.0
GRID    11          20.       10.       0.0
GRID    12          30.       10.       0.0
$

```

6

Listing 24-2. Honeycomb Sandwich Using PCOMP (Continued)

```

CQUAD4  1      100      1      2      6      5
CQUAD4  2      100      2      3      7      6
CQUAD4  3      100      3      4      8      7
CQUAD4  4      100      5      6     10      9
CQUAD4  5      100      6      7     11     10
CQUAD4  6      100      7      8     12     11
$
FORCE   1      12      0      1.    1000.0  0.0    0.0
FORCE   1      4       0      1.    1000.0  0.0    0.0
FORCE   1      8       0      1.    2000.0  0.0    0.0
$
FORCE   2      12      0      1.     0.     0.     1.
FORCE   2      4       0      1.     0.     0.     1.
FORCE   2      8       0      1.     0.     0.     2.
$
FORCE   3      12      0      1.     0.     0.     1.
FORCE   3      4       0      1.     0.     0.    -1.
$
SPC1    1      123456  1      5      9
$
PCOMP   100
        120      0.01    0.     YES    130      0.75    0.0    YES
        120      0.01    0.     YES
$
MAT8    120      10.+6  10.+6  0.3  3.85+6
MAT8    130      1000.  1000.  100.  1.+5  0.5+5
ENDDATA

```

Listing 24-2. Honeycomb Sandwich Using PCOMP

```

$  pshell2.dat
ID    LINEAR, PSHELL2
SOL    101
TIME    5
CEND
TITLE = EQUIVALENT HONEYCOMB COMPOSITE CANTILEVER PLATE
SUBTITLE = SYMMETRIC - .01/.75/.01
$
SPC = 1
DISP = ALL
$
SUBCASE 1
LABEL = AXIAL LOAD
LOAD = 1
$
SUBCASE 2
LABEL = VERTICAL SHEAR LOAD
LOAD = 2
$
SUBCASE 3
LABEL = TWIST LOAD
LOAD = 3
$

```

Listing 24-3. Honeycomb Sandwich Using PSHELL/MAT1 (Continued)

```

BEGIN BULK
PARAM   POST   -1
$
GRDSET                                     6
GRID    1          0.0    -10.    0.0
GRID    2          10.    -10.    0.0
GRID    3          20.    -10.    0.0
GRID    4          30.    -10.    0.0
GRID    5          0.0     0.0     0.0
GRID    6          10.     0.0     0.0
GRID    7          20.     0.0     0.0
GRID    8          30.     0.0     0.0
GRID    9          0.0     10.     0.0
GRID   10          10.     10.     0.0
GRID   11          20.     10.     0.0
GRID   12          30.     10.     0.0
$
CQUAD4  1          100     1       2       6       5
CQUAD4  2          100     2       3       7       6
CQUAD4  3          100     3       4       8       7
CQUAD4  4          100     5       6      10       9
CQUAD4  5          100     6       7      11      10
CQUAD4  6          100     7       8      12      11
$
FORCE   1          12     0       1.    1000.0  0.0    0.0
FORCE   1           4     0       1.    1000.0  0.0    0.0
FORCE   1           8     0       1.    2000.0  0.0    0.0
FORCE   2          12     0       1.     0.     0.     1.
FORCE   2           4     0       1.     0.     0.     1.
FORCE   2           8     0       1.     0.     0.     2.
FORCE   3          12     0       1.     0.     0.     1.
FORCE   3           4     0       1.     0.     0.    -1.
$
SPC1     1          123456  1       5       9
PSHELL 100 100 .02 100 4332. 200 38.5 +PSLL
+PSLL -.385 .385
MAT1 100 10.+6 0.3
MAT1 200 1000. .75+5
$
ENDDATA

```

Listing 24-3. Honeycomb Sandwich Using PSHELL/MAT1

```

1  HONEYCOMB COMPOSITE CANTLEVER PLATE                                JANUARY 13, 2004 NX NASTRAN 12/10/04 PAGE 10
   SYMMETRIC - .01/.75/.01
0  AXIAL LOAD                                                         SUBCASE 1

                                D I S P L A C E M E N T   V E C T O R

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
   1       G         .0        .0        .0        .0        .0        .0
   2       G      9.938049E-03  3.192859E-03  .0        .0        .0        .0
   3       G      1.951274E-02  3.057678E-03  .0        .0        .0        .0
   4       G      2.948488E-02  2.936741E-03  .0        .0        .0        .0
   5       G         .0        .0        .0        .0        .0        .0
   6       G      9.154656E-03 -1.381969E-17  .0        .0        .0        .0
   7       G      1.958685E-02 -4.734998E-17  .0        .0        .0        .0
   8       G      2.954503E-02 -8.476228E-17  .0        .0        .0        .0
   9       G         .0        .0        .0        .0        .0        .0
  10       G      9.938049E-03 -3.192859E-03  .0        .0        .0        .0
  11       G      1.951274E-02 -3.057678E-03  .0        .0        .0        .0
  12       G      2.948488E-02 -2.936741E-03  .0        .0        .0        .0
0  VERTICAL SHEAR LOAD                                              SUBCASE 2

                                D I S P L A C E M E N T   V E C T O R

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
   1       G         .0        .0        .0        .0        .0        .0
   2       G         .0        .0        8.081975E-03  2.360743E-04 -1.524400E-03  .0
   3       G         .0        .0        2.971653E-02  2.389541E-04 -2.685955E-03  .0
   4       G         .0        .0        5.863158E-02  1.268257E-04 -2.985823E-03  .0
   5       G         .0        .0        .0        .0        .0        .0
   6       G         .0        .0        9.109288E-03 -7.489637E-18 -1.693645E-03  .0
   7       G         .0        .0        3.090988E-02 -1.128154E-17 -2.562822E-03  .0
   8       G         .0        .0        5.927901E-02 -1.922195E-17 -3.002027E-03  .0
   9       G         .0        .0        .0        .0        .0        .0
  10       G         .0        .0        8.081975E-03 -2.360743E-04 -1.524400E-03  .0
  11       G         .0        .0        2.971653E-02 -2.389541E-04 -2.685955E-03  .0
  12       G         .0        .0        5.863158E-02 -1.268257E-04 -2.985823E-03  .0
0  VERTICAL TWIST LOAD                                              SUBCASE 3

                                D I S P L A C E M E N T   V E C T O R

POINT ID.   TYPE      T1      T2      T3      R1      R2      R3
   1       G         .0        .0        .0        .0        .0        .0
   2       G         .0        .0        -1.255258E-03  1.145127E-04  2.226868E-04  .0
   3       G         .0        .0        -3.594521E-03  3.597052E-04  2.259500E-04  .0
   4       G         .0        .0        -5.950768E-03  5.769896E-04  2.259347E-04  .0
   5       G         .0        .0        .0        .0        .0        .0
   6       G         .0        .0        -1.634409E-17  1.417446E-04  2.934695E-18  .0
   7       G         .0        .0        -5.219971E-17  3.523428E-04  4.029600E-18  .0
   8       G         .0        .0        -9.308500E-17  5.806674E-04  4.117684E-18  .0
   9       G         .0        .0        .0        .0        .0        .0
  10       G         .0        .0        1.255258E-03  1.145127E-04 -2.226868E-04  .0
  11       G         .0        .0        3.594521E-03  3.597052E-04 -2.259500E-04  .0
  12       G         .0        .0        5.950768E-03  5.769896E-04 -2.259347E-04  .0

```

Figure 24-8. Displacement Outputs for Honeycomb Sandwich Using PCOMP

1	EQUIVALENT HONEYCOMB COMPOSITE CANTILEVER PLATE				JANUARY 13, 2004 NX NASTRAN 12/10/04 PAGE 9			
	SYMMETRIC - .01/.75/.01							
0	AXIAL LOAD				SUBCASE 1			
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	9.977354E-03	3.215835E-03	.0	.0	.0	.0	
3	G	1.958953E-02	3.081032E-03	.0	.0	.0	.0	
4	G	2.960244E-02	2.958689E-03	.0	.0	.0	.0	
5	G	.0	.0	.0	.0	.0	.0	
6	G	9.187398E-03	1.784324E-18	.0	.0	.0	.0	
7	G	1.966428E-02	9.149946E-18	.0	.0	.0	.0	
8	G	2.966328E-02	2.280977E-17	.0	.0	.0	.0	
9	G	.0	.0	.0	.0	.0	.0	
10	G	9.977354E-03	-3.215835E-03	.0	.0	.0	.0	
11	G	1.958953E-02	-3.081032E-03	.0	.0	.0	.0	
12	G	2.960244E-02	-2.958689E-03	.0	.0	.0	.0	
0	VERTICAL SHEAR LOAD				SUBCASE 2			
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	8.084896E-03	2.362314E-04	-1.527135E-03	.0	
3	G	.0	.0	2.977032E-02	2.404278E-04	-2.690023E-03	.0	
4	G	.0	.0	5.873456E-02	1.270834E-04	-2.989493E-03	.0	
5	G	.0	.0	.0	.0	.0	.0	
6	G	.0	.0	9.143887E-03	-1.200619E-18	-1.694704E-03	.0	
7	G	.0	.0	3.096434E-02	-3.004362E-18	-2.565396E-03	.0	
8	G	.0	.0	5.937611E-02	-4.946220E-18	-3.006373E-03	.0	
9	G	.0	.0	.0	.0	.0	.0	
10	G	.0	.0	8.084896E-03	-2.362314E-04	-1.527135E-03	.0	
11	G	.0	.0	2.977032E-02	-2.404278E-04	-2.690023E-03	.0	
12	G	.0	.0	5.873456E-02	-1.270834E-04	-2.989493E-03	.0	
0	TWIST LOAD				SUBCASE 3			
D I S P L A C E M E N T V E C T O R								
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
1	G	.0	.0	.0	.0	.0	.0	
2	G	.0	.0	-1.277012E-03	1.157813E-04	2.219504E-04	.0	
3	G	.0	.0	-3.622530E-03	3.624341E-04	2.244598E-04	.0	
4	G	.0	.0	-5.978552E-03	5.825486E-04	2.239124E-04	.0	
5	G	.0	.0	.0	.0	.0	.0	
6	G	.0	.0	-3.011317E-18	1.428981E-04	5.786049E-19	.0	
7	G	.0	.0	-1.073759E-17	3.549739E-04	9.055212E-19	.0	
8	G	.0	.0	-2.047949E-17	5.861556E-04	1.015648E-18	.0	
9	G	.0	.0	.0	.0	.0	.0	
10	G	.0	.0	1.277012E-03	1.157813E-04	-2.219504E-04	.0	
11	G	.0	.0	3.622530E-03	3.624341E-04	-2.244598E-04	.0	
12	G	.0	.0	5.978552E-03	5.825486E-04	-2.239124E-04	.0	

Figure 24-9. Displacement Outputs for Honeycomb Sandwich Using PSHELL/MAT1

Chapter 25: Fluid-Structure Interaction

- *Overview of Fluid-Structure Interaction*
- *Coupled Acoustic-Structural Analysis*
- *Axisymmetric Acoustic Cavity Modeling*

25.1 Overview of Fluid-Structure Interaction

NX Nastran provides you with several different methods to analyze the fluid-structure interaction.

- NX Nastran contains an axisymmetric hydroelastic modeling capability that lets you solve a variety of fluid problems with small motion, compressibility, and gravity effects.
- NX Nastran allows you to compute virtual fluid mass.
- NX Nastran lets you perform a fully coupled acoustic-structural analysis.
- NX Nastran lets you perform axisymmetric acoustic cavity modeling.

Hydroelastic modeling and analyzing virtual fluid mass are described in detail in the NX Nastran Advanced Dynamic Analysis User's Guide. Coupled acoustic-structural analysis and axisymmetric acoustic cavity modeling are described in the following sections.

See also

- “[Axisymmetric Fluids in Tanks](#)” in the *NX Nastran Advanced Dynamic Analysis User's Guide*
- “[Virtual Fluid Mass](#)” in the *NX Nastran Advanced Dynamic Analysis User's Guide*

25.2 Performing a Coupled Acoustic-Structural Analysis

NX Nastran allows you to perform a fully coupled acoustic-structural analysis. Its principal application is in the area of acoustics and noise control. Applications include automobile or aircraft interiors, musical instruments, loudspeaker boxes, and solid rocket motors. Any type of problem that involves small, linear motions of the fluid and structure may be a candidate.

You model the fluid with the three-dimensional elements CHEXA, CPENTA, CPYRAM or CTETRA. These elements can assume the properties of irrotational and compressible fluids suitable for acoustic analysis or other types of analyses governed by the three dimensional wave equation. In addition, the following are available.

- 3-D frequency dependent absorbers, in which the absorptive properties are included on the MAT10 bulk entry (fluid material)
- 2-D/1-D frequency dependent absorbers (CAABSF)
- Acoustic barrier element (CHACBR)
- Spring damper acoustic absorber (CHACAB)

The interface between the fluid and structure can be modeled with coincident or non-coincident unconnected grids. The fluid and structural grids are unconnected. In addition, the fluid and structural meshes can be dissimilar. Typically, the fluid mesh is a very coarse mesh compared to the structural mesh.

Unconstrained fluid boundaries are treated as rigid boundaries (velocity is zero but pressures are nonzero). Pressure release fluid boundaries are specified by constraining fluid pressure degree-of-freedom to zero. The fluid pressure degree-of-freedom is defined as component 1 on the SPC, SPC1, or SPCD entries.

The ACMODL bulk entry defines various coupling options. If you do not want the software to include coupling, specify PARAM,ASCOUP,NO in your input file.

NX Nastran allows importing of SFE AKUSMOD coupling matrix. In order for NX Nastran to automatically detect and read the acoustic coupling matrix from AKUSMOD, you will need to specify the parameter PARAM,SFEF70,YES.

Coupled acoustic-structural analysis is available in the direct dynamic solution sequences 107 through 109 and 200 with ANALYSIS = DFREQ, and the modal dynamic solution sequences 103, 110, 111, 112, and 200 with ANALYSIS = MFREQ and ANALYSIS = MTRAN. It should be noted that in solutions 103, 110, 111, 112 and 200, the normal modes are computed separately for the fluid and structural parts of the model; in other words, the uncoupled modes of the fluid and structure are used in the modal formulation of the stiffness, mass, and damping. In SOLs 110 through 112, the SDAMPING Case Control command and the parameters G and W3 are applied only to the structural portion of the model. Design Sensitivities may be computed in SOL 200.

Defining the Fluid

You define fluid grid points on the GRID bulk entry by specifying a value of -1 for CD in field 7. Fluid elements are defined with the CHEXA, CPENTA, CPYRAM, and CTETRA bulk entries. On the referenced PSOLID entry, you must specify the character value PFLUID for FCTN in field 8 and MID, field 3, must reference a MAT10 material entry. The MAT10 entry defines the bulk modulus and the mass density properties of the fluid, along with optional absorber properties.

Additionally, you can model different fluid domains and connect them using a [surface-to-surface glue](#) definition.

3-D Frequency Dependent Fluid-Absorber

You can define 3-D frequency dependent absorbers by including absorptive properties on the MAT10 bulk entry associated with the fluid. By specifying a nonzero value in the GAMMA field, the solution becomes complex and frequency-dependent with the imaginary part of the governing matrix equation representing acoustic damping. The TIDBULK, TIDRHO, TIDGE, and TIDGAMMA fields on the MAT10 entry allow you to model frequency dependence of the bulk modulus, density, damping coefficient, and ratio of imaginary to real bulk modulus, respectively. A non-zero GAMMA field also introduces a frequency dependence.

See [3-D Absorber Mathematical Description](#).

1-D / 2-D Frequency Dependent Acoustic Absorber Element

The acoustic absorber element CAABSF is an element connecting 1 to 4 fluid grids at the fluid-structure interface. Its frequency dependent impedance is specified on the PAABSF entry.

If one grid point is given, then the impedance $Z(f) = Z_R + i Z_I$ is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area. $Z_R(f) = TZREID(f) + B$ and $Z_I(f) = TZRIMD(f) - K/(2\pi f)$.

The resistance represents a damper quantity B . The reactance represents a quantity of the type $(\omega M - K/\omega)$. The impedance is defined as $Z = p/\dot{u}$ where p is the pressure and \dot{u} is the velocity. The scale factor S is used in computing element stiffness and damping terms as:

$$K = \frac{A}{S} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int (\text{of shape functions})$$

$$B = \frac{A}{S} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int (\text{of shape functions})$$

The absorption coefficient is calculated as:

$$\alpha = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

CHACAB Sandwich Acoustic Absorber Element

The CHACAB acoustic absorber element is simulated by an assembly consisting of a mass attached to a spring and damper which are in parallel. The properties are defined by the PACABS entries. The element consists of 8 to 16 structural grid points defining two surfaces.

The PACABS entry references three TABLEDi entries which define the resistance, Y1; reactance, Y2; and the confidence level, Y3 of the material. These properties are a function of frequency and are measured in the Standing Wave Tube Test (ASTM Standard). This data is used to calculate the value of the mass, spring, and damper used in the structural model that will emulate the behavior of the absorber material.

The last field on the PACABS entry defines the cutoff frequency which determines the extent of the data used in the least square fit operations. This is needed because finite element analysis is not applicable to acoustic analysis in the high frequency range. In fact, it is recommended that this methodology be restricted to frequencies with wavelengths much larger than the absorber thickness. Nevertheless, the user may override this cutoff frequency so that the experimental results of short wavelengths are used in the calculations.

CHACBR Acoustic Barrier Element

The acoustic barrier element is synthesized with two masses sandwiching a spring. The element and its properties are defined by the CHACBR and PACBAR entries, respectively. The element consists of 8 to 16 structural grid points defining two surfaces.

The PACBAR entry defines the mass-per-unit area of the backing and the septum, and the resonant frequency of the barrier. The resonant frequency of the sandwich is defined as the frequency with the lowest value of transmission loss (ASTM test). This frequency will be used to calculate the value of the spring used in the structural model that will emulate the behavior of the barrier material.

Acoustic Loading

Loads on fluid elements are, in most cases, analogous to enforced displacements on structural elements. The types of loading available for fluid elements are:

1. Constant, frequency-dependent, or time-dependent enforced pressure at the grid points.
2. An acoustic source characterized by a volumetric flow rate and corresponding to a power spectral density function.

An acoustic source is assumed to be a pulsating sphere in infinite space and is defined on the ACSRCE Bulk Data entry. The ACSRCE entry is selected by the DLOAD Case Control command and contains the material properties of the source and references a DAREA and TABLEDi entry. The TABLEDi entry defines the power-versus-frequency curve characterizing the acoustic source. The ACSRCE entry may also define a delay time and phase angle which is useful whenever multiple sources are present.

In addition to the above acoustic loads, you can also apply any typical dynamic loads to the structural portion of the model.

Single-Point Constraints

Single-point constraints of the fluid ($P = 0.0$) may be enforced on the fluid boundary using the SPC entry or the PS field on the GRID entry. The fluid pressure degree-of-freedom is defined as component 1 on the SPC entry.

Acoustic Output Requests

The following output can be requested on the fluid.

- You can use the DISPLACEMENT or PRESSURE case control command to request:
 - o Pressure and peak sound pressure levels at the fluid points.
 - o RMS sound pressure level is requested by the PARAM,ACOUT,RMS Bulk Data entry. This output is available in Modal and Direct Frequency Response analysis only (SOLs 111, 108, and 200).
 - o Sound pressure level in dB and dBA. A peak reference pressure must be specified on the PARAM, PREFDB Bulk Data entry. The dB level is defined as $\text{dB} = 20 \log(P/\text{PREFDB})$ (for air $\text{PREFDB} = 20 \times 10^{-6}$ Pascal).
- In SOLutions 110, 111, 112, and 200, the SVECTOR, SDISPLACEMENT, SVELOCITY, SACCELERATION Case Control commands result in separate output for the fluid and structural portion of the model. The first output is for the structural grid points and the second for the fluid grid points.
- The FORCE case control command can be used to request particle velocity.
- The MODCON case control command can be used to request modal contribution output.
- The PANCON case control command can be used to request panel contribution output.
- The FLSTCNT case control command can be used to specify control parameters and to request debug files.
- The ERP case control command can be used to request equivalent radiated power for shell elements (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, and CTRIAR). and output

equivalent radiated power for panels that contain shell elements. See “ERP Output Description” in the *Basic Dynamic Analysis User's Guide*.

Defining Panels

A panel is a selection of structural grid points located on the structural mesh boundary. The ERP, MODCON, PANCON, and FLSPOUT case control commands each have descriptors to select panels. Panels are defined using the PANEL bulk entry, along with the SET3 bulk entries. The SET3 bulk entry can list identification numbers for structural grid points, elements, or physical properties.

Modal and Panel Contribution Output

PANCON Case Control

Panel contribution output using the PANCON case control command is used to determine which of the panels has the most influence on the response. The PANCON command has the following capabilities.

- Either SORT1 and SORT2 output can be requested.
- The ABS, NORM and BOTH descriptors determines if the output is absolute, normalized or both.
- The TOPP descriptor specifies the number of structural panels to list in the output that have the greatest contribution to the response at each frequency.
- The TOPS descriptor specifies the number of structural grids to list in the output that have the greatest contribution to the response at each frequency.
- The SOLUTION descriptor is used to select which frequencies are used in the panel contribution output
- The PANEL descriptor specifies the set of panels to output panel contributions.
- The SETMC case control command, which works similar to the XYPLOT case control command, is used to specify which individual responses you would like the modal contribution output. For example:

```
SETMC 121 = ACCE/ 99(T3), 1200(T1), 1399(R2)
`SETMC 161 = DISP/ 99(T3), 1200(T1), 1399(R2)
SETMC 222 = STRESS/ 134(22)
```

then using the option “=ALL” on PANCON

```
PANCON (SORT1, REAL, PRINT, ABS, TOPP=3) =ALL
```

would output panel contributions for all of the SETMC IDs 121, 161, and 222. The SETMC case control supports the response types ACCE, DISP, VELO, ELFO, SPCF, STRE, STRA. You are not required to include, for example, the ACCELERATION case control request when using the ACCE response type on the SETMC case control.

MODCON Case Control

Modal contribution output using the MODCON case control command is used to determine which of the modes has the most influence on the response. The MODCON command has the following capabilities.

- Either SORT1 and SORT2 output can be requested.

- The ABS, NORM and BOTH descriptors determines if the output is absolute, normalized or both.
- The TOPS and TOPF descriptors specify the number of structural and fluid modes, respectively, to list in the output that have the greatest contribution to the response at each frequency.
- The PANELMC descriptor can be used to request modal contributions by panels. The panels are defined with the PANEL bulk entry, then listed with the SET case control. PANELMC then selects the SET ID.
- The SOLUTION descriptor is used to select the frequencies or time steps used in the modal contribution output.
- The SETMC case control command, which works similar to the XYPLOT case control command, is used to specify which individual responses you would like the modal contribution output. For example:

```
SETMC 121 = ACCE/ 99(T3), 1200(T1), 1399(R2)
`SETMC 161 = DISP/ 99(T3), 1200(T1), 1399(R2)
SETMC 222 = STRESS/ 134(22)
```

then using the option “=ALL” on MODCON

```
MODCON (SORT1, REAL, PRINT, ABS, TOPS=3) =ALL
```

would output modal contributions for all of the SETMC IDs 121, 161, and 222. The SETMC case control supports the response types ACCE, DISP, VELO, ELFO, SPCF, STRE, STRA. You are not required to include, for example, the ACCELERATION case control request when using the ACCE response type on the SETMC case control.

See “MODCON Output Description” in the *Basic Dynamic Analysis User's Guide*.

Superelement Analysis for Acoustic-Structure Analysis

Superelements may be used with the following restrictions:

1. A superelement may contain either fluid or structural points, but not both. (The residual structure may contain both.)
2. The grid points at the fluid-structure interface may be assigned to the residual structure only.

This requires the specification of q-set points using the SEQSETi and EIGR or EIGRL Bulk Data entries and the METHOD Case Control command.

Normal Modes Analysis for Acoustic-Structure Models

Since the fluid portion of the model may resonate within a different frequency range than the structural portion, it is possible to specify a different eigenvalue extraction method for each in the residual structure only. In modal analysis (SOLs 103, 110 through 112 and 200), this is achieved by the specification of a keyword on the METHOD Case Control command. The METHOD(FLUID) command references the EIGR or EIGRL entry to be used for the fluid portion; METHOD(STRUCTURE) references the same entries for the structural portion. Either one or both may be specified in the Case Control Section. The default is STRUCTURE.

Acoustic-Structural Coupling Punch Output

You can request the output of the acoustic-structure coupling matrix AGG to the punch file with the AGGPCH describer on the FLSTCNT case control command.

For example,

```
FLSTCNT AGGPCH=YES
```

will write the AGG matrix to the punch file.

You can then import and optionally scale this AGG matrix in a consecutive solution that includes the same model definition at the coupled interface. Use the A2GG case control command and the ASCOUP describer on the FLSTCNT case control command to request this import.

See the FLSTCNT case control command in the *NX Nastran Quick Reference Guide*.

SFE AKUSMOD Acoustic-Structural Coupling Interface

The acoustic-structural coupling created by SFE AKUSMOD can be imported into a NX Nastran dynamic analysis.

To request the import of the AKUSMOD coupling file, the SFEF70 describer on the FLSTCNT case control command must be set to YES. For example,

```
FLSTCNT SFEF70=YES
```

When SFEF70=YES, NX Nastran does not compute the coupling, and instead uses the coupling definition from the external file. When SFEF70=NO (default), the external coupling is not imported.

NX Nastran expects the AKUSMOD file in the same directory where the job is being run, and expects the file name to be fort.70. An ASSIGN statement which uses UNIT=70 must be defined in the file management section of your input file if the coupling file is not named fort.70 or if it is in a location other than where the job is run. For example,

```
ASSIGN OUTPUT2='/directory_path/user_file_name.70' UNIT=70
```

See the FLSTCNT case control command in the *NX Nastran Quick Reference Guide*.

Acoustic-Structure Coupling

The model for fluid/structural systems has an explicit FE mesh for both the structural components and the interior fluid cavity. The equation of motion for the coupled system is given as

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p}_f \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p}_f \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p_f \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

where M, B, and K are mass, damping, and stiffness matrices respectively, and P is the loading vector. The subscripts s and f represent the partitions of the structure and fluid DOF respectively. The DOF are displacements, u, for the structure and pressure, p, for the fluid. The A matrix is the coupling between the fluid and structure DOF at the wetted interface.

Coupling Algorithm Description

The coupling algorithm uses the following multi-step approach to generate the coupling matrix given by

$$A = \int_S \{N_s\} [N_f] dS$$

where N_s and N_f are structure and fluid shape functions.

- **Free Face Determination**

NX Nastran automatically determines the fluid and structural element free faces.

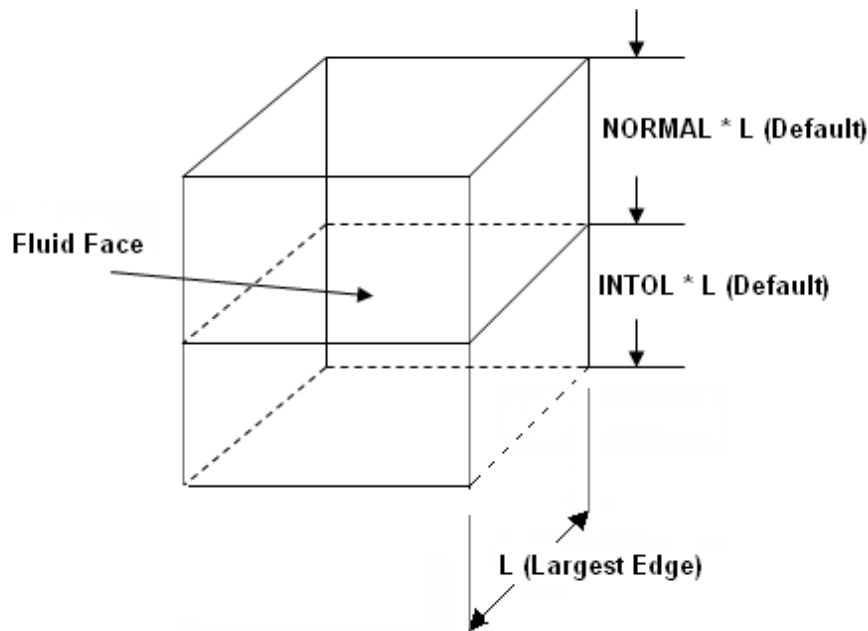
- **Pairing**

Starting with this step, the pairing algorithm works with one fluid face at a time.

A bounding box is created around the fluid face as shown below to find one or more structural faces which it overlaps. The height of the bounding box is controlled with the NORMAL field (outward normal), and the INTOL field (inward normal), on the ACMODL bulk data entry.

The SRCHUNIT field on the ACMODL bulk data entry changes the meaning of the NORMAL and INTOL fields:

- o If SRCHUNIT = "REL", NORMAL and INTOL are a ratio of the search box height to the maximum edge length of the fluid free face (Default).
- o If SRCHUNIT = "ABS", NORMAL and INTOL are a search distance in the model/absolute units.



- **Parallel Check**

The structural faces which are found within the bounding box must be more or less parallel with this fluid face. This is checked by ensuring that the normal of the fluid face and structure faces form an angle less than the value of the OVLPANG field on the ACMODL bulk entry (default = 60 degrees).

- **Subdivision**

The grids from the structural faces from step 3 are projected onto the fluid face. A mesh is generated on the fluid face from the set of fluid grids and the projected structural grids such that the fluid and structural faces share a matching discretization. This "virtual" discretization is only used for evaluating the coupling matrix and is not included in the results output.

- **Trimming**

Subdivided structural face regions which are outside the fluid face are trimmed and eliminated from coupling with the current fluid face. However, these eliminated structural regions will be considered for coupling when the above steps are repeated with other adjacent fluid faces.

- **Integration**

The subdivided regions on the fluid face are associated to subdivided regions on the structural faces for integration of matrix A. If multiple overlapping regions are paired with a single fluid region, the structural region which is the closest to the fluid region is coupled to the fluid face. Since the subdivided fluid and structural regions use a matching discretization, the resulting surface integral will be exact. That is, the overlap area between a fluid face and the given overlapping structural face will be exact and independent of structural/fluid mesh discretization.

Glue Conditions in Acoustic Analysis

Surface-to-surface gluing can be used between acoustic mesh faces. The acoustic mesh, which uses the solid elements CHEXA, CPENTA, CPYRAM and CTETRA can consist of several independent meshes with adjacent acoustic faces glued. Similar to structural glue conditions, the meshes between the acoustic glued faces can be dissimilar. The acoustic-to-acoustic glue inputs are consistent with the structural-to-structural capability, except on the BGPARM bulk entry, GLUETYPE, PENTYPE, and PENT are all ignored. PENN is used to calculate the acoustics penalty matrix K.

See the "Gluing Elements" chapter for more information.

Requesting Coupling Debug Files

You can include the SKINOUT descriptor on the FLSTCNT case control command to request debug files. Depending on the option used, an NX Nastran punch file *.pch and/or debug data deck *.dat is created. The resulting files will have a "_acdbg" appended to their name. For example, if the original input file is "test.dat", the debug files will be "test_acdbg.dat" and "test_acdbg.pch".

The SKINOUT input options are:

SKINOUT=PUNCH: Both a punch file and a debug data deck are created. The punch file (*.pch) contains a list of the original structural and fluid element IDs which participated in the coupling. The debug data deck (*.dat) contains:

- Dummy shell elements representing the *coupled* structural free faces, and are assigned to a dummy pshell with ID=1.
- Dummy shell elements representing the *coupled* fluid free faces, and are assigned to a dummy pshell with ID=2.

SKINOUT=STOP: Works like SKINOUT=PUNCH, except the solution stops immediately after the debug files are created. It is recommended to use SKINOUT,STOP in the first run, then review the coupling interface in a pre/post system before continuing with the remainder of the solution.

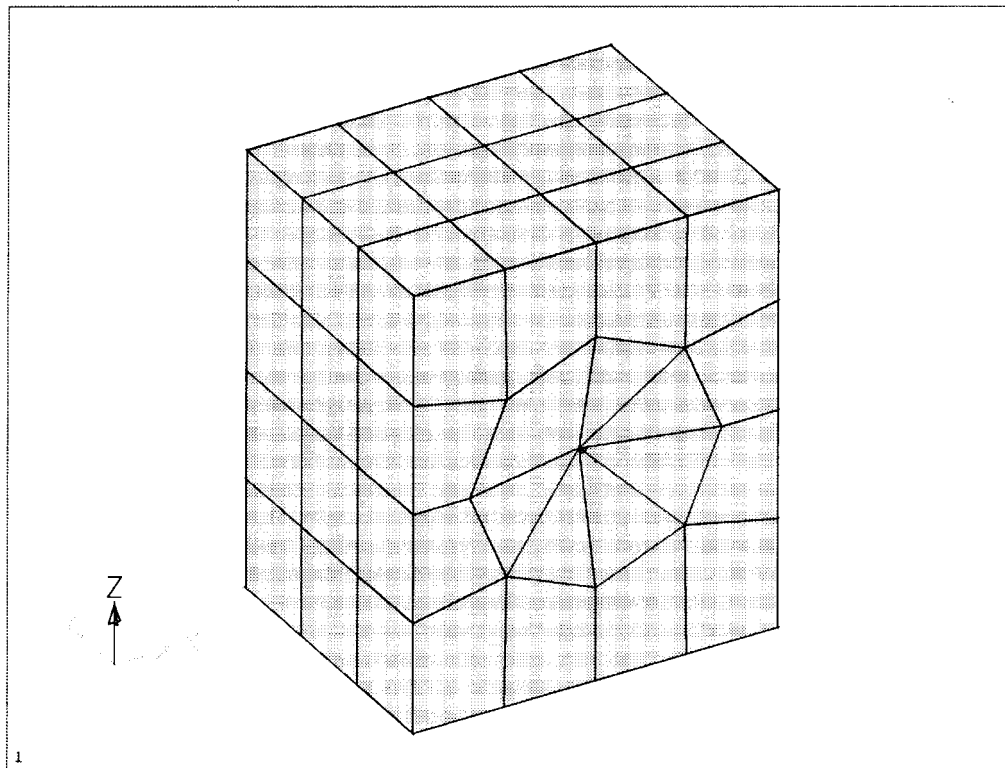
SKINOUT=FREEFACE: A debug data deck is created (no punch file) containing:

- Dummy shell elements representing the *coupled* structural free faces, and are assigned to a dummy pshell property with ID=1.
- Dummy shell elements representing the *coupled* fluid free faces, and are assigned to a dummy pshell property with ID=2.
- Dummy shell elements representing the *uncoupled* structural free faces, and are assigned to a dummy pshell property with ID=3.
- Dummy shell elements representing the *uncoupled* fluid free faces, and are assigned to a dummy pshell property with ID=4.

By default, SKINOUT=NONE and no debug files are created.

After importing a resulting debug *.dat file into any pre/post application which supports bulk data file format (which includes all Siemens PLM Software pre/post applications, NX, I-deas, or Femap), you can use the dummy pshell ID's as a selection tool. For example, superimposing coupled and uncoupled element faces as different colors can be very useful in troubleshooting an unexpected coupling condition.

Acoustic Example



A simple test problem illustrates the fundamentals of acoustic analysis for enclosed containers with flexible walls. The physical problem represents an initial attempt at design and analysis of an enclosure for an acoustic suspension loudspeaker system. The objective was to calculate the resonant frequencies and responses of the system without the mass and impedance of the speaker coil and magnet components.

The structure consists of a simple rectangular box with wood walls. A single cutout is provided for the speaker and a thin polyethylene cone was modeled with triangular shells. The physical properties of the model are listed below. Note the use of the MKS system of units, which illustrates the flexibility of NX Nastran.

Width:	0.5 M	Depth:	0.4 M
Height:	0.6 M	Hole Diam.	0.345 M
Box E Modulus:	11.61E9	Box Density:	562 Kg/M**3
Wall thickness:	0.015 M		
Air Wave speed:	344 M/Sec	Air Density:	1.11KG/M**3
Cone E Modulus:	3.4E9	Cone Density:	450.0
Cone thickness:	0.1E-3 M	Cone Depth:	0.04 M

Executive and Case Control

The Executive and Case Control inputs are used for selecting the type of solution, loads, boundaries, and outputs. Several different runs using the same model were made to check results and investigate the effects of structural coupling. The configuration shown below was a Direct Complex Modal solution using the Lanczos method to extract eigenvalues. An SPC request was used to constrain the structure and an output set was defined to limit the printed displacements. All other data was normal, recommended control and label input.

Bulk Data

The key data of note in the Bulk Data section are as follows:

1. The ACMODL input uses the minimum form which connects fluid and structural points at identical locations.
2. Several eigenvalue methods were used. It is recommended that complex roots always be checked by another method.
3. The speaker cone consisted of a light, thin plastic material. The virtual mass of the exterior air was estimated and added as nonstructural mass on the PSHELL input.

Results from NX Nastran

Three runs (out of many) are described below. They illustrate a recommended sequence for the analysis process.

Run 1: Real Eigenvalue Analysis

Before the coupled structure is analyzed, it is important to understand the behavior of the structural and fluid models separately. Fortunately they can be included in the same data file and the Real Eigenvalue Solution Sequence (SOL 103) may be used. The results for the uncoupled speaker box and the acoustic modes are shown below. Note that the natural frequencies for both systems occur in the same range, which indicates that the enclosure modes will interact with the acoustic cavity resonances. Results for the acoustics were checked by one-dimensional wave solutions. Results for the box were checked against calculated natural frequencies of simply-supported plates.

Mode No.	Frequency, Hz	Type
1	0.0	Air- Constant Pressure (fictitious)
2	292	Air- 1st z (up/down)

Mode No.	Frequency, Hz	Type
3	297	Box- u(y) Back panel
4	351	Air- 1st x (Left/Right)
5	359	Box- u(x) Sides in-Phase
6	379	Box- u(x) Sides out-of-phase
7	450	Box- u(z) Top/bottom
8	453	Air- 1st y (Fore/Aft)

-etc.-

Run 2: Complex Eigenvalue Analysis, Coupled Modes.

The coupled natural frequencies must be obtained from an unsymmetrical matrix equation which requires a Complex Eigenvalue method even in the undamped case. The CLAN method is recommended for most large-order non-superelement jobs. The only changes from the Real Modes job was the addition of an ACMODL input and the changes in eigenvalue method.

Mode No.	Frequency	Type of Dominant Motion
1	0.0	Air- Constant Pressure
2	291	Air- u(z)
3	295	Coupled- u(y)- back panel
4	336	Coupled- u(x)- sides
5	371	Coupled- u(y)- cone
6	373	Coupled- u(x)- sides
7	445	Coupled- u(z)- top/bottom
8	453	Coupled- u(y)- cone

Results above are interpreted by examining the frequency shifts from the uncoupled system. The modes with u(y) fore-aft motion are most likely to be excited by the speaker. Modes 1, 2, 3, 4, 6, and 7 are close to their uncoupled equivalent. Modes 5 and 8 are new combinations of higher modes. The results were confirmed by several methods:

1. The job was rerun using the INV method on the EIGC Bulk Data and produced nearly identical results.
2. A printout of the interface area matrix was obtained by using a DMAP alter and verified by hand calculations.
3. The natural symmetry of the geometry produced symmetric and antisymmetric results relative to the natural structural planes of symmetry (except for roots with close frequencies).

Run 3: Complex Eigenvalue Analysis, Modal Formulation

A more efficient method for solving a coupled matrix problem is to reduce the size of the matrices by using modal coordinates. In this case, both the structural displacements and fluid pressures are replaced in the matrix solution by generalized coordinates representing the uncoupled real modes. For the modal cases below the real modes (25 total) below 800 Hz. were used. In the coupled cases 13 modes were obtained below 500 Hz. All results compared to within 1%.

On a Sun SPARCstation 1+ the CPU time comparisons are:

Type Solution	Solution Size	CPU Time (Seconds)
Uncoupled Modes	538	110.2
Direct, No Modes	538	586.3
Struct. Modes Only	114	143.2
All Modes	25	106.9

The conclusion is that the modal method (SOL 110) will reduce the costs with very little effect on accuracy. The user is cautioned to use a liberal number of real modes (twice the number of coupled modes) to represent the system.

```

ACOUSTIC TEST PROBLEM DATA FILES
ID SPEAK39F,DNH
SOL 107
DIAG 8,12 $ PRINTS MATRIX TRAILERS AND ROOT-TRACKING MESSAGES.
CEND
TITLE=SPEAKER BOX -WITH CONE, SIMPLE CORNER SUPPORTS
SUBTITLE = COUPLED BOUNDARY, NON-MATCHING ELEMENTS
ECHO= UNSORT
SEALL=ALL
SPC=20
$ USES MKS SYSTEM
CMETHOD = 7 $LANCZOS
SET 20= 3,13,23,43,82,83,84,91,93,95,103,113,123,
        131,135,153,163,171,173,175,183,
        1013,1023,1043,1082,1083,1084,1091,1095,1113,
        1131,1135,1163,1173,1183,1193
DISP= 20 $ FOR MINIMUM PRINTOUT
$ DISP(PLOT)=ALL
$
BEGIN BULK
$
$ DEFINE FLUID/STRUCTURE INTERFACE: COINCIDENT POINTS
ACMODL,IDENT
$
PARAM,POST,0
PARAM,COUPMASS,1
$
$ BOX PROPERTIES - WOOD
MAT1,11,11.61+9,,0.3,562.0
PSHELL,1000,11,.015,11,,11
$
$ SPEAKER CONE
MAT1,3,3.4+9,,0.3,450.0
PSHELL,10,3,0.1-3,3,,0.223
$
$ PROPERTIES OF AIR
MAT10,100,131.94+3,1.115
PSOLID,100,100,,2,,1,PFLUID
$
$ EIGEN METHODS
EIGR,20,MGIV,1.0,600.0
EIGC,7, CLAN,,,,,,+CLAN
+CLAN,0.0,10.0,0.0,1600.0,100.0,,20
$ USE INVERSE POWER TO CHECK LOW ROOTS
EIGC,107,INV,MAX,,,,,+EC1
+EC1,0.0,100.0,0.0,1800.0,100.0,12,9
$
$ FIX BOX AT BOTTOM CORNERS
SPC1,20,123,1001,1005,1031,1035
$ COORDINATE SYSTEM AT CENTER OF HOLE
CORD1C,83,283,113,85
$
$

```

```

$FLUID GRID POINTS
$ NOTE VALUE OF -1 IN FIELD 7 INDICATES 1 DOF.
GRID,1,, -.25, -.2, -.3,-1
GRID,2,, -.125, -.2, -.3,-1
GRID,3,, 0.0, -.2, -.3,-1
GRID,4,, .125, -.2, -.3,-1
GRID,5,, .25, -.2, -.3,-1
GRID,11,, -.25, -.0667, -.3,-1
GRID,12,, -.125, -.0667, -.3,-1
GRID,13,, 0.0, -.0667, -.3,-1
GRID,14,, .125, -.0667, -.3,-1
GRID,15,, .25, -.0667, -.3,-1
GRID,21,, -.25, 0.0667, -.3,-1
GRID,22,, -.125, 0.0667, -.3,-1
GRID,23,, 0.0, 0.0667, -.3,-1
GRID,24,, .125, 0.0667, -.3,-1
.25, -.2, 0.,-1
-
ETC
-
$ STRUCTURE GRIDS
GRID,1001,, -.25, -.2, -.3
GRID,1002,, -.125, -.2, -.3
GRID,1003,, 0.0, -.2, -.3
GRID,1004,, .125, -.2, -.3
GRID,1005,, .25, -.2, -.3
GRID,1011,, -.25, -.0667, -.3
GRID,1012,, -.125, -.0667, -.3,,6
GRID,1013,, 0.0, -.0667, -.3,,6
-
ETC.
-
$ OPTIONAL ASET DATA TO USE WITH HESS METHOD
$ASET1,1,1,THRU,195
$ASET1,123,1003,1012,1014,1022,1024,1033
$ASET1,123,1163,1172,1174,1182,1184,1193
$ASET1,123,1042,1043,1044,1081,1082,1083,1084
$ASET1,123,1065,1122,1123,1124
$ASET1,123,1072,1074,1112,1113,1114,1152,1154
$ASET1,123,1051,1061,1131,1141
$ASET1,123,1055,1065,1135,1145
$
$
$SOLID ELEMENTS FOR AIR
CHEXA 1 100 1 2 12 11 41 42 +001
+001 52 51
CHEXA 2 100 2 3 13 12 42 43 +011
+011 53 52
CHEXA 3 100 3 4 14 13 43 44 +021
+021 54 53
CHEXA 4 100 4 5 15 14 44 45 +031
+031 55 54
CHEXA 11 100 11 12 22 21 51 52 +041
+041 62 61
CHEXA 12 100 12 13 23 22 52 53 +051
+051 63 62
CHEXA 13 100 13 14 24 23 53 54 +061
+061 64 63
CHEXA 14 100 14 15 25 24 54 55 +071
+071 65 64
CHEXA 21 100 21 22 32 31 61 62

+081
+081 72 71
-
ETC
-

```

```

$STRUCTURAL ELEMENTS
CQUAD4 1001 1000 1001 1002 1012 1011
CQUAD4 1002 1000 1002 1003 1013 1012
CQUAD4 1003 1000 1003 1004 1014 1013
CQUAD4 1004 1000 1004 1005 1015 1014
CQUAD4 1011 1000 1011 1012 1022 1021
CQUAD4 1012 1000 1012 1013 1023 1022
CQUAD4 1013 1000 1013 1014 1024 1023
CQUAD4 1014 1000 1014 1015 1025 1024
-
ETC
-
$
$SIMPLE CONE
$
CTRIA3, 5042,10,1042,1043,1083
CTRIA3, 5043,10,1043,1044,1083
CTRIA3, 5044,10,1044,1084,1083
CTRIA3, 5084,10,1084,1124,1083
CTRIA3, 5124,10,1124,1123,1083
CTRIA3, 5123,10,1123,1122,1083
CTRIA3, 5122,10,1122,1082,1083
CTRIA3, 5082,10,1082,1042,1083
ENDDATA

```

NX Nastran Acoustic Equations

In NX Nastran, the acoustic equations are based on small motion theory with negligible convective momentum terms and locally linear pressure-density relationship. Thus Euler's equation is

$$\ddot{\vec{u}}_f = -\frac{1}{\rho_f} \vec{\nabla} p$$

Equation 25-1.

and the Continuity equation is

$$p = -\beta \vec{\nabla} \cdot \vec{u}_f$$

Equation 25-2.

with the compressibility β defined as

$$\beta = c^2 \rho_f$$

Equation 25-3.

Take the second time derivative of continuity to get

$$\vec{\nabla} \cdot \ddot{\vec{u}}_f = -\frac{1}{\beta} \ddot{p}$$

Equation 25-4.

and operate on Euler's equation with the del operator to get

$$\vec{\nabla} \cdot \ddot{\vec{u}}_f = -\frac{1}{\rho_f} \vec{\nabla} \cdot \vec{\nabla} p$$

Equation 25-5.

or

$$\frac{1}{\beta} \ddot{p} - \frac{1}{\rho_f} \vec{\nabla} \cdot \vec{\nabla} p = 0$$

Equation 25-6.

integrate this equation over the fluid volume and multiply by a virtual pressure δp to get the integral

$$\int_V \left[\frac{1}{\beta} \ddot{p} - \frac{1}{\rho_f} \vec{\nabla} \cdot \vec{\nabla} p \right] dV \delta p = 0$$

Equation 25-7.

This represents the virtual work expression for acoustic pressure. Integrating the second term by parts using Green's theorem and the identities:

$$\vec{\nabla} \cdot (\alpha \vec{v}) = \vec{\nabla} \alpha \cdot \vec{v} + \alpha \vec{\nabla} \cdot \vec{v}$$

$$\alpha = \delta p$$

$$\vec{v} = \vec{\nabla} p$$

$$\vec{\nabla} \delta p = \delta \vec{\nabla} p$$

$$\delta(\vec{\nabla} p \cdot \vec{\nabla} p) = 2\delta \vec{\nabla} p \cdot \vec{\nabla} p$$

we get

$$\int_V \frac{1}{\beta} \ddot{p} \delta p dV + \delta \int_V \frac{1}{2\rho_f} \vec{\nabla} p \cdot \vec{\nabla} p dV - \int_S \frac{1}{\rho_f} \vec{\nabla} p \cdot d\vec{S} \delta p = 0$$

Equation 25-8.

The trick is to factor out the δ operator from each integral. The first term in the above equation, however, contains \ddot{p} which prevents a simple factoring out of δp . Thus no simple variational statement is directly possible. Because of this, it is standard practice at this point to apply the Galerkin method. This is equivalent to assuming that p but not \ddot{p} is subject to variation. This allows us to simply factor out the δp from the integral containing \ddot{p} . Doing this and also defining \ddot{u}_{no}^f as the acceleration of the fluid in the direction of the outward fluid normal, we obtain for the integral:

$$\Pi = \int_V \left[\frac{1}{\beta} p \ddot{p} + \frac{1}{2\rho_f} \vec{\nabla} p \cdot \vec{\nabla} p \right] dV + \int_S \ddot{u}_{no}^f p dS$$

Equation 25-9.

and the variational statement becomes

$$\delta \Pi = 0$$

Equation 25-10.

Let the fluid pressure p at any point be defined as

$$p = [N_f]\{p\}$$

and

$$\ddot{p} = [N_f]\{\ddot{p}\}$$

where $[N_f]$ is the row matrix of pressure shape functions and $\{p\}$ is a vector of nodal pressures. The expression for Π can be expressed as:

$$\begin{aligned}\Pi = & [p] \int_V \frac{1}{\beta} \{N_f\} [N_f] dV \{\ddot{p}\} \\ & + [p] \int_V \frac{1}{2\rho_f} \{\nabla N_f\} [\nabla N_f] dV \{p\} \\ & + [p] \int_S \{N_f\} \ddot{u}_{n_o}^f dS\end{aligned}$$

Equation 25-11.

If \vec{n}_0 is the fluid outward normal at the fluid-structure and \vec{n} is the structure boundary normal pointing into the fluid, then $\vec{n}_0 = -\vec{n}$ and the surface integral can be written as

$$[p] \int_S \{N_f\} \ddot{u}_{n_0}^f dS = -[p] \int_S \{N_f\} \ddot{u}_n^s dS$$

where \ddot{u}_n^s is the normal acceleration of the structure.

If $[C]$ is the direction cosine matrix for \vec{n} and $[N_s]$ is the matrix of shape functions for the structure, we can define

$$\begin{aligned}\ddot{u}_n^s &= [C][N_s]\{\ddot{u}_s\} \\ &= [N_s]\{\ddot{u}_s\}\end{aligned}$$

Equation 25-12.

where $\{\ddot{u}_s\}$ is the matrix of nodal accelerations at the boundary and N_s is the row matrix of shape functions for the structural normal.

$$-[p] \int_S \{N_f\} \ddot{u}_n^s dS = -[p] \int_S \{N_f\} [N_s] dS \{\ddot{u}_s\}$$

Equation 25-13.

We may then compute $\delta\Pi$ and define the following integrals:

$$[M_f] = \int_V \frac{1}{\beta} \{N_f\} [N_f] dV$$

Equation 25-14.

$$[K_f] = \int_V \frac{1}{\rho_f} \{\nabla N_f\} [\nabla N_f] dV$$

Equation 25-15.

$$[A^T] = \int_S \{N_f\} [N_s] dS$$

Equation 25-16.

and arrive at the relationship

$$[M_f]\{\ddot{p}\} + [K_f]\{p\} - [A^T]\{\ddot{u}_s\} = 0$$

Equation 25-17.

NX Nastran allows for the effect of acoustic source density, which represents a term Q on the right-hand side of the continuity equation. When included into the virtual work expression, this will yield an acoustic load $\{P_f\}$ on the right-hand side of Eq. 25-17. Also, NX Nastran allows for frequency dependent wall impedance $= p/\dot{u}_f$ at the structure-fluid $[B_f]\{\dot{p}\}$. Thus, Eq. 25-17 becomes:

$$[M_f]\{\ddot{p}\} + [B_f]\{\dot{p}\} + [K_f]\{p\} - [A^T]\{\ddot{u}_s\} = \{P_f\}$$

Equation 25-18.

The fluid pressure on the structure boundary causes surface tractions on the structure represented by the standard relationship

$$\{F_s\} = \int_S [N_s]^T \{\Phi\} dS$$

Equation 25-19.

But, the traction Φ is related to the fluid pressure by the relation

$$\{\Phi\} = -p\{C\}$$

Equation 25-20.

or

$$\begin{aligned} \{F_s\} &= - \int_S [N_s]^T \{C\} p dS \\ &= - \int_S \{N_s\} [N_f] dS \{p\} \\ &= -[A]\{p\} \end{aligned}$$

Equation 25-21.

The structure equations can now be written as:

$$[M_s]\{\ddot{u}_s\} + [B_s]\{\dot{u}_s\} + [K_s]\{u_s\} = \{P_s\} - [A]\{p\}$$

Equation 25-22.

The fluid and structure equations can now be combined as

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

Equation 25-23.

This is the fluid structure equation solved in NX Nastran. Note that this equation is nonsymmetric. By default, NX Nastran solves a symmetric version of this equation developed by Everstine as follows:

Let the velocity potential q be defined as

$$p = \dot{q}$$

Equation 25-24.

and note the following:

$$\begin{aligned} \ddot{u}_s &= \frac{d\dot{u}_s}{dt} \\ \ddot{p} &= \frac{d\dot{p}}{dt} \\ \dot{p} &= \frac{dp}{dt} \\ p &= \frac{dq}{dt} \end{aligned}$$

and define G as

$$G = -\int_0^t P_f(\tau) d\tau$$

Equation 25-25.

Substitute \dot{q} for p in the structure equation and for the fluid equation replace the vector terms with their derivative equivalents. Then multiplying the fluid equation by -1 and integrating with respect to time and recombining, we get the symmetric equation

$$\begin{bmatrix} M_s & 0 \\ 0 & -M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{q} \end{Bmatrix} + \begin{bmatrix} B_s & A \\ A^T & -B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{q} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & -K_f \end{bmatrix} \begin{Bmatrix} u_s \\ q \end{Bmatrix} = \begin{Bmatrix} P_s \\ G \end{Bmatrix}$$

Equation 25-26.

The output from the solution is, however, expressed in terms of u_s and p .

In the frequency domain, we assume a harmonic function of the form $q = Q_0 e^{i\omega t}$ as a solution.

Then the load integral becomes $\int G_0 e^{i\omega t} d\tau = G_0 / (i\omega) e^{i\omega t}$. The pressure is recovered from the relationship $p = i\omega q$.

3-D Absorber Mathematical Description

You can define 3-D frequency dependent absorbers by including absorptive properties on the MAT10 bulk entry associated with the fluid. By specifying a nonzero value in the GAMMA field, the solution becomes complex and frequency-dependent with the imaginary part of the governing matrix equation representing acoustic damping. The TIDBULK, TIDRHO, TIDGE, and TIDGAMMA fields on the MAT10 entry allow you to model frequency dependence of the bulk modulus, density, damping coefficient, and ratio of imaginary to real bulk modulus, respectively.

Acoustic absorbers can be characterized by a fluid that has equivalent, frequency-dependent acoustical characteristics. For such a case, the governing equation takes the form:

$$\frac{1}{\rho_e(\omega)} \nabla^2 p + \frac{\omega^2}{\beta_e(\omega)} p = 0$$

Equation 25-27.

where $\rho_e(\omega)$ is the equivalent density as a function of frequency and $\beta_e(\omega)$ is the equivalent bulk modulus as a function of frequency.

Using a Galerkin formulation, Equation 25-27 reduces to:

$$[K_f] - \omega^2 [M_f] = \{P_f\}$$

Equation 25-28.

where

$$[K_f] = \frac{1}{\rho_e(\omega)} \int_V \{ \nabla N_f \} [\nabla N_f] dV$$

Equation 25-29.

and

$$[M_f] = \frac{1}{\beta_e(\omega)} \int_V \{N_f\} [N_f] dV$$

Equation 25-30.

In [Equation 25-29](#) and [Equation 25-30](#), N_f are the element shape functions. The integration is performed over the element volume.

Assuming a harmonic solution of the form $e^{i\omega t}$, the equivalent density and equivalent bulk modulus can be written in complex form as $\rho_e(\omega) = \rho_{er}(\omega) + i\rho_{ei}(\omega)$ and $\beta_e(\omega) = \beta_{er}(\omega) + i\beta_{ei}(\omega)$, respectively. By multiplying the numerator and denominator of the reciprocal of equivalent density by the complex conjugate of equivalent density, the reciprocal of equivalent density can be written:

$$\frac{1}{\rho_e(\omega)} = \frac{1}{\rho_{er}(\omega) + i\rho_{ei}(\omega)} = \frac{1}{\rho_s(\omega)} (1 + iG_e(\omega))$$

Equation 25-31.

where

$$\rho_s(\omega) = \frac{\rho_{er}^2(\omega) + \rho_{ei}^2(\omega)}{\rho_{er}(\omega)}$$

Equation 25-32.

and

$$G_e(\omega) = -\frac{\rho_{ei}(\omega)}{\rho_{er}(\omega)}$$

Equation 25-33.

where $\rho_s(\omega)$ is the frequency dependent fluid density, and is defined with the TIDRHO field on the MAT10 entry. $G_e(\omega)$ is the frequency dependent damping for the fluid, which is analogous to structural damping, and is defined with the TIDGE field on the MAT10 entry.

Substituting [Equation 25-31](#) into [Equation 25-29](#) yields:

$$[K_f] = \frac{(1 + iG_e(\omega))}{\rho_s(\omega)} \int_V \{\nabla N_f\} [\nabla N_f] dV$$

Equation 25-34.

Similarly, the reciprocal of equivalent bulk modulus can be written:

$$\frac{1}{\beta_e(\omega)} = \frac{1}{\beta_{er}(\omega) + i\beta_{ei}(\omega)} = \frac{1}{\beta_s(\omega)}(1 - i\gamma(\omega))$$

Equation 25-35.

where

$$\beta_s(\omega) = \frac{\beta_{er}^2(\omega) + \beta_{ei}^2(\omega)}{\beta_{er}(\omega)}$$

Equation 25-36.

and

$$\gamma(\omega) = \frac{\beta_{ei}(\omega)}{\beta_{er}(\omega)}$$

Equation 25-37.

where $B_s(\omega)$ is the frequency dependent fluid bulk modulus, and is defined with the TIDBULK field on the MAT10 entry. $\gamma(\omega)$ is the frequency dependent ratio of the imaginary bulk modulus to the real bulk modulus, and is defined with the TIDGAMMA field on the MAT10 entry.

Substituting [Equation 25-35](#) into [Equation 25-30](#) and rearranging yields:

$$[M_f] = [M'_f] - i \frac{\gamma(\omega)}{\beta_s(\omega)} \int_V \{N_f\} [N_f] dV$$

Equation 25-38.

where

$$[M'_f] = \frac{1}{\beta_s(\omega)} \int_V \{N_f\} [N_f] dV$$

Equation 25-39.

Substituting [Equation 25-38](#) into [Equation 25-28](#) yields:

$$[K_f] - \omega^2 [M'_f] - i\omega^2 \frac{\gamma(\omega)}{\beta_s(\omega)} \int_V \{N_f\} [N_f] dV = \{P_f\}$$

Equation 25-40.

[Equation 25-40](#) can be rewritten as:

$$[K_f] + i\omega[B'_f] - \omega^2 [M'_f] = \{P_f\}$$

Equation 25-41.

where

$$[B'_f] = \omega \left(\frac{\gamma(\omega)}{(\beta_s(\omega))} \int_V \{N_f\} [N_f] dV \right)$$

Equation 25-42.

By defining $\gamma(\omega) \neq 0$, Equation 25-42 is frequency-dependent even if $\gamma(\omega) = \text{constant}$ and $\beta_s(\omega) = \text{constant}$.

Equation 25-42 and the complex portion of Equation 25-34 represent acoustic damping.

Participation Factor Mathematical Description

Following Sung and Nefske, the participation factors are obtained from Eq. 25-23 as follows: Define the transformations

$$\{u_s\} = [\Phi_s] \{\xi_s\}$$

Equation 25-43.

$$\{p\} = [\Phi_f] \{\xi_f\}$$

Equation 25-44.

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. 25-23 and pre-multiplying by the modal matrices, we get the equation

$$\begin{bmatrix} \Phi_s^T M_s \Phi_s & 0 \\ -\Phi_f^T A^T \Phi_s & \Phi_f^T M_f \Phi_f \end{bmatrix} \begin{Bmatrix} \ddot{\xi}_s \\ \ddot{\xi}_f \end{Bmatrix} + \begin{bmatrix} \Phi_s^T B_s \Phi_s & 0 \\ 0 & \Phi_f^T B_f \Phi_f \end{bmatrix} \begin{Bmatrix} \dot{\xi}_s \\ \dot{\xi}_f \end{Bmatrix} + \begin{bmatrix} \Phi_s^T K_s \Phi_s & \Phi_s^T A \Phi_f \\ 0 & \Phi_f^T K_f \Phi_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = \begin{Bmatrix} \Phi_s^T P_s \\ \Phi_f^T P_f \end{Bmatrix}$$

Equation 25-45.

or

$$\begin{bmatrix} m_s & 0 \\ -a^T & m_f \end{bmatrix} \begin{Bmatrix} \ddot{\xi}_s \\ \ddot{\xi}_f \end{Bmatrix} + \begin{bmatrix} b_s & 0 \\ 0 & b_f \end{bmatrix} \begin{Bmatrix} \dot{\xi}_s \\ \dot{\xi}_f \end{Bmatrix} + \begin{bmatrix} k_s & a \\ 0 & k_f \end{bmatrix} \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = \begin{Bmatrix} Q_s \\ Q_f \end{Bmatrix}$$

Equation 25-46.

Define the following harmonic solution forcing frequency ω .

$$\{\xi\} = \begin{Bmatrix} \xi_s \\ \xi_f \end{Bmatrix} = e^{i\omega t}$$

Equation 25-47.

$$\{\dot{\xi}\} = i\omega e^{i\omega t}$$

Equation 25-48.

$$\{\ddot{\xi}\} = -\omega^2 e^{i\omega t}$$

Equation 25-49.

For the bottom equation of [Eq. 25-46](#), we get:

$$\omega^2 [a]^T \{\xi_s\} + [-\omega^2 [m_f] + i\omega [b_f] + [k_f]] \{\xi_f\} = \{Q_f\}$$

Equation 25-50.

Define $[Z_2]$ as

$$[Z_2] = [-\omega^2 [m_f] + i\omega [b_f] + [k_f]]^{-1}$$

Equation 25-51.

then

$$\{\xi_f\} = -\omega^2 [Z_2] [a]^T \{\xi_s\} + [Z_2] \{Q_f\}$$

Equation 25-52.

The fluid mode participation is defined as

$$[P_f] = [\Phi_f] [\{\xi_f\}]$$

Equation 25-53.

where $[\{\xi_f\}]$ is the diagonalized vector of fluid modal amplitudes per excitation frequency.

The structure mode participation is defined as

$$[P_s] = -\omega^2 [\Phi_f][Z_2][a]^T [\{\xi_s\}]$$

Equation 25-54.

where $[\{\xi_s\}]$ is the diagonalized vector of structural modal amplitudes per excitation frequency.

The fluid load participation is defined as

$$\{P_f\} = [\Phi_f][Z_2]\{Q_f\}$$

Equation 25-55.

The fluid-structure panel participation is defined as

$$[P_p] = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T [A]_{\text{panel}}^T [\Phi_s][\{\xi_s\}]$$

Equation 25-56.

The fluid-structure panel-grid participation is defined as

$$[P_g]_i = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T \left\{ A_b^T \right\} [[\Phi_s]] [\xi_s]$$

Equation 25-57.

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.

25.3 Axisymmetric Acoustic Cavity Modeling

With acoustic cavity modeling, you define a mesh over the longitudinal cross section of an acoustic cavity. You connect finite elements are connected between these points to define the enclosed volume.

You define the points with GRIDF data entries for the axisymmetric central fluid cavity and by GRIDS for the radial slots. The GRIDF points are interconnected by finite elements via the CAXIF2, CAXIF3, and CAXIF4 data entries to define a cross-sectional area of the body of rotation. The CAXIF2 element data entry defines the area of the cross section between the axis and two points off the axis (the GRIDF points may not have a zero radius). The CAXIF3 and CAXIF4 data entries define triangular or quadrilateral cross sections and connect three or four GRIDF points, respectively. The density and/or bulk modulus at each location of the enclosed fluid may also be defined on these entries.

The GRIDS points in the slot region are interconnected by finite elements via the CSLOT3 and CSLOT4 data entries. These define finite elements with triangular and quadrilateral cross-sectional shapes, respectively. The width of the slot and the number of slots may be defined by default values on the AXSLOT data entry. If the width of the slots is a variable, the value is specified on the GRIDS entries at each point. The number of slots, the density, and/or the bulk modulus of the fluid may also be defined individually for each element on the CSLOT3 and CSLOT4 entries.

The AXSLOT Bulk Data entry is used to define the overall parameters for the system. Some of these parameters are called the “default” values and may be selectively changed at particular cross sections of the structure. The values specified on the AXSLOT entry will be used if the corresponding fields on the GRIDS, CAXIFI, or CSLOTi entries are left blank. The fields RHOD and RHO (density) and BD and B (bulk modulus) are properties of the fluid. If the value given for bulk modulus is zero, the fluid is considered incompressible by the program. The fields MD and M (number of slots) and WD (slot width) are properties of the geometry. The fields MD and M define the number of equally spaced slots around the circumference, with the first slot located at $\phi = 0^\circ$. The field N (harmonic number) is selected by the user to analyze a particular set of acoustic modes. The pressure is assumed to have the following distribution:

$$p(r,z,\phi) = (r,z) \cos N\phi$$

If $N = 0$, the breathing and longitudinal modes will result. If $N = 1$, the pressure at $\phi = 180^\circ$ will be the negative of the pressure at $\phi = 0^\circ$. If $N = 2$, the pressures at $\phi = 90^\circ$ and $\phi = 270^\circ$ will be the negative of that at $\phi = 0^\circ$. Values of N larger than $M/2$ have no significance.

The interface between the central cavity and the slots is defined with the SLBDY Bulk Data entries. The data for each entry consist of the density of the fluid at the interface, the number of radial slots around the circumference, and a list of GRIDS points that are listed in the sequence in which they occur as the boundary is traversed. In order to ensure continuity between GRIDF and GRIDS points at the interface, the GRIDF points on the boundary between the cylindrical cavity and the slots are identified on the corresponding GRIDS data entries rather than on GRIDF entries. Thus, the locations of the GRIDF points will be exactly the same as the locations of the corresponding GRIDS points.

Various standard Bulk Data entries may be used for special purposes in acoustic analysis. You can use the SPCi Bulk Data entry to constrain the pressures to zero at specified points, such as at a free boundary. Dynamic load entries, direct input matrices, and scalar elements may be introduced to account for special effects.

Assumptions and Limitations

The accuracy of the acoustic model depends upon the selection of the mesh of finite elements. The assumption for each element is that the pressure field has a linear variation over the cross section and a sinusoidal variation around the axis in the circumferential direction. In areas where the pressure gradient changes are large, such as near a sharp corner, the points in the mesh should be placed closer together so that large changes in flow may be defined accurately by the finite elements.

The shape of the finite elements play an important part in the accuracy of the results. It has been observed that long, narrow elements produce disproportionate errors. Cutting a large square into two rectangles will not improve the results, whereas dividing the square into four smaller squares may decrease the local error by as much as a factor of ten.

The slot portion of the cavity is limited to certain shapes because of basic assumptions in the algorithms. The cross section of the cavity normal to the axis must have a shape that is reasonably

well defined by a central circular cavity having equally spaced, narrow slots. Various shapes are shown in **Figure 25-1** in the order of increasing expected error.

It is recommended that shapes such as the cloverleaf and square cross section be analyzed with a full three-dimensional technique. The assumption of negligible pressure gradient in the circumferential direction within a slot is not valid in these cases.

The harmonic orders of the solutions are also limited by the width of the slots. The harmonic number, N , should be no greater than the number of slots divided by two. The response of the higher harmonics is approximated by the slot width correction terms discussed in Section 17.1 of *The NASTRAN Theoretical Manual*.

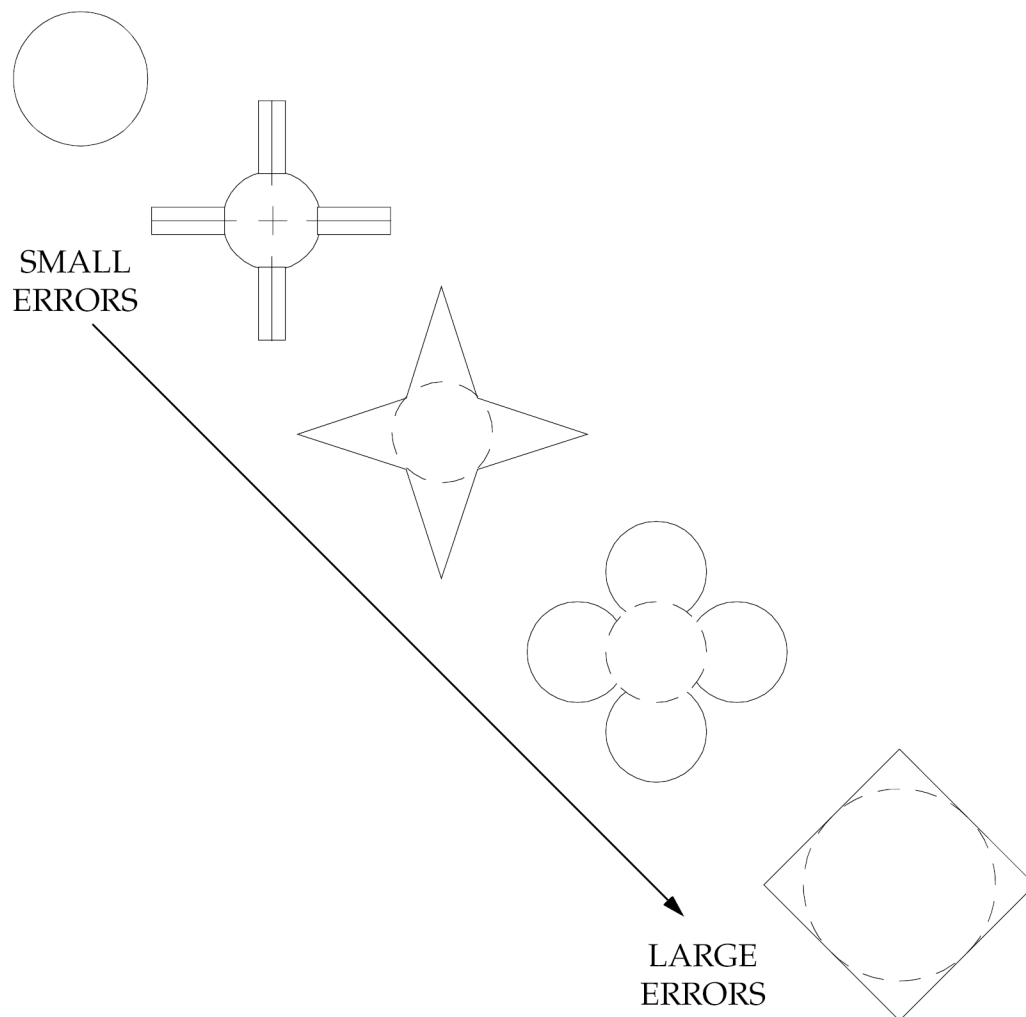


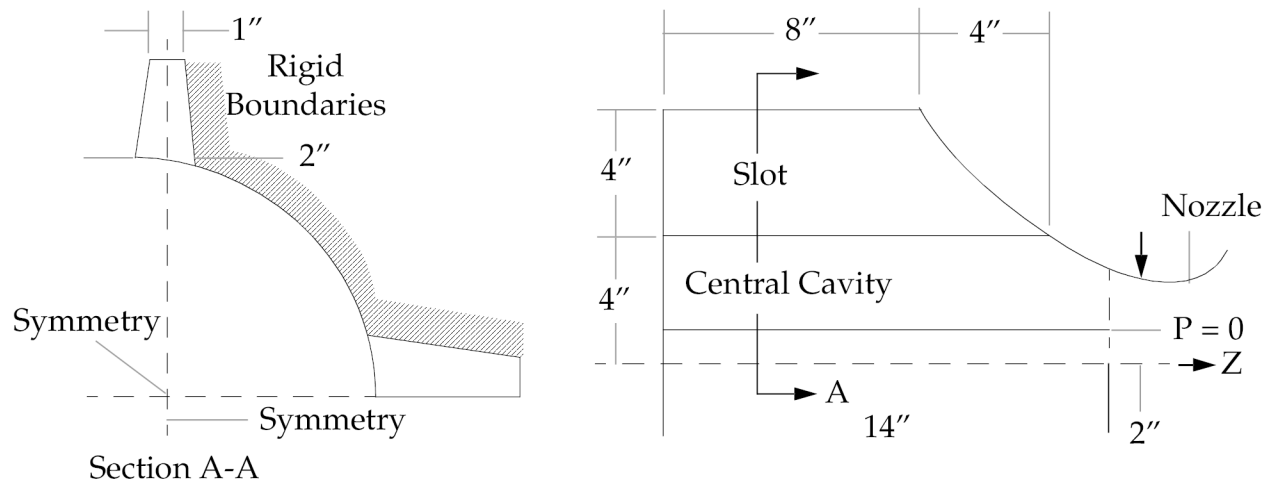
Figure 25-1. Modeling Errors for Various Shapes

The output data for the acoustic analysis consist of the values of pressure in the displacement vector selected via the Case Control command `PRESSURE = n`. The velocity vector components corresponding to each mode may be optionally requested by the Case Control command `STRESS = n` where n is the set number indicating the element numbers to be used for output, or by the words `STRESS = ALL`.

You can request plots of the finite element model and/or of the pressure field with the NX Nastran plot commands. The central cavity cross section will be positioned in the XY plane of the basic coordinate system. The slot elements are offset from the XY plane by the width of the slot in the +Z direction. The radial direction corresponds to X and the axial direction corresponds to the Y direction. Pressures will be plotted in the Z direction for both the slot points and the central cavity points. The PLOTTEL elements are used for plotting the acoustic cavity shape. The plot request entry SET n INCLUDE PLOTTEL must be used where n is a set number.

Acoustic Cavity Example Problem

Table 25-1 contains a listing of the data entries used as a simple example of acoustic cavity analysis. The problem to be solved is illustrated in Figure 25-2. The model was subdivided into only ten finite elements in order to limit the number of data entries. For reasonable engineering accuracy, this model should be subdivided into at least four times that number of elements.



Parameters: Density: $RHOD = 1.2 \times 10^{-7} \text{ lb-sec}^2/\text{in}^4$
 Bulk Modulus: $BD = ra^2 = gRT = 21.0 \text{ lb/in}^2$
 Harmonic: $N = 1$
 Number of Slots: $MD = 4$

Finite Element Model:

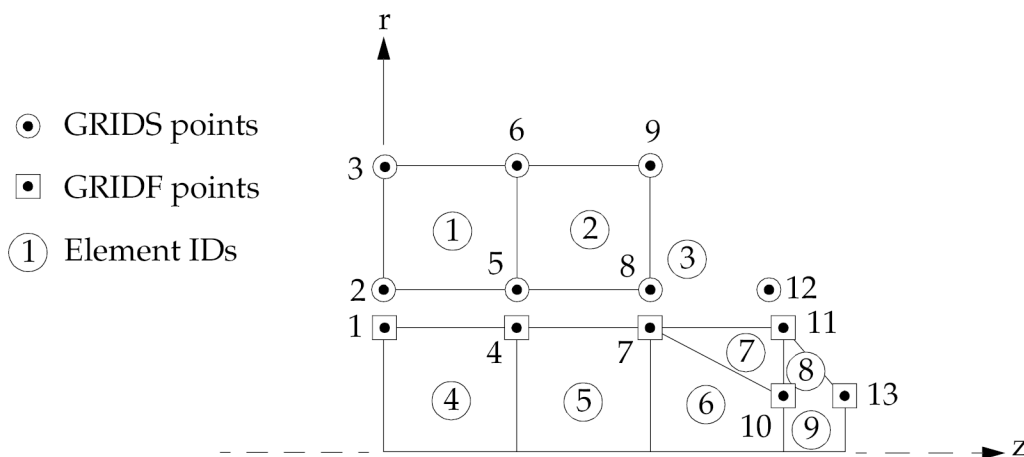


Figure 25-2. Description of Acoustic Cavity Example Problem

Each data entry in **Table 25-1** is given a number on the left side. The format for each type of Bulk Data entry is given in parentheses above the group of that type. The following is a brief description of each part of the input data:

Lines	Description
1-4	Each statement in the Executive Control Section has the format of a request word and a selection separated by blanks or a comma. The CEND statement is last, but the statements above may appear in any order. Solution Sequence 103, normal modes analysis, was chosen. A limit of 2 minutes CPU time was set (TIME2).

Lines	Description
5-7	The TITLE, SUBTITLE, and LABEL commands may contain any list of letters and numbers following the (=) sign. This list will appear on the first three lines of each output page.
8	The method of eigenvalue extraction is selected with the METHOD command. The number 11 refers to the identification number of an EIGRL Bulk Data entry which appears below as line 32.
9-11	A simple output request is illustrated with these commands. PRES = ALL will result in the printout of all pressures at the GRIDF and GRIDS points. STRESS = ALL will result in the printout of all velocities in the elements. This printout will occur for all extracted eigenvectors. Selected points or elements can be printed via the SET command.
12	The BEGIN BULK command denotes the beginning of the Bulk Data Section. The Bulk Data entries may occur in any order. Putting these entries in alphabetic sort will save sorting time in large problems, however.
13	In this problem, all the parameters except slot width WD are constant throughout the volume. The values on AXSLOT will be used whenever a corresponding field in the following entries is blank.
14-20	The location of points on the slot are defined with these entries. Entries 14, 16, 18, and 20 serve a dual purpose by defining a GRIDS point identification number in field 2 and a GRIDF point identification number in field 6. The two types of points thereby are forced to have the same locations at the interface.
21-22	The location of points within the axisymmetric fluid cavity are described by the GRIDF entry. No points are allowed to have a zero or negative radius.
23-31	These entries describe the elements shown in Figure 25-2 . Each element is given a unique identification number and a list of the connected GRIDS or GRIDF points. Since the parameters p and B are constants, these fields are left blank, so the values on the AXSLOT entry will be used.
32	The EIGRL entry is used to define parameters for eigenvalue extraction (resonant frequencies). More than one of these entries may appear. The method to be used is selected with the METHOD command (command 8). With this particular command, the Lanczos method was selected with three (ND = 3) output mode shapes specified.
33	The SLBDY entry defines the boundary between the slot and the central cavity. Both the density (RHO and the number of radial slots (M) are blank, so the AXSLOT defaults are used; i.e., RHOD = 1.2×10^{-7} and M = 4. Only four GRIDS points are on the boundary, so a continuation entry is not necessary. Field 8 is left blank to signify the last entry.
34	The ENDDATA entry is required to denote the end of the Bulk Data Section. Any following entries will be ignored.

Table 25-1. Acoustic Cavity Example Problem Data

Line No.		
1 2 3 4	\$DEMO PROBLEM D03D92 WITH SOL 103 AND LANCZOS SOL 103 TIME 2 CEND	Executive Control statements
5 6 7 8 9 10 11 12	TITLE = ACOUSTIC CAVITY EXAMPLE PROBLEM SUBTITLE = FIRST HARMONIC LABEL = NORMAL MODES (LANCZOS) METHOD = 11 OUTPUT PRES = ALL STRESS = ALL BEGIN BULK	Case Control commands

Table 25-1. Acoustic Cavity Example Problem Data

Line No.	
	(AXSLOT, (RHOD, BD, N, WD, MD) AXSLOT, 1.2-7, 21.0, 1, , 4 (GRIDS, ID, R, Z, W, ID GRIDS, 2, 4.0, 0.0, 0.23 01, 1 GRIDS, 3, 8.0, .0, 1.0 13 GRIDS, 5, 4.0, 4.0, 2.0, 4 14 GRIDS, 6, 8.0, 4.0, 1.0 15 GRIDS, 8, 4.0, 8.0, 2.0, 7 16 GRIDS, 9, 8.0, 8.0, 1.0 17 GRIDS, 12, 4.0, 1.2+1, 2.0, 11 18 (GRIDF, ID, R, Z) 19 GRIDF, 10, 2.0, 12.0 20 GRIDF, 13, 2.0, 1.4E1 21 (CSLOT4, 1D, P ₁ , P ₂ , P ₃ , P ₄ , RHO, , M) 22 CSLOT4, 1 , 2, 3, 6, 5 23 CSLOT4, 2, 5, 6, 9, 8 24 (CSLOT3, 1D, P ₁ , P ₂ , P ₃ , , RHO, ,) 25 CSLOT3, 3, 8, 9, 12 26 (CAXIF2, 1D, P ₁ , P ₂ , , , RHO) 27 CAXIF2, 4, 1 , 4 28 CAXIF2, 5, 4, 7 29 CAXIF2, 6, 7, 10 30 CAXIF2, 9, 10, 13 31 (CAXIF3, ID, P ₁ , P ₂ , P ₃ , , RHO, B) 32 CAXIF3, 7, 7, 10, 11 33 CAXIF3, 8, 10, 11, 13 34 (EIGRL, SID, V1, V2, ND) EIGRL, 11, , , 3 (SLBDY, RHO, M, ID1, ID2, ID4, etc.) SLBDY, , , 12, 8, 5, 2 ENDDATA

Bulk Data entries

Chapter 26: Cyclic Symmetry

- *Understanding Cyclic Symmetry*
- *Understanding Types of Symmetry*
- *Cyclic Symmetry Theory*
- *Using Cyclic Symmetry*
- *Performing Axisymmetric Analysis*

26.1 Understanding Cyclic Symmetry

Many structures, including pressure vessels, rotating machines and antennas for space communications, are made up of virtually identical segments that are symmetrically arranged with respect to an axis.

When you're performing analyses on symmetrical structures, you can exploit that symmetry to reduce greatly reduce the time needed to create and solve an FE model. One advantage is that you will be required to provide an analytical model for only the “fundamental regional” of a structure, and this fundamental region is generally substantially smaller than the analytical model required by more traditional analysis techniques. Analyzing only the fundamental region lets you:

- Reduce the input data describing the finite element model to just that of the fundamental region
- Restrict the computation of g-size stiffness and mass matrices to just the fundamental region
- Exploit the orthogonality of symmetrical component sets, such as the symmetrical and antisymmetrical sets in reflective symmetry, to reduce the cost of matrix operations

For example, if a body consists of N identical segments in the sense that the entire body can be generated through reflections and/or rotations of one of these N segments (the fundamental region) , you only have to define a finite element model for the fundamental region and provide a list of the boundary grid points for this fundamental region. With NX Nastran's cyclic symmetry option (the CYSYM bulk data entry), the software performs all these tasks automatically for each of the three supported subclasses of cyclic symmetry: rotational symmetry (ROT), dihedral symmetry (DIH), and axisymmetry (AXI).

Supported Solution Sequences

In NX Nastran, you can use cyclic symmetry techniques in the following solution sequences:

SOLution 114	Superelement Static Analysis
SOLution 115	Superelement Normal Modes Analysis
SOLution 116	Cyclic Buckling
SOLution 118	Superelement Direct Frequency Response

Comparing Cyclic Symmetry and Traditional Reflective Symmetry Techniques

The following table compares the major steps in the analysis procedures required with traditional reflective techniques and with cyclic symmetry techniques and highlights some of the many advantages of cyclic symmetry.

	Reflective Symmetry	Cyclic Symmetry
Analytical Model	Reduces the size of the analytical model by one-half for each plane of reflective symmetry.	Reduces the size of the model to a “fundamental region,” which is generally substantially smaller than the entire model

	Reflective Symmetry	Cyclic Symmetry
Boundary Conditions	In the general case of p planes of reflective symmetry, you must explicitly define $2p$ sets of boundary conditions with SPCi Bulk Data entries.	You supply a list of the grid points on the boundary of the fundamental region. Boundary conditions are automatically imposed by the program.
Applied Loads	You must decompose the various loading conditions into symmetric and antisymmetric components.	Applied loads are defined to act at arbitrary locations throughout the entire structure respective of whether any or all of these locations are in the explicitly defined fundamental region.
Computer Solution	For each set of defined boundary conditions, a lengthy matrix decomposition and equation solution are required. Static condensations aren't generally economically feasible in static analysis.	A static condensation is performed once to reduce the size of the analysis set. In static analysis, all but the boundary degrees-of-freedom should be omitted. In real eigenvalue analysis, buckling analysis, and dynamic analysis, the boundary degrees-of-freedom and selected internal points are kept in the analysis set. The static condensation involves a lengthy decomposition which should be less lengthy than any one of the decompositions required by traditional techniques because it only involves the fundamental region. Several additional decompositions of relatively trivial magnitude are required after the static condensation.
Data Recovery	You must manually combine the results for each boundary condition with SUBCOM or SYMCOM commands to obtain the complete response of the structure.	NX Nastran automatically provides the response for the entire structure without any special user action. SUBCOM and SYMCOM entries aren't necessary and shouldn't be used.

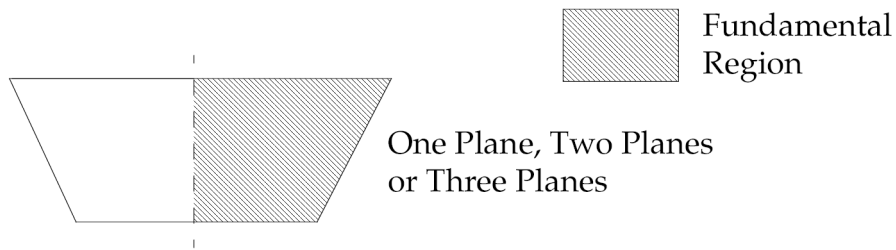
26.2 Understanding Types of Symmetry

There are a number of different types of geometric symmetry, as shown in [Figure 26-1](#).

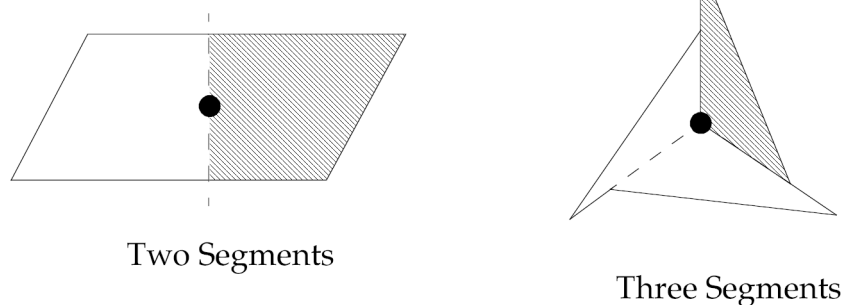
Cyclic symmetry is a subclass of the general class of geometric symmetry. Cyclic symmetry is defined by the rotation of a fundamental region about an axis. Within the category of cyclic symmetry, there are a number of different subtypes. You can use NX Nastran to perform cyclic symmetry analyses on the following types of geometric symmetry:

- Rotational symmetry
- Dihedral symmetry (a combination of reflective plus rotational symmetry)
- Axisymmetry (with and without skew)

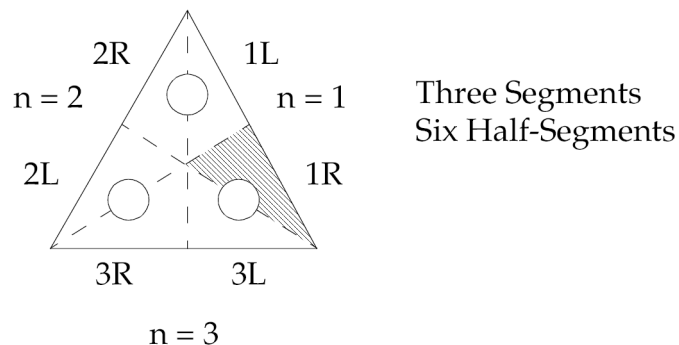
(a) Reflective Symmetry



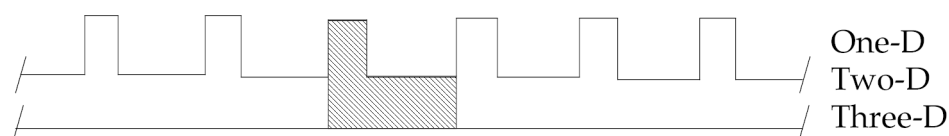
(b) Rotational Symmetry



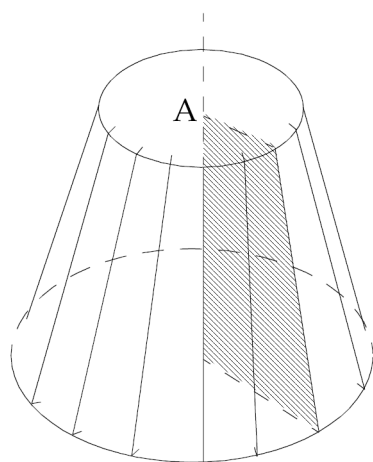
(c) Dihedral Symmetry



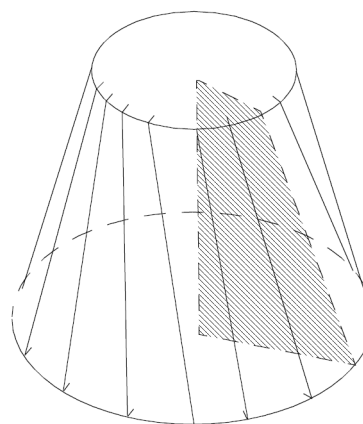
(d) Translational Symmetry



(e) Axisymmetry



Without Skew



With Skew

Figure 26-1. Types of Geometrical Symmetry

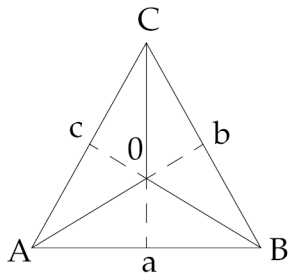
Most of the types of symmetry shown in [Figure 26-1](#) are subclasses of cyclic symmetry. There are inter-relationships between these different subclasses. For example, translational symmetry in one direction, see [Figure 26-1\(d\)](#), can be considered to be a limiting case of rotational symmetry in which the axis of symmetry is infinitely far away. Likewise, axisymmetry without skew is a limiting case of dihedral symmetry as the number of segments is increased without limit, and axisymmetry with skew is a limiting case of rotational symmetry.

The relationship of the different classes of symmetry to the general class called cyclic symmetry is summarized in [Table 26-1](#). With few exceptions, all important types of symmetry are subclasses of cyclic symmetry, and they can, therefore, be analyzed with the cyclic symmetry capability in NX Nastran.

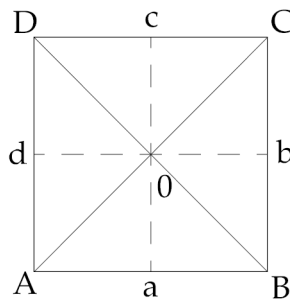
Table 26-1. Types of Symmetry

Type	Subclass of Cyclic Symmetry
Reflective Symmetry	
One Plane	Yes
Two Planes	Yes
Three Planes	No
Rotational Symmetry (Repetition about an axis)	Yes
Dihedral Symmetry (Reflective symmetry plus rotational symmetry)	Yes
Translational Symmetry	
One-D	Yes (special case)
Two-D	No
Three-D	No
Axisymmetry	
With Skew	Yes
Without Skew	Yes

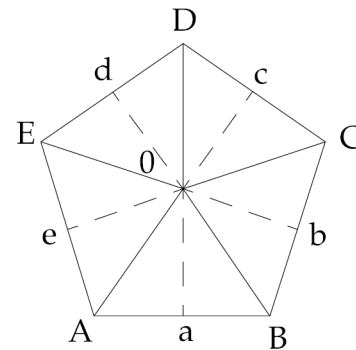
Note that in more general cases, a model may be susceptible to both dihedral and rotational symmetry. For example, consider the three regular N-gons: the equilateral triangle, the square, and the pentagon.



N = 3



N = 4



N = 5

In these figures, the internal solid lines in conjunction with the edges define fundamental regions, e.g., AOB, that may be permuted through rotational symmetry operations. The axis of rotation is normal to the plane of the page and passes through the point O.

In addition, the dotted internal lines such as aO may be used to, for example, reflect A0a into a0B. Also, a0B may be reflected through B0 to obtain B0b, and so on around each N-gon until the identity of A0a is reached. Clearly, the fundamental region A0a for the dihedral symmetry operations is smaller than the fundamental region A0B for rotational symmetry operations. Importantly, both of these fundamental regions are substantially smaller than the fundamental region required by traditional reflective symmetry techniques.

26.3 Cyclic Symmetry Theory

NX Nastran's cyclic symmetry capabilities rely on the development of transformation relations that let you manipulate the mathematical representation of the fundamental region in accordance with the appropriate symmetry operations. The following operations are performed:

1. You create an NX Nastran input file for the fundamental region.
2. You define the mathematical representation of the model in terms of physical quantities, such as $[K]\{u\} = \{P\}$
3. The software transforms the mathematical representation of the model to cyclic quantities, such as $[K]\{u\} = \{P\}$
4. The software solves the problem in terms of cyclic quantities.
5. The results are transformed from the cyclic quantities to physical quantities.

Steps 3 and 5 both depend on the existence of the necessary transformations. Step 4 is also of interest in that it must be demonstrated that problem solution procedures in terms of cyclic quantities are valid and require less computer time than more traditional solution procedures. These matters are addressed in the following sections.

Symmetrical Components

Consider an arrangement of N identical objects symmetrically disposed about a normal. The angle between these objects is

$$a = \frac{2\pi}{N}$$

Equation 26-1.

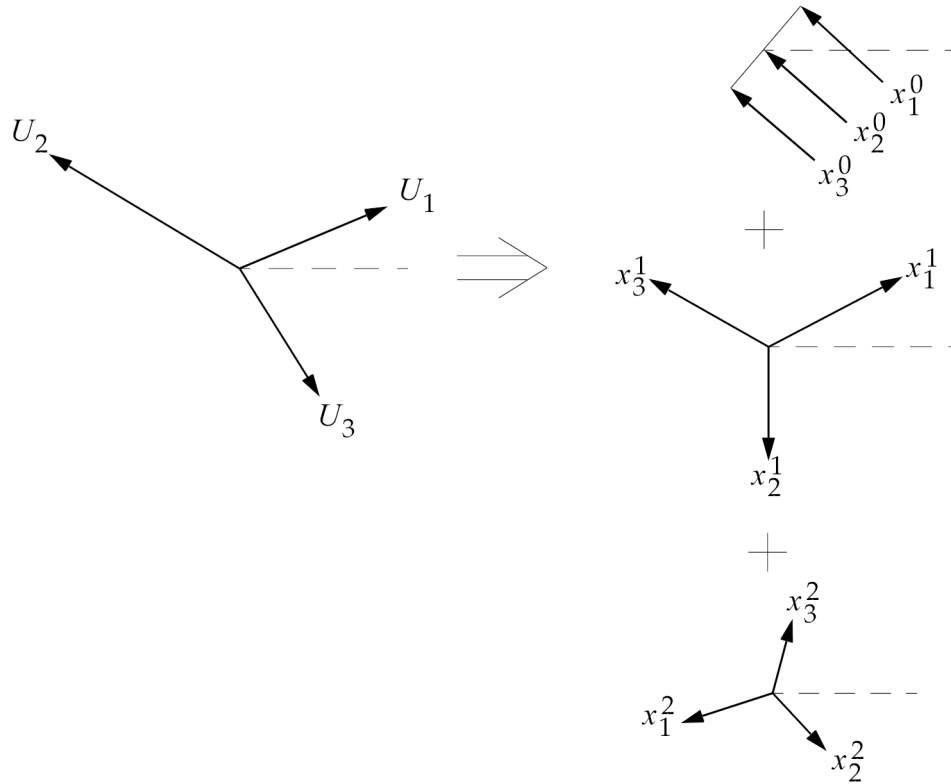
and, in addition, let $U_1(r,t)$ be any variable displacement, force, stress, temperature, etc. in object 1, the fundamental region. The variables r and t denote distance and time respectively. The corresponding quantity in the n -th object is $U_n(r,t)$ and as a single valued solution is required

$$U_n = U_{n+N} \quad n = 1, 2, \dots, N$$

Equation 26-2.

The various U_n 's are, in general, unsymmetrical quantities that are not related in a simple manner.

Fortescue's symmetrical component theory states that a general vector field on N -vectors with arbitrary phase relationships may be resolved into N -sets of N -symmetrically phased vectors of equal magnitude. For $N = 3$, the vectors may be illustrated by the following diagram.



Now, in accordance with the theory of symmetrical components, the following vector sums may be stated.

$$\begin{aligned} U_1 &= x_1^0 + x_1^1 + x_1^2 \\ U_2 &= x_2^0 + x_2^1 + x_2^2 \\ U_3 &= x_3^0 + x_3^1 + x_3^2 \end{aligned}$$

Equation 26-3.

Here, the U_n 's represent physical quantities and the x_n^k terms represent symmetrical components. The superscripts k denote the order of the symmetric components. The subscripts n are taken to be the number of a particular vector in the vector field. In the general context of cyclic symmetry, the subscripts n denote the number applied to each of the N identical objects under consideration. The object $n = 1$ is arbitrarily taken as the fundamental region.

Understanding Rotational Symmetry

In rotational symmetry, [Figure 26-1\(b\)](#), the structure is comprised of identical segments that are symmetrically arranged about an axis. Common examples of structures with rotational symmetry

are skew slabs and many types of rotating machinery including gears, electric motors, turbines, and helicopter rotors.

With rotational symmetry, if the structure has only two segments, the second segment is created by rotating the first segment through 180 about an axis A. In general, the n-th segment is created by rotating the fundamental region (i.e., the first segment) through $360(n-1)/N$ degrees about axis A where N is the total number of segments.

Mathematically, a structure has rotational symmetry if each of its relevant physical properties Q repeats at even angular intervals around an axis A. Thus, referring to the structure shown in [Figure 26-2](#),

$$Q(\theta + \theta_n) = Q(\theta)$$

Equation 26-4.

where:

$$\theta_n = \theta_n = \frac{2\pi(n-1)}{N} \quad \text{and } n = 1, 2, 3, \dots, N.$$

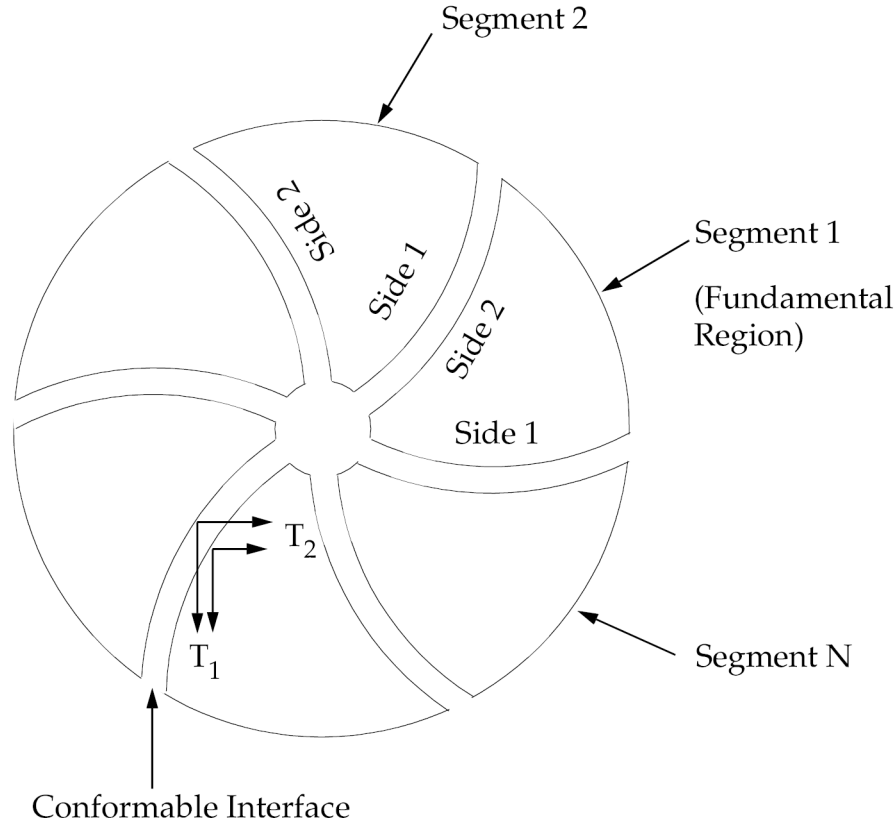


Figure 26-2. Conforming Boundaries for the ROT Option

You can apply rotational symmetry by using the following procedure:

1. Model one segment. Segment boundaries may be curved surfaces.
2. Ensure each segment has its own coordinate system that rotates with the segment. The local displacement coordinate system must conform at the joining points.
3. Specify a paired list of points on side 1 and side 2 that are to be joined.

Transforming Physical Components to Cyclic Components (ROT)

These transformations for the cyclic symmetric components in terms of the general field vectors may be written in summation form for the general case of N vectors.

$$x_1^k = \frac{1}{N} \sum_{n=1}^N U_n e^{i(n-1)ka}$$

Equation 26-5.

where a is given by [Eq. 26-1](#).

Transforming Cyclic Components Back to Physical Components (ROT)

After the solution phase of the problem, it will normally be necessary to transform the results from cyclic components back into physical components.

Eq. 26-3 may be written in the following form.

$$\begin{aligned}
 U_n &= \sum_{k=0}^{N-1} x_1^k e^{-1(n-1)ka} \\
 &= x_1^0 + [x_1^1 e^{-1(n-1)a} + x_1^{(N-1)} e^{i(n-1)a}] \\
 &\quad + [x_1^2 e^{-1(n-1)a} + x_1^{(N-2)} e^{i(n-1)2a}] \\
 &\quad + x_1^3 e^{-1(n-1)a} + x_1^{(N-3)} e^{i(n-1)3a} \\
 &\quad + \dots, \\
 &\quad + x_1^{N/2} (-1)^{(n-1)}
 \end{aligned}$$

Equation 26-6.

where,

$$e^{-1(n-1)(N-k)a} = e^{-1(n-1)ka} e^{-1(n-1)Na} = e^{-1(n-1)ka}$$

and

$$e^{-1(n-1)(N/2)a} = (-1)^{n-1}$$

If the exponentials in the bracketed terms in the above summation are replaced by the relations $e^{\pm ix} = \cos x \pm i \sin x$, the typical bracketed term takes the following form:

$$[(x_1^k + x_1^{(N-k)}) \cos(n-1)ka - i(x_1^k - x_1^{(N-k)}) \sin(n-1)ka]$$

The purpose for the somewhat arbitrary introduction of Eq. 26-6 is now recognized, and the typical bracketed term assumes the following form:

$$[U^{kc} \cos(n-1)ka + U^{ks} \sin(n-1)ka]$$

The final form of the transformation from cyclic components to physical components is therefore,

$$U_n = x_1^0 + \sum_{k=1}^{K_L} [U^{kc} \cos(n-1)ka + U^{ks} \sin(n-1)ka] + (-1)^{n-1} X_1^{N/2}$$

$$n = 1, 2, 3, \dots, N$$

Figure 26-3.

In Eq. 26-3, U_n represents physical quantities such as displacement force, stress, temperature in the n -th segment in terms of the cyclic components in the fundamental region. The following points should be noted:

1. $K_L = \frac{(N-1)}{2}$ and $x_1^{N/2}$ is nonexistent when N is odd.
2. $K_L = \frac{(N-2)}{2}$ when N is even.
3. The notation for cyclic components of order $k = 0$ and $k = N/2$ is distinct from the notation for cyclic components of order $1 \leq k \leq K_L$ because of the somewhat peculiar nature of these two components.

Boundary Conditions and Problem Solution (ROT)

The transformation equations of “Cyclic Symmetry Theory” are assumed to allow one to transform a problem that is mathematically defined in terms of physical components into a set of mathematical statements written in terms of cyclic components. Specifically, the mathematical description of the n replications of the fundamental region may be written in the form

$$\{F_n\} = [Y_n]\{U_n\}$$

Equation 26-7.

This expression represents the mathematical description of the system in terms of physical coordinates.

The application of previously defined transform equations may now be introduced to obtain the desired mathematical description of the system in terms of cyclic components. First, however, consider the matrix representation of the transform Eq. 26-33 and Eq. 26-3, respectively. Eq. 26-33 assumes the form

$$\begin{Bmatrix} x_1^0 \\ U^{1c} \\ U^{1s} \\ U^{2c} \\ \vdots \\ U^{K_L s} \\ x_1^{N/2} \end{Bmatrix} = \begin{bmatrix} \frac{1}{N} & \frac{1}{N} & \frac{1}{N} & \cdots & \frac{1}{N} \\ \frac{2}{N} & \frac{2}{N} \cos a & \frac{2}{N} \cos 2a & \cdots & \frac{2}{N} \cos (N-1)a \\ 0 & \frac{2}{N} \sin a & \frac{2}{N} \sin 2a & \cdots & \frac{2}{N} \sin (N-1)a \\ \frac{2}{N} & \frac{2}{N} \cos 2a & \frac{2}{N} \cos 4a & \cdots & \frac{2}{N} \cos (N-1)2a \\ \cdots & \cdots & \cdots & \cdots & \cdots \\ 0 & \frac{2}{N} \sin K_L a & \frac{2}{N} \sin 2K_L a & \cdots & \frac{2}{N} \sin (N-1)K_L a \\ \frac{1}{N} & -\frac{1}{N} & \frac{1}{N} & \cdots & \frac{1}{N} (-1)^{N-1} \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ \vdots \\ \vdots \\ \vdots \\ U_N \end{Bmatrix}$$

$$\{U^k\} = [T_{xU}]\{U_n\}$$

Equation 26-8.

Eq. 26-3 assumes the form

$$\begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ \vdots \\ U_N \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 0 & 1 & \cdots & 0 & 1 \\ 1 & \cos a & \sin a & \cos 2a & \cdots & \sin K_L a & -1 \\ 1 & \cos 2a & \sin 2a & \cos 4a & \cdots & \sin 2K_L a & 1 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & \cos (N-1)a & \sin (N-1)a & \cos (N-1)2a & \cdots & \sin (N-1)K_L a & (-1)^{(N-1)} \end{bmatrix} \begin{Bmatrix} x_1^0 \\ U^{1c} \\ U^{1s} \\ \vdots \\ U^{K_L s} \\ x_1^{N/2} \end{Bmatrix}$$

$$\{U_n\} = [T_{Ux}]\{U^k\}$$

Equation 26-9.

It is easily demonstrated that

$$[T_{xU}][T_{Ux}] = [I]$$

Return now to Eq. 26-7, the mathematical representation of the system in terms of physical coordinates, and represent the energy of the system in the form

$$U = [U_n]^T \{F_n\}$$

Equation 26-10.

From Eq. 26-7, Eq. 26-9, and Eq. 26-10

$$U = \begin{cases} \{U_n\}^T \{Y_n\} \{U_n\} \\ \{U^k\}^T [T_{Ux}]^T [Y_n] [T_{Ux}] \{U^k\} \end{cases}$$

Equation 26-11.

The system energy in terms of cyclic components may be written as

$$U = \{U^k\}^T [Y_k] \{U^k\}$$

Equation 26-12.

Since the energy must be invariant to the transformations, Eq. 26-12 and Eq. 26-13 may be equated

$$[Y_k] = [T_{Ux}]^T [Y_n] [T_{Ux}]$$

Equation 26-13.

That is, the equations of the system in terms of physical components are transformed into a set of equations in terms of cyclic components. As there is no coupling between the N identical objects of the system, the matrix Y_n may be written so that it consists of N identical blocks along the diagonal. These blocks represent, in general, the equations of motion of each of the N identical objects of the system.

As an example, assume $N = 4$ ($K_L = 1$), and expand Eq. 26-13. Note that $Y_1 = Y_2 = Y_3 = Y_4$.

$$\begin{aligned}
 [Y_k] &= Y_1 \begin{bmatrix} 1 & 1 & 1 & 1 \\ 1 & \cos a & \cos 2a & \cos 3a \\ 0 & \sin a & \sin 2a & \sin 3a \\ 1 & -1 & 1 & -1 \end{bmatrix} \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1 \end{bmatrix} \begin{bmatrix} 1 & 1 & 0 & 1 \\ 1 & \cos a & \sin a & -1 \\ 1 & \cos 2a & \sin 2a & 1 \\ 1 & \cos 3a & \sin 3a & -1 \end{bmatrix} \\
 &= Y_1 \begin{bmatrix} 4 & 0 & 0 & 0 \\ 0 & 2 & 0 & 0 \\ 0 & 0 & 2 & 0 \\ 0 & 0 & 0 & 4 \end{bmatrix}
 \end{aligned}$$

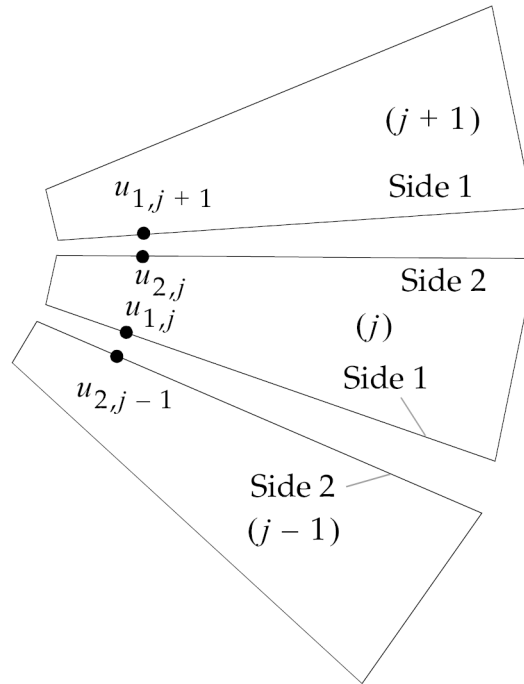
In general, for N objects the above equation may be written

$$\begin{aligned}
 Y_k &= Y_1 N \text{ for } k = 0, N/2 \\
 Y_k &= \frac{N}{2} Y_1 \text{ for } 1 \leq k \leq K_L
 \end{aligned}$$

These equations imply that the equations of motion in terms of cyclic components are either N or $N/2$ times the corresponding equations written in terms of physical components. This argument provides an explanation for the presence of the coefficients that appear in the terms of Eq. 26-8.

To this point, it has been found that except for boundary conditions, the equations for motion in terms of cyclic components ($k = 0, 1, \dots, N/2$) may be developed from the equations of motion of the fundamental region ($n = 1$). In essence, an arbitrarily large physical system may be investigated in terms of a fundamental region which has been transformed into a number of models defined in terms of cyclic components. The boundary conditions between cyclic models must now be defined.

The only connection between the n identical objects in question exists along the boundaries between adjacent segments. Specifically, consider three of the N objects as indicated in the following figure.



These objects are displaced from one another to more clearly define the boundary conditions between the objects. Note the designation of the two sides of each of the elements as side 1 and side 2 as one proceeds in the counterclockwise direction. From the requirement of displacement compatibility, it is necessary that the typical boundary degrees-of-freedom, u , satisfy the equations

$$\begin{aligned} u_{2,j-1} &= u_{1,j} \\ u_{2,j} &= u_{1,j+1} \end{aligned}$$

Equation 26-14.

These physical degrees-of-freedom may be transformed into cyclic components of order K through [Eq. 26-33](#).

$$\begin{aligned} (x^k)_{\text{side 1}} &= \frac{1}{N} u_{1,j} e^{i(j-1)ka} \\ (x^k)_{\text{side 2}} &= \frac{1}{N} u_{1,j+1} e^{ijk a} \end{aligned}$$

Equation 26-15.

A comparison of Eq. 26-15 provides the result

$$(x^k)_{\text{side 2}} = (x^k)_{\text{side 1}} e^{ika}$$

Equation 26-16.

for $1 \leq k \leq k_L$. For $k = 0$,

$$(x^k)_{\text{side 2}} = (x^k)_{\text{side 1}}$$

Equation 26-17.

and for $k = \frac{N}{2}$,

$$(x^k)_{\text{side 2}} = -(x^k)_{\text{side 1}}$$

Equation 26-18.

The cyclic component of Eq. 26-18 may be written in terms of the variables U^{kc} and U^{ks} in accordance with the definitions.

$$\begin{aligned} (U^{kc})_{\text{side 1}} &= [(x^k)_{\text{side 1}} + (x^{N-k})_{\text{side 1}}] \\ (U^{ks})_{\text{side 1}} &= -i[(x^k)_{\text{side 1}} - (x^{N-k})_{\text{side 1}}] \\ (U^{kc})_{\text{side 2}} &= [(x^k)_{\text{side 2}} + (x^{N-k})_{\text{side 2}}] \\ (U^{ks})_{\text{side 2}} &= -i[(x^k)_{\text{side 2}} - (x^{N-k})_{\text{side 2}}] \end{aligned}$$

Eq. 26-16 may now be introduced to yield the equations

$$\begin{aligned} (U^{kc})_{\text{side 2}} &= [(x^k)_{\text{side 1}} e^{ika} + (x^{N-k})_{\text{side 1}} e^{-ika}] \\ (U^{ks})_{\text{side 2}} &= -i[(x^k)_{\text{side 1}} e^{ika} - (x^{N-k})_{\text{side 1}} e^{-ika}] \end{aligned}$$

Upon the replacement of the exponentials with Euler's equation, the appropriate boundary conditions may be represented as follow:

$$\begin{Bmatrix} (U^{kc})_{\text{side 2}} \\ (U^{ks})_{\text{side 2}} \end{Bmatrix} = \begin{bmatrix} \cos ka & -\sin ka \\ \sin ka & \cos ka \end{bmatrix} \begin{Bmatrix} (U^{kc})_{\text{side 1}} \\ (U^{ks})_{\text{side 1}} \end{Bmatrix}$$

Equation 26-19.

These boundary conditions are of the form of constraint equations in which the variables on side 2 are dependent and the variables on side 1 are independent. Indeed, these equations may be viewed as MPC equations that couple kc and ks terms of side 1 to the corresponding terms of side 2.

The equations of motion in terms of cyclic components may be written in the following form.

$$[Y_1] \begin{bmatrix} \frac{N}{2} & & & & & \\ & N & & & & \\ & & N & & & \\ & & & N & & \\ & & & & \ddots & \\ & & & & & \ddots & \\ & & & & & & N \\ & & & & & & & \frac{N}{2} \end{bmatrix} \begin{Bmatrix} x^0 \\ U^{1c} \\ U^{1s} \\ U^{2c} \\ \vdots \\ \vdots \\ \vdots \\ U^{KLs} \\ x^{N/2} \end{Bmatrix} = \begin{Bmatrix} F^0 \\ F^{1c} \\ F^{1s} \\ F^{2c} \\ \vdots \\ \vdots \\ \vdots \\ F^{KLs} \\ F^{N/2} \end{Bmatrix}$$

Equation 26-20.

From the previous discussion on boundary conditions, the above equation may be rewritten in terms of the following disjoint problems.

$$\begin{aligned} k &= 0 & [Y_0] \{X^0\} &= F^0 \\ 1 \leq k \leq K_L & & \begin{bmatrix} Y_K & | & 0 \\ \hline 0 & | & \bar{Y}_k \end{bmatrix} \begin{Bmatrix} U^{kc} \\ U^{ks} \end{Bmatrix} &= \begin{Bmatrix} F^{kc} \\ F^{ks} \end{Bmatrix} \\ k &= \frac{N}{2} & [Y_{N/2}] \{x^{N/2}\} &= \{F^{N/2}\} \end{aligned}$$

Equation 26-21.

The boundary [Eq. 26-17](#), [Eq. 26-18](#), and [Eq. 26-19](#) must be introduced to remove the dependent variables that are defined to be on side 2 of the region. If you also use the OMIT feature, the only independent degrees-of-freedom in the above disjoint problems are the degrees-of-freedom on side

1 that have not previously been declared dependent through MPC, SPCi, and/or OMITi Bulk Data entries. Thus, the matrices in the above disjoint problems, although dense, are of small size relative to, say, the matrices required to perform the same analysis by more traditional methods.

Understanding the Harmonic Index

The harmonic index k plays an important role in the analysis of structures with cyclic symmetry. The harmonic index k has the values $k = 1, 2, \dots, N/2$. If the number of segments is large, the higher harmonics may contribute very little to the total solution. In NX Nastran you can select the harmonics (values of k) to use in the analysis.

If the only nonzero symmetrical load components are for a particular value of k , then the only nonzero symmetrical response components are for the same value of k , provided only that the structure is linear and satisfies Eq. 26-4. This fact leads directly to an efficient method of analysis in which each symmetric load component is computed from Eq. 26-23 and applied to a finite element model that is very much smaller than the complete structure.

The symmetrical components are evaluated from the physical components by means of the following summations that replace the Fourier integrals used in axisymmetric analysis. The argument (ρ) is dropped for convenience.

$$\begin{aligned}\bar{u}^0 &= \frac{1}{N} \sum_{n=1}^N u^n \\ \bar{u}^{ks} &= \frac{2}{N} \sum_{n=1}^N u^n \sin(k\theta_n) & k &= 1, 2, \dots, k_L \\ \bar{u}^{kc} &= \frac{2}{N} \sum_{n=1}^N u^n \cos(k\theta_n) & k &= 1, 2, \dots, k_L \\ \bar{u}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} u^n\end{aligned}$$

Equation 26-22.

Note that stiffness damping, and mass matrices become uncoupled for \bar{u}^k .

Eq. 26-22 also applies to the expansion of applied loads into symmetrical component sets. Thus

$$\begin{aligned}
 \bar{P}^0 &= \frac{1}{N} \sum_{n=1}^N P^n \\
 \bar{P}^{ks} &= \frac{2}{N} \sum_{n=1}^N P^n \sin(k\theta_n) & k = 1, 2, \dots, k_L \\
 \bar{P}^{kc} &= \frac{2}{N} \sum_{n=1}^N P^n \cos(k\theta_n) & k = 1, 2, \dots, k_L \\
 \bar{P}^{N/2} &= \frac{1}{N} \sum_{n=1}^N (-1)^{n-1} P^n
 \end{aligned}$$

Equation 26-23.

The implied coordinate systems in the physical segments remain fixed in the segments and are rotated with them. To satisfy continuity between segments, the grid point on side 2 of segment 1 must coincide with the grid point on side 1 of segment 2 and their coordinate systems must be aligned (see [Figure 26-2](#)). As a result, you must align the grid points on sides 1 and 2 of the fundamental region so that their positions and displacement coordinate systems differ only by a rotation through the angle $a = 2\pi/N$ about the axis of symmetry. This is most easily accomplished by using a cylindrical coordinate system for the grid points on sides 1 and 2. If this is done, the boundary conditions are satisfied by imposing the following constraints on the symmetrical components of displacement at corresponding points on sides 1 and 2 (indicated by subscripts).

$$\begin{aligned}
 \bar{u}_1^0 &= \bar{u}_2^0 \\
 \bar{u}_1^{kc} \cos(ka) + \bar{u}_1^{ks} \sin(ka) &= \bar{u}_2^{kc} \\
 -\bar{u}_1^{kc} \sin(ka) + \bar{u}_1^{ks} \cos(ka) &= \bar{u}_2^{kc} \\
 &k = 1, 2, \dots, k_L \\
 -\bar{u}_1^{N/2} &= \bar{u}_2^{N/2}
 \end{aligned}$$

Equation 26-24.

Rotational Symmetry Example

As an example of the application, consider the simple problem illustrated in [Figure 26-4](#). The skew slab is divided into two identical segments. The displacement coordinate system for the second

segment is rotated 180° from the first. A cylindrical coordinate system is used (at least along sides 1 and 2) to ensure the compatibility of displacement components at the points in the boundaries between segments.

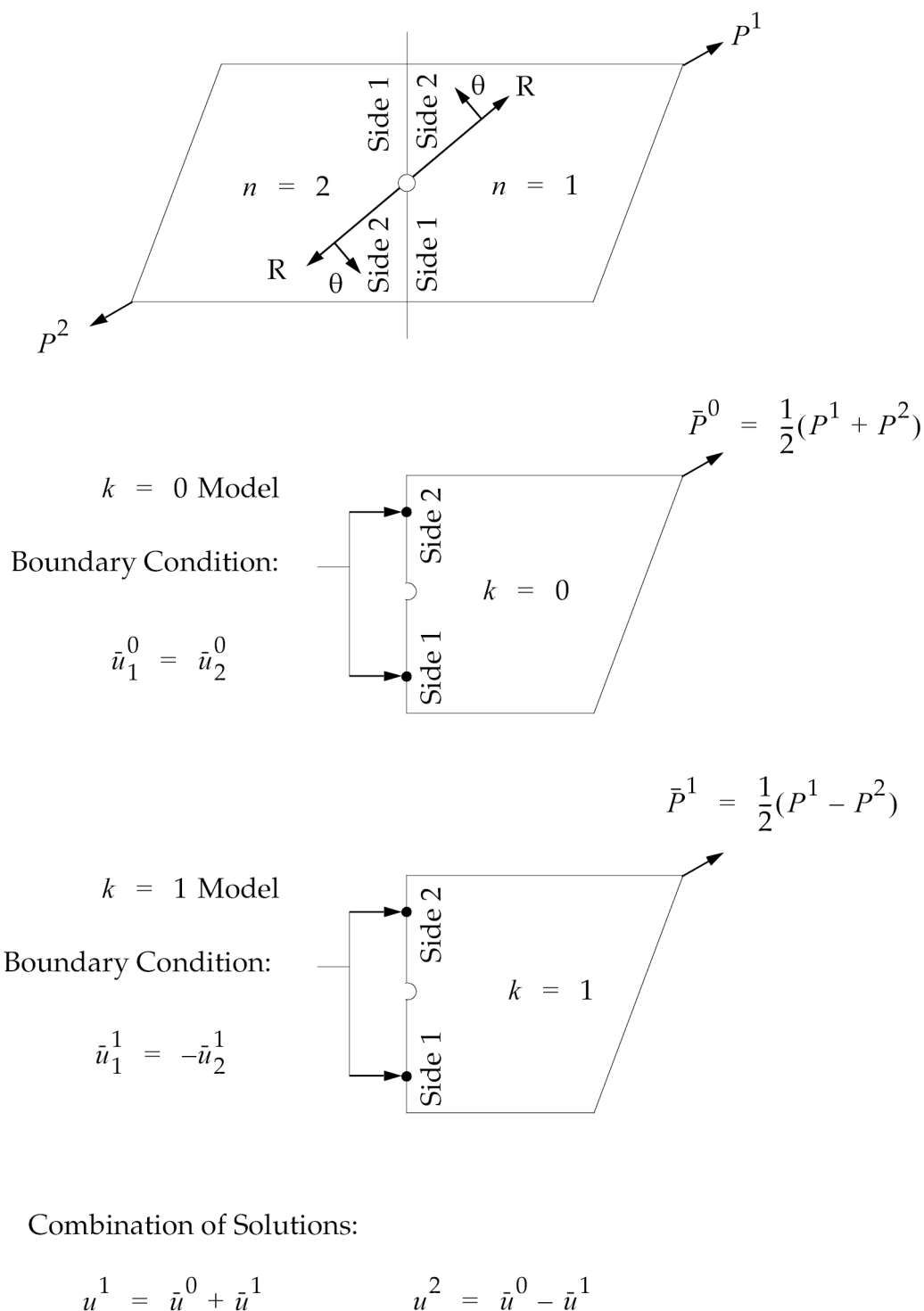


Figure 26-4. Example of Rotational Symmetry

Since $N = 2$, there are two symmetrical component sets \bar{u}^0 and $\bar{u}^{N/2} = \bar{u}^1$. The symmetrical load components, as computed from Eq. 26-23, are

$$\begin{aligned}\bar{P}^0 &= \frac{1}{2}(P^1 + P^2) \\ \bar{P}^1 &= \frac{1}{2}(P^1 - P^2)\end{aligned}$$

Equation 26-25.

\bar{P}^0 and \bar{P}^1 are applied to separate copies of the finite element model for the fundamental region labeled the $K = 0$ model and the $K = 1$ model. The boundary condition linking all six degrees of freedom at corresponding pairs of grid points on sides 1 and 2, as computed from Eq. 26-5, are

$$\begin{aligned}\bar{u}_1^0 &= \bar{u}_2^0 \\ \bar{u}_1^1 &= -\bar{u}_2^1\end{aligned}$$

Equation 26-26.

These boundary conditions can be supplied in Solution 101 with multipoint constraints; however, they are provided automatically in NX Nastran's cyclic symmetry capability.

The $K = 0$ and $K = 1$ models are solved in the usual way to obtain the symmetrical response components \bar{u}^0 and \bar{u}^1 . As a final step, the physical responses are evaluated as follows from Eq. 26-22

$$\begin{aligned}u^1 &= \bar{u}^0 + \bar{u}^1 \\ u^2 &= \bar{u}^0 - \bar{u}^1\end{aligned}$$

Equation 26-27.

Note: You could use the static condensation procedure in this problem. With that procedure, all the interior degrees of freedom are placed in the u_o set and eliminated, resulting in a greatly reduced stiffness matrix for the remaining degrees of freedom on the boundary (sides 1 and 2). Since this only needs to be done once, there is a net saving of computer time.

See also

- “Static Condensation (Guyan Reduction)” in the *NX Nastran User's Guide*

Dihedral Symmetry

Dihedral symmetry, **Figure 26-1(c)**, is a special case of rotational symmetry that combines rotational and reflective symmetry, since each segment has its own plane of reflective symmetry. Thus, in **Figure 26-1(c)**, half-segment 1L is the mirror image of half-segment 1R (the fundamental region), and these two halves of segment 1 are rotated together through 120° to form segment 2 and through 240° to form segment 3. Water towers and reinforced domed roofs are familiar examples of structures with dihedral symmetry.

Dihedral symmetry (DIH) can be useful because:

- DIH takes advantage of reflective symmetry to reduce the fundamental region by one-half relative to the size of the fundamental region required in the ROT option.
- You can use the DIH option for analyses that would normally be handled with traditional reflective symmetry techniques.

Figure 26-5 and **Figure 26-6** show two different examples of dihedral symmetry and the notation system used for the DIH option in NX Nastran.

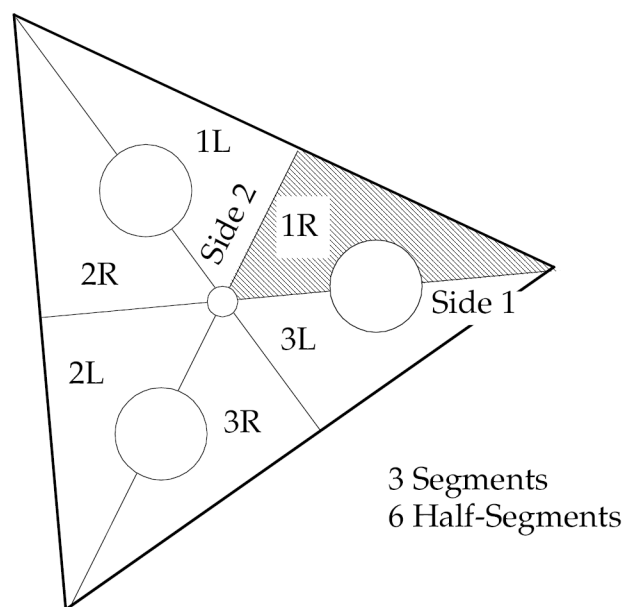


Figure 26-5. Dihedral Symmetry

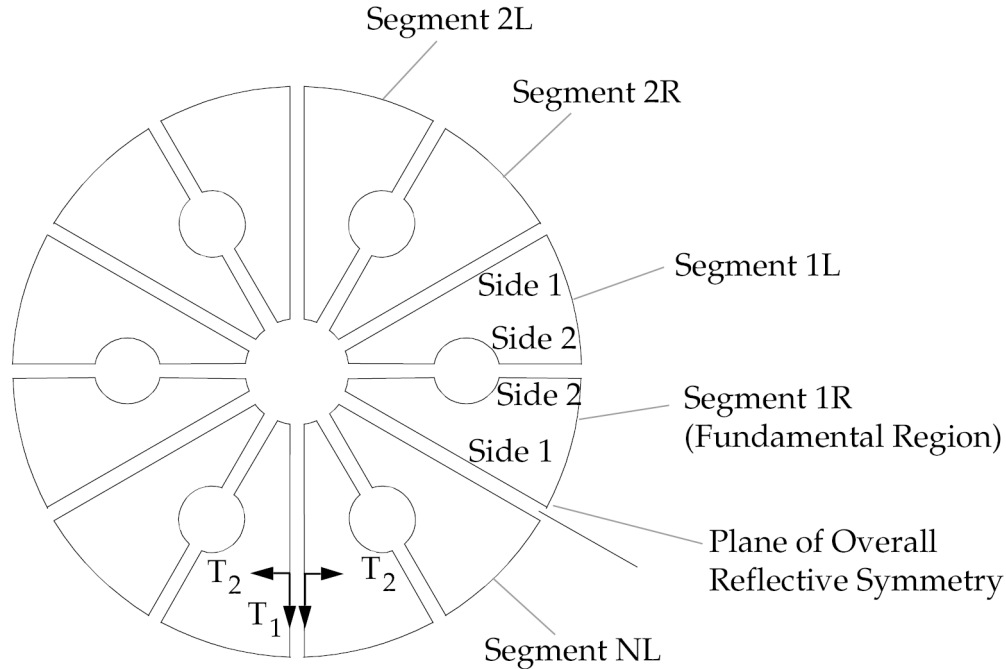


Figure 26-6. Dihedral Symmetry

The fundamental region, for which you create a finite element model, is the right half of the first segment (1R). With dihedral symmetry, all segment boundaries must be planar.

- The plane that forms the boundary between the right and left halves of segment 1 is labeled side 2.
- The other boundary of the fundamental region, which must also be planar, is labeled side 1.

Note that:

- Local displacement systems axes, associated with the inter-segment boundaries, must be in the plane or normal to the plane.
- In the specific case of a structure with a single overall plane of reflective symmetry (i.e., the complete structure is represented by segments 1R and 1L) the reflective plane may be defined as side 1 or side 2.
- The two halves of a segment that join at a reflective plane are denoted as the right, “R”, and left, “L”, halves. The L half segments are mirror images of the R half segments.
- R halves use a right-handed coordinate system and L halves use a left-handed coordinate system. Therefore, at each boundary a degree-of-freedom is associated with its mirror image on the adjacent segment so that compatibility always involves a left- and a right-hand coordinate system. The degrees-of-freedom on the boundaries are further categorized into odd and even components in accordance with the following table.

Component	Degrees-of-Freedom
Even	Displacement vectors parallel to the boundary and rotation vectors normal to the boundary.

Component	Degrees-of-Freedom
Odd	Displacement vectors normal to the boundary and rotation vectors parallel to the boundary.

This has important implications for how you input of loads data and interpret the output. The coordinate systems for other half-segments in [Figure 26-5](#) are either rotated, or rotated and reflected as indicated in [Table 26-2](#).

Table 26-2. Coordinate Systems for Half Segments		
Half-Segment	Type of Coordinate System	Rotation About the Axis of Symmetry
1R	Right-handed	0°
1L	Left-handed	0°
2R	Right-handed	120°
2L	Left-handed	120°
3R	Right-handed	240°
3L	Left-handed	240°

The typical procedure that you follow when using dihedral symmetry is:

1. You model one half-segment (1R).
2. At a boundary between segments or half-segments, the displacement components must be parallel and perpendicular to the boundary. The boundaries are planes passing through the axis of symmetry.
3. You provide separate lists of the points on sides 1 and 2 which must be joined.

As in rotational symmetry, the response quantities at corresponding points in the physical segments can be expressed as a truncated Fourier series with the polar angle of the n -th segment θ_n as the argument where N is the number of segments. The series for response quantities in the right half-segments is

$$u^{nR} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} + \bar{u}^{kc*}) \cos(k\theta_n) + (\bar{u}^{ks} + \bar{u}^{ks*}) \sin(k\theta_n)]$$

Equation 26-28.

where k_L is the largest integer less than or equal to $N/2$ (depending on whether n is odd or even) and \bar{u}^{kc} , \bar{u}^{kc*} , \bar{u}^{ks} , and \bar{u}^{ks*} are symmetrical component sets. The series for response quantities in the left half-segments is

$$u^{nL} = \sum_{k=0}^{k_L} [(\bar{u}^{kc} - \bar{u}^{kc*}) \cos(k(\theta_n + a)) - (\bar{u}^{ks} - \bar{u}^{ks*}) \sin(k(\theta_n + a))]$$

Equation 26-29.

where, in addition to previously defined terms, $a = 2\pi/N$.

The symmetrical component sets are evaluated from the physical components as follows:

$$\begin{Bmatrix} \bar{u}^{kc} \\ \bar{u}^{kc*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \cos(k\theta_n) \pm u^{nL} \cos(k(\theta_n + a))]$$

Equation 26-30.

$$\begin{Bmatrix} \bar{u}^{ks} \\ \bar{u}^{ks*} \end{Bmatrix} = \frac{\delta}{N} \sum_{n=1}^N [u^{nR} \sin(k\theta_n) \mp u^{nL} \sin(k(\theta_n + a))]$$

Equation 26-31.

where, in addition to previously defined terms, $\delta = 1/2$ for $k = 0$ and $N/2$, and $\delta = 1$ for all other values of k .

The number of symmetrical components sets is equal to the number of half-segments in the structure (neglecting the identically null sets \bar{u}^{ks} and \bar{u}^{ks*} for $k = 0, N/2$). As in the case of rotational symmetry, the symmetrical load components for a particular value of k produce only the symmetrical response

components for the same value of k , or, to put it another way, the analyses for the different values of k are uncoupled.

Furthermore, the analyses for the starred (*) and unstarred symmetrical components are uncoupled. For $K = 0$ and $k = N/2$, the complete finite element model for the unstarred terms consists of a single copy of the fundamental region (1 R). For $0 < k < N/2$, the complete finite element model consists of two copies of the fundamental region with coupling between their boundaries. The boundary conditions are different for each value of k . The finite element models for the starred (*) terms are identical to those for the unstarred terms for $k > 0$. Only the symmetrical load components, which are computed by Eq. 26-30 and Eq. 26-31, are different.

As an example, consider the case of a structure with two planes of reflective symmetry as shown in Figure 26-7. Applying dihedral symmetry to this case, you can see that

- The number of segments $N = 2$.
- The segment angles $a = n$.
- $\theta^1 = 0$.
- $\theta^2 = \pi$
- The harmonic indices $k = 0, 1$.

Substitution of these values into Eq. 26-30 and Eq. 26-31 gives

$$\begin{aligned}\bar{u}^{0c} &= \frac{1}{4}(u^{1R} + u^{1L} + u^{2R} + u^{2L}) \\ \bar{u}^{1c} &= \frac{1}{4}(u^{1R} - u^{1L} - u^{2R} + u^{2L}) \\ \bar{u}^{0c*} &= \frac{1}{4}(u^{1R} - u^{1L} + u^{2R} - u^{2L}) \\ \bar{u}^{1c*} &= \frac{1}{4}(u^{1R} + u^{1L} - u^{2R} - u^{2L}) \\ \bar{u}^{0s} &= \bar{u}^{0s*} = \bar{u}^{1s} = \bar{u}^{1s*} = 0\end{aligned}$$

Equation 26-32.

Since all of the signs are positive in the equation for \bar{u}^{0c} , it follows that \bar{u}^{0c} is symmetrical with respect to both side 1 and side 2 in Figure 26-7. The other nonzero symmetrical components are antisymmetrical with respect to one or both of the sides as shown in Table 26-3.

Table 26-3. Symmetrical Components for Reflective Symmetry Example

Symmetrical Component	Symmetry	
	Side 1	Side 2
\bar{u}^{0c}	S	S
\bar{u}^{1c}	S	A
\bar{u}^{0c*}	A	A
\bar{u}^{1c*}	A	S

Regardless of the number of segments, the starred terms are always antisymmetrical, and the unstarred terms are always symmetrical with respect to side 1 of the fundamental region. You can take advantage of this to reduce the amount of calculation when it can be determined beforehand that the applied loading has a plane of symmetry or antisymmetry that coincides with side 1 of the fundamental region, or, in other words, if the problem has an overall plane of symmetry. The amount of computation can be reduced still further if the problem has a second overall plane of symmetry perpendicular to the first. The second overall plane of symmetry can exist only if there is an even number of segments.

Summary of Transformations, Boundary Conditions, and Problem Solution (DIH)

The form of the transformations between cyclic and physical components and the boundary conditions between the segments of the object in question depend somewhat on the notation system.

When you select the DIH option of cyclic symmetry, you can take advantage of overall planes of symmetry when the applied loads are symmetrically and/or antisymmetrically disposed with respect to these planes. In these cases, you only need to specify loads for segments in the first half plane when there is one plane of overall symmetry and in the first quadrant when there are two planes of overall symmetry.

You specify symmetrical or antisymmetrical boundary conditions on the DSYM Case Control command. You can only include one DSYM command per subcase. A single plane of reflective symmetry must be with respect to Side 1, and if a second plane of reflective symmetry exists, it must be perpendicular to Side 1. In the case of two planes of symmetry, the model must contain an even number of whole segments.

The following sections describe several of the equations of interest for the DIH option, along with the equation number for the corresponding expression under the ROT option. In the following equations, the starred (*) terms donate anti-symmetric motions and unstarred terms denote symmetric motions. The subscripts R and L refer to the right and left halves respectively.

See also

- “**DSYM**” in the *NX Nastran Quick Reference Guide*

Transforming Physical Components to Cyclic Components (DIH)

(See [Eq. 26-3](#) for the corresponding equations for the ROT option.)

$$\left. \begin{matrix} U^{kc} \\ U^{kc*} \end{matrix} \right\} = \frac{\delta}{N} \sum_n [U_{n,R} \cos(n-1)ka \pm U_{n,L} \cos(N-n)ka]$$

$$\left. \begin{matrix} U^{ks} \\ U^{ks*} \end{matrix} \right\} = \frac{\delta}{N} \sum_n [U_{n,R} \sin(n-1)ka \pm U_{n,L} \sin(N-n)ka]$$

Equation 26-33.

Here,

$$\delta = \begin{cases} 1/2 & \text{for } k = 0, N/2 \\ 1 & \text{for } 1 \leq k \leq K_L \end{cases}$$

Equation 26-34.

and the positive sign is associated with the unstarred terms.

Transforming Cyclic Components to Back to Physical Components (DIH)

(See Eq. 26-19 for the corresponding relation for the ROT option.)

$$U_{n,R} = \sum_k (U^{kc} + U^{kc*}) \cos(n-1)ka + (U^{ks} + U^{ks*}) \sin(n-1)ka$$

$$U_{n,L} = \sum_k (U^{kc} + U^{kc*}) \cos(N-n)ka + (U^{ks} + U^{ks*}) \sin(N-n)ka$$

$$0 \leq k \leq \frac{N}{2}$$

Equation 26-35.

Boundary Conditions and Problem Solution (DIH)

(See Eq. 26-3 for the corresponding boundary conditions for the ROT option.)

Side 1, EVEN

$$U^{kc*} = 0$$

$$U^{ks} = 0$$

Equation 26-36.

Side 1, ODD

$$\begin{aligned}U^{kc} &= 0 \\U^{ks*} &= 0\end{aligned}$$

Equation 26-37.

Side 2, EVEN

$$\begin{aligned}U^{kc} \sin \frac{ka}{2} + U^{ks} \cos \frac{ka}{2} &= 0 \\U^{kc*} \cos \frac{ka}{2} - U^{ks*} \sin \frac{ka}{2} &= 0\end{aligned}$$

Equation 26-38.

Side 2, ODD

$$\begin{aligned}U^{kc} \cos \frac{ka}{2} - U^{ks} \sin \frac{ka}{2} &= 0 \\U^{kc*} \sin \frac{ka}{2} + \cos \frac{ka}{2} &= 0\end{aligned}$$

Equation 26-39.

In static analysis with cyclic symmetry, the equation

$$[\bar{K}_{aa}]^K \{\bar{u}_a\}^K = \{\bar{P}_a\}^K$$

is solved in SSG3 (Static Solution Generator – Part 3) for $\{\bar{u}_a\}^K$

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

and

$$\{\bar{P}_a\}^K = [G_{ck}^T] \{\bar{P}\}^{KC} + [G_{sk}^T] \{\bar{P}\}^{ks}$$

are formed in CYCLIC3 (Cyclic Symmetry Transformation – Module 3).

The symmetrical components of the loads $\{\bar{P}\}^{kc}$ and $\{\bar{P}\}^{ks}$ are subsets of $\{\bar{P}\}^x$ which is obtained in CYCLIC3 by multiplying the load vector formed in SSG1 by the GFORE matrix formed in CYCLIC1 module

$$\{\bar{P}\}^x = \{P_l\} \{GFORE\}$$

where $\{P_l\}$ is formed in SSG2 (Static Solution Generator – Part 2) and the transformation matrix [GFORE] is formed in CYCLIC1 module.

The cyclic component constraint data (CYCD) used in CYCLIC3 for the formulation of the transformation matrices $[G_{ck}]$ and $[G_{sk}]$ are generated in CYCLIC2 (Cyclic Symmetry – Module 2).

The solution $\{u_a\}^K$ is transformed to symmetric components in CYCLIC4 as follows:

$$\begin{aligned} \{u_x\}^{kc} &= [G_{ck}] \{\bar{u}_a\}^K \\ \{u_x\}^{ks} &= [G_{sk}] \{\bar{u}_a\}^K \end{aligned}$$

The symmetric components for each harmonic index are appended to form $\{\bar{u}\}^x$ in the CYCLIC4 module. The physical segment displacements are obtained as follows:

$$\{u_a\} = \{\bar{u}\}^x [GBACK]$$

where the transformation matrix is formed in the CYCLIC1 module.

In the case of vibration analysis with cyclic symmetry the following equation is used in READ to determine the eigenvalues and eigenvectors,

$$[\bar{K}_{aa} - \lambda \bar{M}_{aa}] \{\bar{u}\}^K = 0$$

where $[\bar{M}_{aa}]^K$ is formed in a manner similar to $[\bar{K}_{aa}]$ in CYCLIC3. The symmetrical components of the eigenvectors are recovered in CYCLIC4 and the physical segment data is recovered using the GBACK1 matrix created in CYCLIC4 module.

Since all the transformation matrices are extremely sparse, none of the matrix multiplications indicated above will require large amounts of computer time. The most significant operation is the triple product associated with the determination of $[\bar{K}_{aa}]^K$. In the usual application this matrix is dense and approximately a-size. The computer time for the triple product can be estimated as ten matrix packing operations on a full matrix of a-size.

Reflective Symmetry

Reflective symmetry with one or two planes of symmetry is a special case of dihedral symmetry (Figure 26-8).

- Reflective symmetry with one plane of symmetry is identical to dihedral symmetry with one segment.
- Reflective symmetry with two planes of symmetry is identical to dihedral symmetry with two segments.

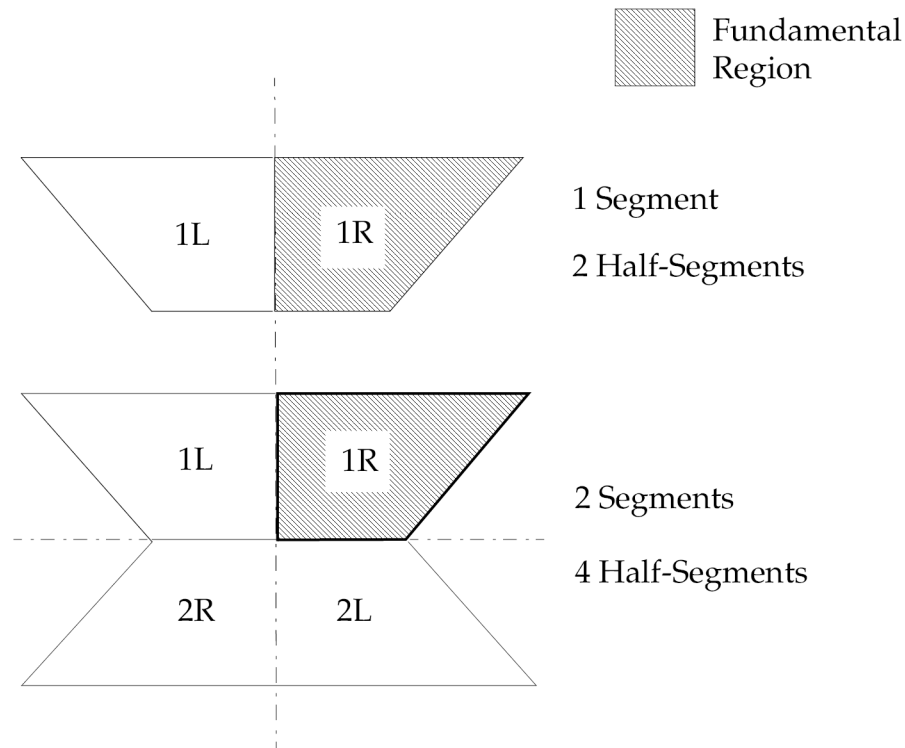


Figure 26-7. Reflective Symmetry with One or Two Planes of Symmetry –A Special Case of Dihedral Symmetry

Mathematically, a structure has reflective symmetry with respect to a yz plane if each of its relevant physical properties Q (such as mass density and elastic modulus) satisfies the equation

$$Q(x, y, z) = Q(-x, y, z)$$

Equation 26-40.

or in other words, if the physical properties are identical at the image points on the two sides of the yz plane. The plane of symmetry divides the structure into two halves that are arbitrarily called the right half and the left half. The right half is also called the fundamental region because, in analysis, it is the half for which a finite element model is prepared, whereas the finite element model for the left half is only implied.

The physical properties included in Q do not include the loads applied to the structure or its responses (displacements, internal forces, or stresses). Let the symbols u_r and u_l represent the values of some particular response u at some point in the right half and at its image point in the left half of the structure. u_r and u_l don't necessarily satisfy the reflective symmetry property Eq. 26-40, but their average u_s clearly does. Thus,

$$u_s = \frac{1}{2}(u_r + u_l)$$

Equation 26-41.

so that

$$u_s(x, y, z) = \frac{1}{2}(u(x, y, z) + u(-x, y, z))$$

Equation 26-42.

and

$$u_s(-x, y, z) = \frac{1}{2}(u(-x, y, z) + u(x, y, z))$$

Equation 26-43.

Since the right-hand sides of Eq. 26-42 and Eq. 26-43 are equal, it follows that

$$u_s(x, y, z) = u_s(-x, y, z)$$

Equation 26-44.

The quantity u_s is called the symmetrical part of the response. The antisymmetrical part of the response u_a is defined as one-half of the difference of u_r and u_l . Thus,

$$u_a = \frac{1}{2}(u_r - u_l)$$

Equation 26-45.

The antisymmetrical part of the response has the property that

$$u_a(x, y, z) = -u_a(-x, y, z)$$

Equation 26-46.

Eq. 26-41 and Eq. 26-45 show how to obtain the symmetrical and antisymmetrical parts of the response from the values of the response for the left and right halves of the structure. They may be written more compactly in matrix form

$$\begin{Bmatrix} u_s \\ u_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ -1 & -1 \end{bmatrix} \begin{Bmatrix} u_r \\ u_l \end{Bmatrix}$$

Equation 26-47.

The vector

$$\begin{Bmatrix} u_s \\ u_a \end{Bmatrix}$$

is called the vector of symmetrical components, and the vector

$$\begin{Bmatrix} u_r \\ u_l \end{Bmatrix}$$

is called the vector of physical components. The physical components may be found from the symmetrical components by solving for them from Eq. 26-47.

$$\begin{Bmatrix} u_r \\ u_l \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} u_s \\ u_a \end{Bmatrix}$$

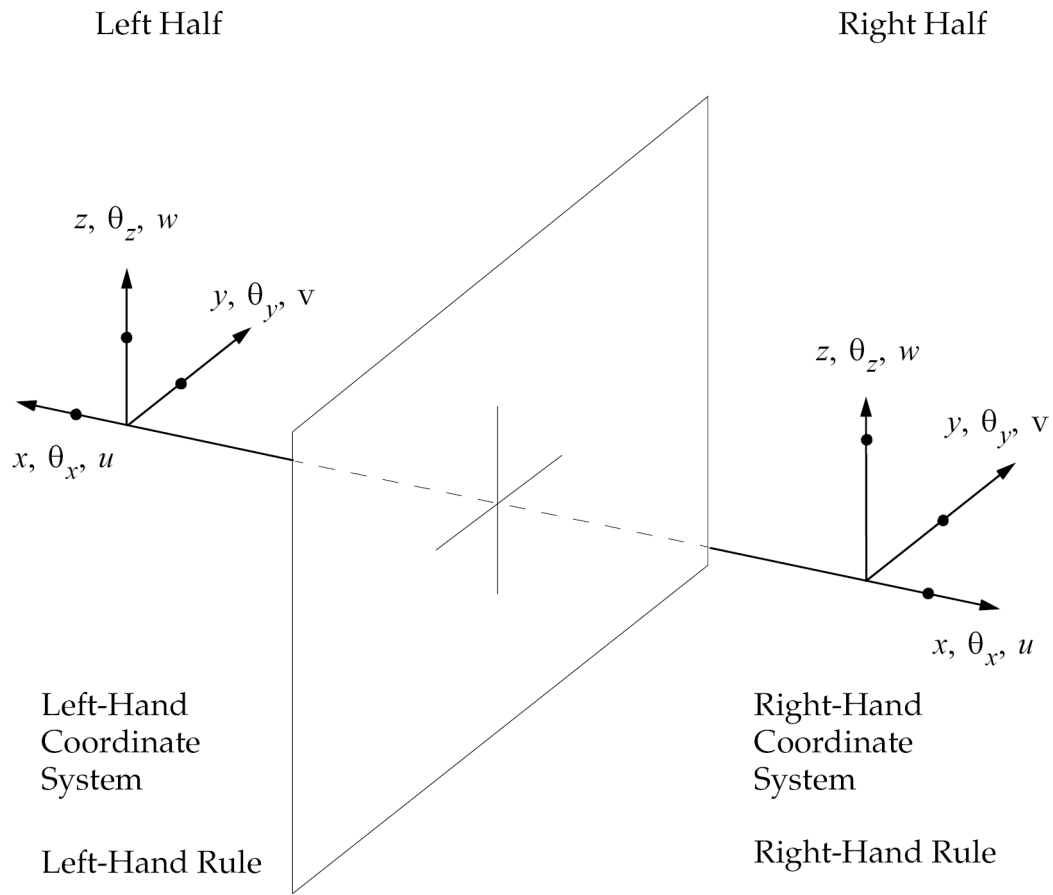
Equation 26-48.

Eq. 26-47 and Eq. 26-48 apply to any response quantity, and they also apply to the applied loads. Thus, any particular applied load P has symmetrical components P_s and P_a evaluated as follows:

$$\begin{Bmatrix} P_s \\ P_a \end{Bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} P_r \\ P_l \end{Bmatrix}$$

Equation 26-49.

In order to include vectors as well as scalars in u and P , it is necessary to select the coordinate systems in the right and left halves to be mirror images of each other as shown in Figure 26-8. The coordinate system in the right half is right-handed, to conform with standard NX Nastran practice. Note, however, that the coordinate system in the implied left half is left-handed. This means, for example, that loads applied indirectly to the left half, by listing the images of their points of application in the right half, must be expressed in the particular left-hand coordinate system shown in Figure 26-8.



Symmetric Boundary Conditions: $u = 0$; $\theta_y = 0$; $\theta_z = 0$

Antisymmetric Boundary Conditions: $v = 0$; $w = 0$; $\theta_x = 0$

Figure 26-8. Reflective Symmetry

If the loads applied to a symmetrical structure are symmetrical, the response is symmetrical. The responses also include the displacements in the plane of symmetry, i.e., u , v , w , θ_x , θ_y , and θ_z at $x = 0$. Since the positive directions of u , θ_y , and θ_z are opposite in the two halves, continuity requires that they be zero at $x = 0$ in a symmetrical solution. Thus, $u = 0$, $\theta_y = 0$, and $\theta_z = 0$ are boundary conditions that can be applied with SPCs if only the right half is represented by a finite element model. The boundary conditions for v , w , and θ_x are natural, i.e., their reaction forces in the plane of symmetry are zero by symmetry, and they may be left unconstrained.

In like manner, if the loads applied to the structure are antisymmetrical, the response is antisymmetrical. By similar reasoning it can be shown that the antisymmetrical boundary conditions for a finite element model of the right half are $v = w = \theta_x = 0$.

If the applied loads are neither symmetrical nor antisymmetrical, the response may be obtained by the following procedure (which is only valid if the response is a linear function of the applied loads):

1. Compute the symmetrical and antisymmetrical load sets P_s and P_a by means of Eq. 26-49.
2. Apply P_s to a finite element model of the right-hand side with symmetric boundary conditions. The response u_s is obtained.
3. Apply P_a to a finite element model of the right-hand side with antisymmetric boundary conditions. The response u_a is obtained.
4. Find the physical responses u_r and u_i by means of Eq. 26-48.

Everything that has been said about one plane of symmetry can be extended to two or three orthogonal planes of symmetry. The steps that were outlined are performed automatically in NX Nastran's cyclic symmetry capability, using the identity between reflective symmetry with one or two planes of symmetry and dihedral symmetry with one or two segments, respectively. The steps may also be performed using basic NX Nastran procedures but with considerable nonautomated work.

See also

- “Symmetry” in the *NX Nastran User's Guide*

Axisymmetry

In axisymmetry, Figure 26-1(e), the object is generated by rotating the fundamental region continuously about an axis, thereby forming a solid of revolution. There are two subclasses of axisymmetry:

- In the first subclass, the fundamental region is a plane surface. This subclass is known as axisymmetry without skew.
- In the second subclass, the fundamental region is a skewed surface. This subclass is known as axisymmetry with skew.

These two subclasses are distinguishable only if the mechanical properties of the fundamental region are anisotropic. Examples of structures with skewed axisymmetry may be found in nature (e.g., the trunks of pine trees) and in spiral-wound reinforced shells.

Mathematically, a structure is axisymmetric (axisymmetric without skew) if each of its relevant physical properties Q (such as mass density and elastic modulus) satisfies the equation

$$Q(r, \theta, z) = Q(r, 0, z)$$

Equation 26-50.

where r , θ , and z are the components of position in a cylindrical coordinate system. The cross section for $\theta = 0$ is the fundamental region whose properties are mapped into the entire structure by Eq. 26-50.

The response of the structure and its applied loads do not necessarily satisfy Eq. 26-50. Let $u(r, \theta, z)$ be some particular response quantity (displacement, internal force, or load). If u is a component of a vector \vec{u} , let the coordinate system that expresses be the cylindrical coordinate system used in connection with Eq. 26-50. Then the Fourier series representation of $u(r, \theta, z)$ may be written as

$$u(r, \theta, z) = \bar{u}^0(r, z) + \sum_{k=1}^{\infty} (\bar{u}^{ks}(r, z) \sin(k\theta) + \bar{u}^{kc}(r, z) \cos(k\theta))$$

Equation 26-51.

or, dropping the explicit dependence on r and z ,

$$u(\theta) = \bar{u}^0 + \sum_{k=1}^{\infty} (\bar{u}^{ks} \sin(k\theta) + \bar{u}^{kc} \cos(k\theta))$$

Equation 26-52.

The harmonic response coefficients \bar{u}^0 , \bar{u}^{ks} , \bar{u}^{kc} and may be regarded as symmetrical component sets that replace $u(\theta)$ in structural analysis. They are evaluated from the physical component $u(\theta)$ by the following Fourier integrals:

$$\begin{aligned}\bar{u}^0 &= \frac{1}{2\pi} \int_0^{2\pi} u(\theta) d\theta \\ \bar{u}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \sin(k\theta) d\theta \\ \bar{u}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} u(\theta) \cos(k\theta) d\theta\end{aligned}$$

Equation 26-53.

Applied loads $P(\theta)$ can also be expanded in a Fourier series. The symmetrical component load sets are, by analogy with Eq. 26-53,

$$\begin{aligned}\bar{P}^0 &= \frac{1}{2\pi} \int_0^{2\pi} P(\theta) d\theta \\ \bar{P}^{ks} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \sin(k\theta) d\theta \\ \bar{P}^{kc} &= \frac{1}{\pi} \int_0^{2\pi} P(\theta) \cos(k\theta) d\theta\end{aligned}$$

Equation 26-54.

Fourier series are often used in the analysis of axisymmetric structures. The first step is to find the harmonic load coefficients from Eq. 26-54. The next step is to find the harmonic response coefficients as functions of the harmonic load coefficients. It can be shown from a consideration of the properties of trigonometric functions that, in linear analysis, \bar{u}^0 depends only on \bar{P}^0 that the harmonic response coefficients for a particular harmonic \bar{u}^{ks} and \bar{u}^{kc} depend only on the harmonic load coefficients \bar{P}^{ks} and \bar{P}^{kc} for the same harmonic index k . Computational effort can be further reduced by noting that higher harmonics where is problem dependent do not contribute significantly to the solution and need not be computed. Once the harmonic response coefficients are found, the physical components can be computed by means of Eq. 26-51.

26.4 Using Cyclic Symmetry

Overview of Performing a Cyclic Symmetry Analysis

In a cyclic symmetry analysis, your total model consists of N identical segments which are numbered consecutively from 1 to N . In your input file, you use the CYSYM bulk data entry to specify the type of cyclic symmetry (rotational, dihedral, or axisymmetric) and the number of segments in the model.

You then generate the model for one segment, using regular elements and standard modeling techniques. You can use any element in the *NX Nastran Element Library*, except axisymmetric elements.

NX Nastran automatically rotates all other segments and their coordinate systems to equally spaced positions about the polar axis. The segment boundaries must be conformable; i.e., when the segments are put together, the grid points and the displacement coordinate systems of adjacent segments must coincide. This is easiest to ensure if you use a cylindrical or spherical coordinate system.

The two boundaries are called Sides 1 and 2.

- In the case of rotational symmetry, Side 2 of segment n is connected to Side 1 of segment $n+1$, as shown in [Figure 26-2](#).
- In the case of dihedral symmetry, Side 1 is on the boundary of the plane of overall reflective symmetry, and Side 2 is on the midplane or reflective plane of the segment, as shown in [Figure 26-6](#).

When you create your input file, you specify the grid point numbers on Sides 1 and 2 on the CYJOIN Bulk Data entry. You also list grid points that lie on the axis of symmetry on the CYAX Bulk Data entry.

Specifying Loading and Constraints

Complete generality in the definition of loading conditions is achieved by allowing you to independently specify loads, enforced motions, and/or temperatures on each of the N segments, even though the fundamental region is the only segment explicitly represented in the analysis. Output requests are handled with the same degree of generality in that solutions are provided for each of the N segments.

If your loading has the same symmetry as the structure, only a single analysis is required for a symmetrical subregion of the structure with symmetrical boundary conditions. If your loading is unsymmetrical, the following additional tasks must be performed:

1. Separate applied loads into symmetry sets (e.g., symmetrical and antisymmetrical sets for a structure with one plane of symmetry).
2. Apply separate constraint sets to a subregion of the structure in order to model the boundary conditions corresponding to each symmetry condition.
3. Compute separate solutions for each symmetry condition.
4. Combine the results for each symmetry condition to obtain the complete solution for the complete structure.

NX Nastran also includes a way of handling the special case of unsymmetrical loads on a structure with one or two planes of symmetry. With this method:

- Tasks 1 and 2 (listed above) you perform by hand by hand.
- Task 3 is accomplished by creating subcases for each symmetry condition, with different SPC sets and loading conditions for each subcase.
- Task 4 is accomplished with SYMCOM and SYMSEQ Case Control commands.

In a cyclic symmetry analysis, you apply loads to the model with the following bulk data entries:

- LOADCYH (harmonic load)
- LOADCYN (physical segment load)
- LOADCYT (AXI option) Bulk Data entry

You only need to define a single subcase is needed for each loading condition. That subcase must contain a LOAD, TEMP(LOAD), or DEFORM command that selects a LOADCYH, LOADCYN, or

LOADCYT Bulk Data entry (see [Figure 26-9](#)). The LOADCYH and LOADCYN entries in turn reference FORCE, MOMENT, PLOAD, SLOAD, SPCD, DEFORM, and TEMP entries. The LOADCYH entry may also reference an RFORCE or GRAV entry in order to specify centrifugal or gravity loading, respectively. The LOADCYT entry is used only with the AXI option to specify loads as a function of azimuth angle.

Importantly, you can't use the SUBCOM/SUBSEQ commands to combine loads in a cyclic symmetry analysis. To combine loads, use the LOAD Bulk Data entry.

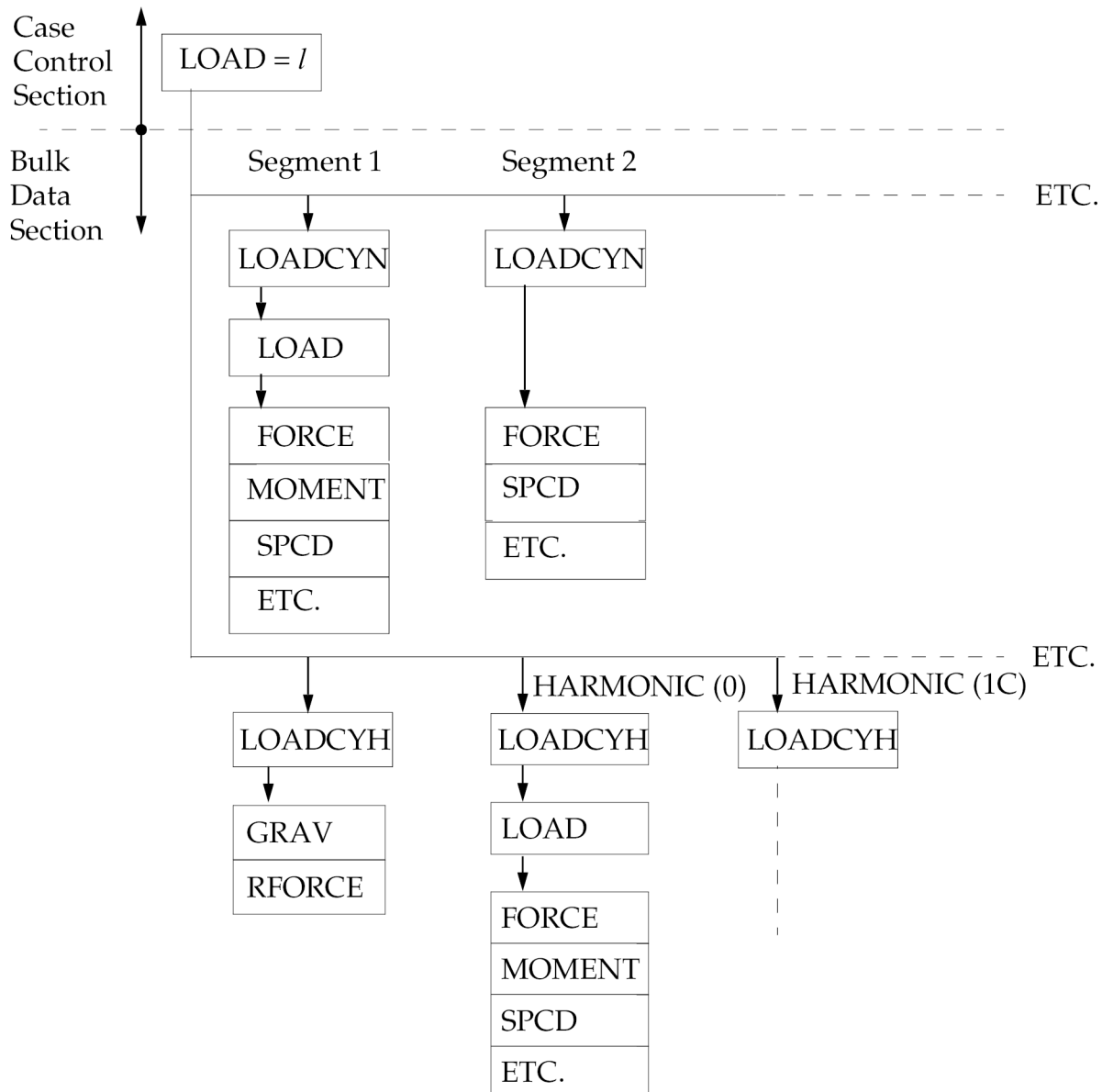


Figure 26-9. Cyclic Symmetry Flowchart

There are no restrictions on using single-point constraints, multipoint constraints, or the OMIT feature. The software automatically applies constraints between segments to the degrees-of-freedom at grid

points specified on CYJOIN Bulk Data entries which aren't otherwise constrained. You can also use the SPCD Bulk Data entry to vary the magnitude of enforced displacements for each of the segments.

In static analyses, you can use the OMIT feature to remove all degrees-of-freedom at internal grid points without losing accuracy. Since this reduction is applied to a single segment prior to the symmetry transformations, it can greatly reduce the amount of subsequent calculation. In the case of vibration analysis, you can use the OMIT feature to reduce the size of the analysis set with the typical approximations. You can't use the SUPORTi entries for the free bodies in a cyclic symmetry analysis.

Specifying Harmonics

You specify the harmonics to use in the cyclic symmetry analysis on the HARMONICS Case Control command. The HARMONICS command must be present in Case Control and it must appear above the subcase level. In buckling analysis, the program limits the static preload to the zero harmonic and the information on the HARMONICS command is used only for the eigenvalue analysis.

Bulk Data Entries Specific to Cyclic Symmetry

Table 26-4. lists bulk data entries that are unique to cyclic symmetry analyses

Table 26-4. Bulk Data Entries Unique to Cyclic Symmetry	
Bulk Data Entry	Function
CYJOIN	Defines the grid points on sides 1 and 2 of the fundamental region.
CYSYM	Specifies the number of segments in the entire model and the type of symmetry. Allowable symmetry types are ROTational, DIHedral, and AXIsymmetric.
CYAX	List of grid points that lie on the axis of symmetry.
CYSUP	List of supported components at a single grid point that are “constrained” in order to prevent rigid body motion for free bodies.
LOADCYN	Defines applied loads in terms of physical components.
LOADCYH	Defines applied loads in terms of their harmonic components.
LOADCYT	Specifies loads as a function of azimuth angle for the AXI symmetry type.

These entries are described below.

Using CYSYM

The CYSYM entry lets you define important parameters for cyclic symmetry analysis. For example, you use it to select the type (STYPE) of cyclic symmetry to perform in the analysis, such as dihedral (DIH). The format of the CYSYM entry is:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

Field

Contents

NSEG

Number of segments that comprise the total structure.

STYPE

Symmetry type ("ROT", "AXI", "DIH").

- For the ROT and AXI options, the polar angle θ subtended by the element model for the fundamental region, must be such that $\theta = 360^\circ/\text{NSEG}$ exactly. For the AXI option, the NSEG value should be large enough (typically 360 segments) to approximate true axisymmetry.
- For the DIH option, θ must equal $180^\circ/\text{NSEG}$ exactly if there are any points on side 1. In this case, the polar angle θ is smaller because the fundamental region is the right half of segment 1. If there are no points on side 1, θ must be less than or equal to $180^\circ/\text{NSEG}$ in the DIH option because otherwise the segments would overlap.
- To properly apply loads and interpret output data, you must be aware of the orientation of the coordinate system associated with the individual segments. For the ROT and AXI options, the coordinate system used to define the model of the fundamental region is rotated through the symmetry operation that "generates" each of the other identical segments. For the DIH option, the coordinate system used to define the model of the fundamental region (segment 1 R) is rotated through the necessary symmetry operations that generate the R-half of each of the identical segments. The coordinate system for the L-half of a segment is the mirror image of the coordinate system associated with the R-half of the segment in question; thus, the L-half of any segment has a left-handed coordinate system.

See also

- "CYSYM" in the *NX Nastran Quick Reference Guide*

Using CYJOIN

The boundary points of a segment in cyclic symmetry problems using the CYJOIN entry are as follows:

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

Field	Contents
SIDE	Side identification.
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems.
Gi	Grid or scalar point identification numbers.

One CYJOIN entry plus all necessary continuation entries are used to specify the GRID and/or scalar points (SPOINT), on side 1 of the fundamental region. A second CYJOIN entry plus all necessary continuation entries are used to specify the GRID and/or scalar points (SPOINT) on side 2 of the fundamental region. As will be shown, points that lie on the axis of symmetry appear on a CYAX Bulk Data entry but not on a CYJOIN entry.

- For the ROT option, the two CYJOIN lists must be of the same length and, since the boundary conditions for the fundamental region involve these two lists, it is required that the two lists be ordered. That is, the n-th point on side 1 is related through boundary conditions to the n-th point of side 2. Side 2 is related to side 1 as indicated in [Figure 26-2](#).

Note that displacement compatibility must be maintained at the boundaries. This requirement necessitates that the displacement coordinate systems for the boundary points be either cylindrical or spherical.

- The AXI option is a subset of the ROT option in which all the grid points in the fundamental region lie either on side 1 or on side 2. The rules for the CYJOIN entry are identical for both the ROT and AXI options of cyclic symmetry.
- For the DIH option, side 1 denotes the boundary between segments, and side 2 denotes the side that is contained in the plane of reflective symmetry of the segment. Since the boundary conditions do not relate sides 1 and 2, it is not necessary that the two lists be of equal length. It is, however, necessary under the DIH option to provide an entry in the third field for the parameter C. The appropriate entries are listed in [Table 26-5](#).

Unlike the ROT option (in which case, one and only one logical CYJOIN entry must exist for each of side 1 and side 2), any number of CYJOIN entries may be used to define the points on a side of the fundamental region in the DIH option. Thus, each point along a boundary may appear on a separate CYJOIN entry if, for example, each point has a different coordinate system. For the DIH option, the displacement coordinate systems associated with boundary points must be oriented so that the components of motion are normal and parallel to the boundary planes.

Table 26-5. Parameter C for the DIH Option

Type of Point	C	Description
Grid	T1	T1 component of translation motion normal to boundary.
	T2, R, or C	T2 component of translation motion normal to boundary.
	T3 or S	T3 component of translation motion normal to boundary.
SCALAR	T1	If the user-defined translation is normal to the boundary or if the user-defined rotation vector is in the plane containing the boundary.
	Blank, T2, or T3	If the user-defined translation is parallel to the boundary or if the user-defined rotation vector is normal to the boundary.

See also

- “CYJOIN” in the *NX Nastran Quick Reference Guide*

Using CYAX

The CYAX Bulk Data entry lets you list points on the axis of symmetry. The format of the CYAX entry is:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

Field**Contents**

Gi

A list of grid points on the axis of symmetry.

- The coordinate system that locates points on the axis of symmetry and the displacement coordinate system for these points must be rectangular with the z-component aligned with the axis of symmetry. In addition, if the DIH option is selected, the y-axis must be perpendicular to side 1 of the fundamental region.
- Grid points on the axis of symmetry may not appear on a CYJOIN Bulk Data entry.
- Grid points on the axis of symmetry may not be referenced as dependent points by MPCs and/or rigid elements.
- Grid points on the axis of symmetry may have SPCs applied in accordance with the following rule that is necessary to satisfy symmetry. If NSEG > 3, SPCs must be applied to both components 1 and 2 or to neither. The same rule applies to components 4 and 5.
- Grid points listed on CYAX Bulk Data entries must be in the *a*-set.

See also

- “CYAX” in the *NX Nastran Quick Reference Guide*

Using CYSUP

The CYSUP bulk data entry lets you define rigid body supports for cyclic symmetry analysis. All supported degrees-of-freedom for a rigid body must be specified at a single grid point. The format of the CYSUP entry is:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

Field	Contents
GID	Grid point identification number.
C	Component numbers.

The total structure can have, at most, six rigid body degrees of freedom. Supports that remove only the rigid body motions are applied to the $k = 0$ and $k = 1$ harmonics of the components of motion listed on this entry. These components may not be constrained by SPCs, may not be referenced as dependent degrees of freedom by MPCs and/or rigid elements, and may not be referenced on OMIT or OMIT1 Bulk Data entries.

Rigid body motion is possible only for harmonics $k = 0$ and $k = 1$. Translation along and rotation about the axis of symmetry are the only possible rigid body motions for $k = 0$, while translations normal to the axis of symmetry and rotations about these translational directions are the only possible rigid body motions for $k = 1$. The *NX Nastran Quick Reference Guide* lists special rules relative to the proper support of rigid body motion for CYSUP.

See also

- “CYSUP” in the *NX Nastran Quick Reference Guide*

Using LOADCYN

The LOADCYN entry lets you identify the segments on which referenced loads, defined in terms of physical components, are to be applied. The format of the LOADCYN entry is:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYPE	S1	L1	S2	L2	

Field	Contents
SID	Load set identification number.
S	Scale factor.
SEGID	Segment identification number.
SEGTYPE	Segment type.
Si	Scale factors.
Li	Load set ID numbers.

The loads are referenced by listing the identification numbers of one or two LOAD Bulk Data entries. Scale factors on the LOADCYN Bulk Data entry are used to form a linear combination of the load sets referenced on the LOAD Bulk Data entries. Alternatively, the LOADCYN Bulk Data entries may directly reference load sets comprised of the types of static load definition entries shown in [Table 26-6](#).

Table 26-6. Static Load Definitions for Cyclic Symmetry			
Point Loads	Distributed Loads	Temperature Loads	Enforced Displacement
FORCE	PLOAD1	TEMP	SPCD
FORCE1	PLOAD2	TEMPP1	
FORCE2	PLOAD4	TEMPRB	
MOMENT		TEMD	
MOMENT1			
MOMENT2			
SLOAD			

The loads listed on these entries are applied at the images in the segment referenced in field 4 of the LOADCYN entry of the grid points and elements listed on these entries. The coordinate systems for image points are rotated and/or reflected from the coordinate system in the fundamental region.

Note that GRAV and RFORCE Bulk Data entries are not included in the above list. These static load entries are handled automatically in a very straightforward fashion through the LOADCYH Bulk Data entry.

See also

- “[LOADCYN](#)” in the *NX Nastran Quick Reference Guide*

Using [LOADCYH](#)

The LOADCYH entry provides for the identification of referenced loads that are defined in terms of harmonic components. The format of the LOADCYH entry is as follows:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

Field	Contents
SID	Load set identification number.
S	Scale factor.
HID	Harmonic component.
HTYPE	Harmonic type.
Si	Scale factor on Li.
Li	Load set identification number.

This option is particularly useful for the definition of gravity (GRAV) and centrifugal loads (RFORCE) because you only need to specify GRAV or RFORCE under the HTYPE field on the entry and the correct harmonic components of the specified load are automatically generated. The component of gravity parallel to the axis of symmetry produces harmonic loads of the order $k = 0$, and components of gravity perpendicular to the axis of symmetry produce loads of the order $k = 1$. Components of centrifugal force produce harmonic loads of the order $k = 0$ if the spin axis coincides with the axis of symmetry, and of the order $k = 0$, $k = 1$, and $k = 2$ if it does not.

See also

- “LOADCYH” in the *NX Nastran Quick Reference Guide*

Using LOADCYT

You use the LOADCYT Bulk Data entry with the AXI option to specify loads in terms of physical components as a tabular function of the azimuth angle (measured in degrees). The format of the LOADCYT entry is:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TABLEID1	LOADSET1	METHOD1	TABLEID2	LOADSET2	METHOD2		

Field

Contents

SID	Load set identification number.
TABLEID _i	Table ID for table load input for load set Li.
LOADSET _i	Load set Li.
METHOD _i	Method of interpolation.
0	interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics.
1	interpolate the magnitude of the load at corresponding grid points in all segments.

This capability is implemented by referencing a set of load definition entries under the heading LOADSET on the LOADCYT entry.

These entries define a magnitude for the load. The azimuthal variation of the applied load is specified on the TABLED_i entry referenced under the heading TABLEID. The net applied load is the product of the magnitude given on the load definition entry times the appropriate value on the TABLED_i entry. The heading METHOD_i on the LOADCYT entry is utilized to specify the type of interpolation to be used in assigning loads to degrees of freedom.

See also

- “LOADCYT” in the *NX Nastran Quick Reference Guide*

Case Control Commands Specific to Cyclic Symmetry

Table 26-7 lists Case Control commands that are unique to cyclic symmetry analyses.

Table 26-7. Case Control Commands Unique to Cyclic Symmetry	
Command	Function
HARMONICS	Specifies harmonics for which the computation is desired.
DSYM	Provides for either one or two planes of overall symmetry under the DIH option.
NOUTPUT	Controls requests for the output of physical components.
HOUTPUT	Controls requests for the output of harmonic components.

With cyclic symmetry, note that:

- References to SPC and/or MPC sets in the Case Control Section must appear above the subcase level, i.e., before the first subcase.
- You must have one subcase per loading condition. Loads, enforced deformations, and enforced displacements are requested in the standard manner, i.e., through LOAD, TEMP(LOAD), DEFORM and SPC set selections. As described above, loads can be input in terms of physical components and/or harmonic components. With the AXI option, you can also specify loads in terms of physical components that are tabulated as a function of the azimuth angle (in degrees).

Using HARMONICS Case Control Command

A HARMONICS command must appear above the subcase level (i.e., before the commands that describe the first subcase) to specify the harmonics for which solutions are to be computed. The HARMONIC command cannot reference a set definition that specifies “ALL”. If you need to use all the harmonics, specify:

```
HARMONICS = ALL
```

See also

- “HARMONICS” in the *NX Nastran Quick Reference Guide*

Requesting Output

The NOUTPUT and HOUTPUT Case Control commands let you request output from a cyclic symmetry analysis.

- You can use the NOUTPUT command to request physical output within a subcase for all output quantities, such as forces and displacements, requested in the subcase.
- You can use the HOUTPUT command to request harmonic output within a subcase for all output quantities requested in the subcase.

These commands let you specify the segments for which you want output and, with HOUTPUT, the cyclic components for which output is desired.

In the case of thermal or deformation loading, you can only request element force and stress output that's consistent with the form of the loading. In other words, you can only request physical output if you defined the loads with a LOADCYN Bulk Data entry and harmonic output if you defined the loads with a LOADCYH Bulk Data entry.

Using a single subcase for each loading condition introduces the need to refer to “coded” subcases for SORT2 output and plotting purposes. Since a single subcase now refers to NSEG segments (in the case of ROT and AXI options) and two times NSEG segments (including Right and Left halves, in the case of DIH option), you need a unique coded subcase ID to refer to a particular segment of the structure for plotting purposes in PLOT commands wherever a subcase ID is to be supplied. This coded ID is obtained as follows:

Coded Subcase ID for a Particular Segment

= SUBCASE ID · 10000 + SEGMENT ID · 10 + zero (for ROT or AXI option)

$$= \text{SUBCASE ID} \cdot (10000) + \text{SEGMENT ID} \cdot 10 + \begin{cases} 1 & \text{(if R segment)} \\ 2 & \text{(if L segment)} \end{cases}$$

See also

- “**HOUTPUT**” in the *NX Nastran Quick Reference Guide*
- “**NOUTPUT**” in the *NX Nastran Quick Reference Guide*

Coded Subcase ID for a Particular Harmonic

$$= \left[\text{SUBCASE ID} \cdot 10000 + \text{HARMONIC ID} \cdot 10 + \begin{cases} 1 & \text{(if component = C)} \\ 2 & \text{(if component = S)} \\ 3 & \text{(if component = C*)} \\ 4 & \text{(if component = S*)} \end{cases} \right]$$

Note:

- If a subcase contains TEMP(LOAD) and/or DEFORM commands, irrespective of the presence of a LOAD command, element force and stress output are only correct in terms of physical components if the TEMP(LOAD) and the DEFORM loads are defined in terms of physical components. Similarly, if the TEMP(LOAD) and DEFORM data are provided in terms of harmonic components, element force and stress output are only correct in terms of harmonic components.
- If, under the DIH option of cyclic symmetry, a DSYM command is explicitly present under a subcase, the output is, by default, limited to the segments in the first half plane if there is one plane of overall symmetry, and to the segments in the first quadrant if there are two planes of overall symmetry. This default applies irrespective of the segments listed on NOUTPUT commands.

You can request undeformed and deformed plots of the fundamental region.

Typical Case Control Section Examples

To illustrate some of the rules described above, typical Case Control sections for the cylinder (Figure 26-10) and square plate (Figure 26-11) problems are shown in Listing 26-2 and Listing 26-3, respectively.

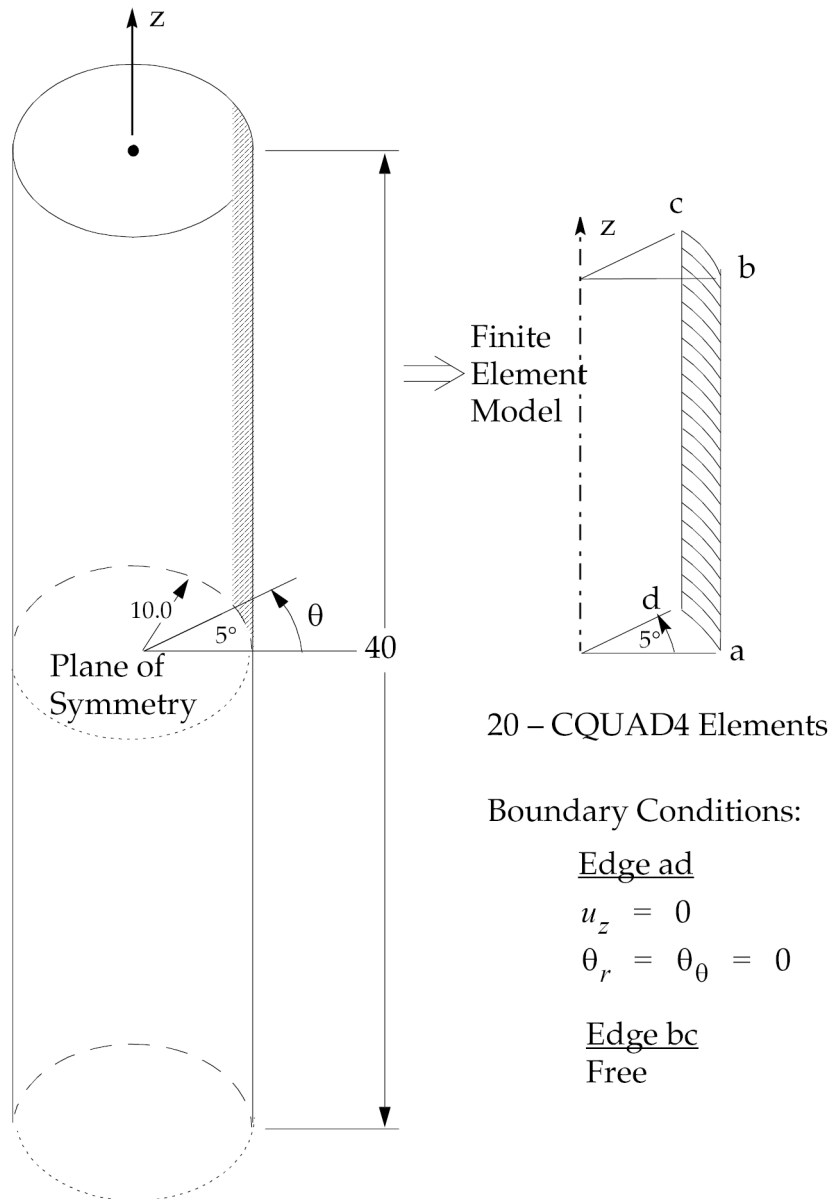


Figure 26-10. Cylinder Example

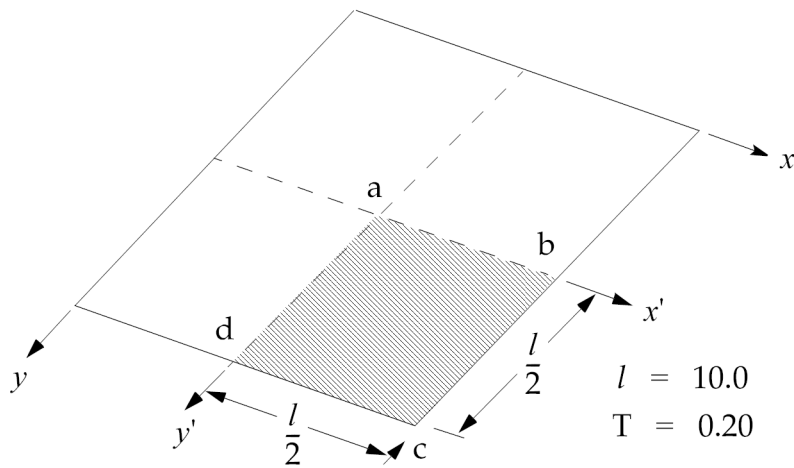


Figure 26-11. Square Plate Example

```

TITLE=CYCLIC SYMMETRY-STATICS
SUBTITLE=FIVE DEGREE SEGMENT OF CYLINDER
LABEL=AXISYMMETRIC LOADING
SPC=1
SET 1=0
SET 2=1
HARMONICS=1
NOUTPUT=2
DISP=ALL
STRESS=ALL
FORCE=ALL
SUBCASE 1
  LABEL=BAND LOADING
  LOAD=1
SUBCASE 2
  LABEL=CENTRIFUGAL LOADING
  LOAD=2
BEGIN BULK

```

Listing 26-1. ROT or AXI Symmetry Type

```

TITLE=CYCLIC SYMMETRY-STATICS
SUBTITLE=SIMPLY-SUPPORTED SQUARE PLATE
SPC=1
HARMONICS=ALL
DISPLACEMENT=ALL
NOUTPUT=ALL
DSYM=SS
SUBCASE 1
  LABEL=LOAD AT CENTER OF PLATE
  LOAD=1
SUBCASE 2
  LABEL=UNIFORM PRESSURE LOAD
  LOAD=2
SUBCASE 3
  LABEL=TEMPERATURE LOAD
  TEMP (LOAD)=3
SUBCASE 4
  LABEL=INPLANE GRAVITY LOAD
  LOAD=4
  DSYM=SA
BEGIN BULK

```

Listing 26-2. DIH Symmetry Type

The following comments apply to the Case Control sections above:

- In **Listing 26-1**, the HARMONICS command specifies that solutions are to be computed only for the zero harmonic. If the applied loads are axisymmetric, no higher harmonics need to be considered. For **Listing 26-2**, HARMONICS = ALL specifies that solutions are to be computed for all meaningful harmonics.
- A DSYM command appears in all four subcases of **Listing 26-2**. The first three subcases specify that the boundary conditions for both side 1 and side 2 are to be symmetric. This specification has two effects: 1) loads need only to be specified for the first quadrant of the structure (the fundamental region in this case), and 2) output is only processed for the segments in the first quadrant of the model irrespective of the analyst's requests for output in other segments. The particular lateral gravity condition considered in the fourth subcase is symmetric with respect to side 1 of the fundamental region and antisymmetric with respect to a plane perpendicular to side 1 of the fundamental region. It must be emphasized that the appropriate boundary conditions for each of the symmetry conditions involved in this example are internally applied by the program.
- Note that the specifications of loading conditions in the Case Control Section conform to the procedural rules for static analysis in NX Nastran.
- The NOUTPUT Case Control command controls output requests for selected segments. For **Listing 26-1**, output requests are limited to the first segment, i.e., the fundamental region. For **Listing 26-2**, output requests for all four subcases are limited to segments in the first quadrant because of the presence of the DSYM commands, even though output for ALL right-handed segments is requested. The specification of requests for specific output quantities in the Case Control Section conform with the procedural rules for static analysis in NX Nastran.

26.5 Performing Axisymmetric Analysis

You can use NX Nastran's cyclic symmetry capabilities to solve axisymmetric problems without defining any special axisymmetric elements.

Solving Axisymmetric Problems

Use the following procedure to solve static load problems:

1. Select the AXI symmetry option on the CYSYM Bulk Data entry.
2. Select a large number of segments (for example, 180) on the CYSYM Bulk Data entry.
3. Construct a model with two rows of grid points and a single row of elements in the meridional direction.
4. Define a set of harmonics on a SET Case Control command. The set of harmonics should be reasonably small for efficiency, but sufficient in size to yield reasonably accurate results.
5. Select the defined set of harmonics on the HARMONICS Case Control command. Convergence of the harmonic output towards zero for the higher harmonics will verify that the number of harmonics specified on the HARMONICS Case Control command and used in the analysis is sufficient.
6. Select a static loading condition with the LOAD, TEMP(LOAD), and DEFORM Case Control commands which reference LOADCYN, LOADCYH, or LOADCYT Bulk Data entries.
7. Place static loads only on Side 1 or on elements. The boundary conditions will take care of loading on Side2.
8. Prepare the referenced TABLED1 entries if LOADCYT entries are used to specify loads as a function of azimuth angle. The number of x-y pairs in a given table should be sufficient to produce reasonably accurate results when linearly interpolated.
9. Select the physical output for a reasonably small number of segments with the NOUTPUT Case Control command or ALL of the harmonic output with the HOUTPUT Case Control command.

Use the following procedure to obtain the vibration modes of an axisymmetric structure:

1. Select the AXI symmetry option on the CYSYM Bulk Data entry.
2. Select a large number of segments (for example, 90) on the CYSYM Bulk Data entry.
3. Construct a model with two rows of grid points and a single row of elements in the meridional direction.
4. Define a set of harmonics on a SET Case Control command where the number of harmonics is reasonably small for efficiency, but sufficient to cover the frequency range of interest.
5. Select the defined set of harmonics on the HARMONICS Case Control command.
6. Select all the harmonic output with the HOUTPUT Case Control command. This will give the eigenvectors, in a concise form, which may be interpreted by the formula

$$U(\phi) = U^{kc} \cos k\phi + U^{ks} \sin k\phi$$

where ϕ is the azimuth angle measured from Side 1 of the fundamental region and K is the harmonic index.

Axisymmetric Example

The following cyclic symmetry example problem uses the same cylinder model as “[Example 5 – Buckling of a Cylinder Under Uniform Axial Load.](#)”

A five degrees strip, as shown in [Figure 26-12](#), is modeled using the axisymmetric features.

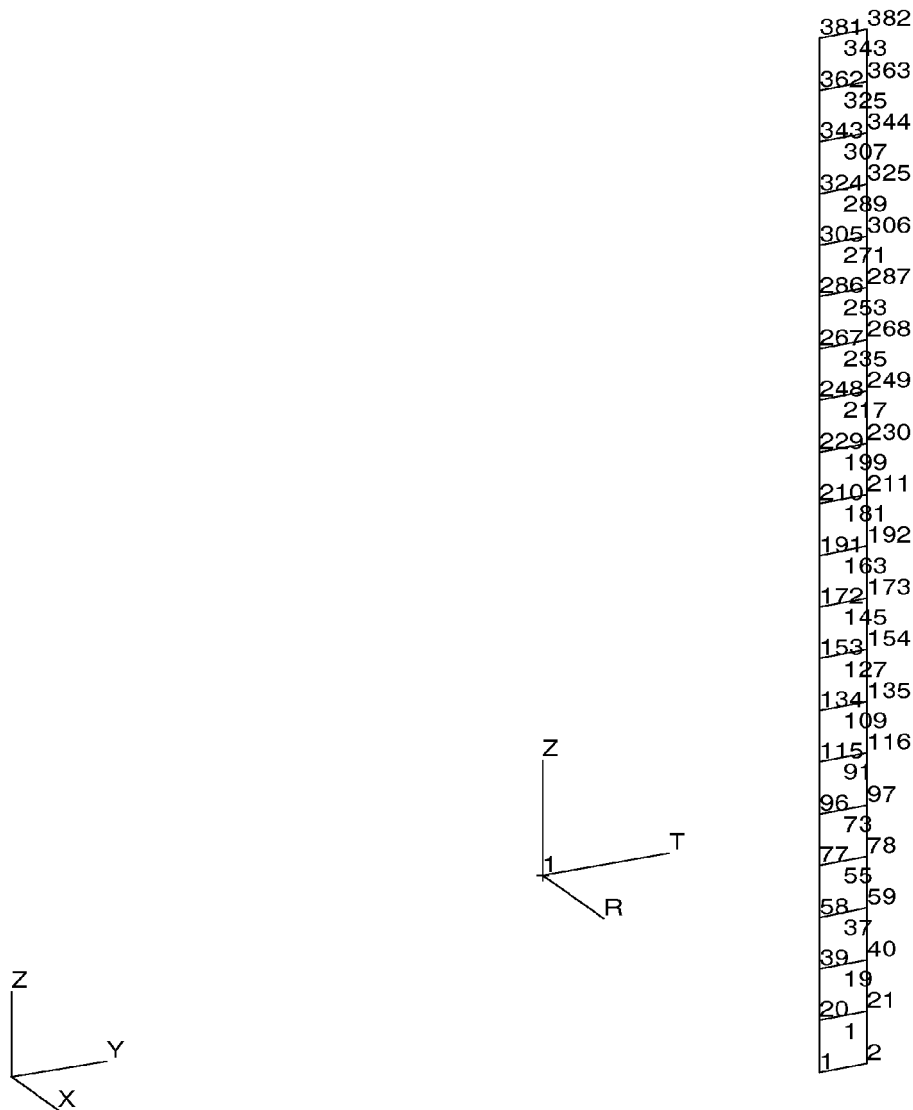


Figure 26-12. A Five Degrees Strip of a Cylinder Using Axisymmetry

A copy of the input file is shown in [Figure 26-8](#). Since only a five degrees strip is modeled, seventy-two segments are required to generate the full cylinder model internally. The CYSYM entry accomplishes this purpose. Furthermore, the boundary points for segment one are defined on the CYJOIN entries—one each for side one and side two. Note that with axisymmetry, no interior points are allowed. The applied load is a uniform compressive load applied at the top of the cylinder, using the LOADCYT/TABLED1/FORCE combinations. The TABLED1 defines the spatial distribution around the circumference of the cylinder—which in this case is evenly distributed. Since the structure

is axisymmetric, and the load is constant around the circumference, only the zeroth harmonic is required for the analysis. The displacements at the top of the cylinder and stresses at the top and bottom of the cylinders are shown in [Figure 26-13](#). The same output using the full model is shown in [Figure 26-14](#) for comparison.

```
$ filename - cyclic1.dat
$
SOL 114
TIME 600
CEND
SUBCASE 1
  TITLE= A 5 DEG STRIP
  SET 100 = 381,382
  SET 200 = 1,343
  SET 1000 = 1
  SET 1 = 0
  HARMONICS = 1
  NOUTPUT = 1000
  LOAD = 1
  SPC = 1
  DISPLACEMENT= 100
  SPCFORCES=ALL
  STRESS= 200
BEGIN BULK
PARAM      POST      -1
PARAM
,SNORM,
20.
$ Elements and Element Properties for region : pshell.1
CYSYM,72,AXI
CYJOIN,1,T2,1,20,39,58,77,96
,115,134,153,172,191,210,229,248,
,267,286,305,324,343,362,381
$
CYJOIN,2,T2,2,21,40,59,78,97
,116,135,154,173,192,211,230,249,
,268,287,306,325,344,363,382
$
SPC1,1,123,1,2
SPC1,1,12,381,382
$
LOADCYT,1,10,11,1
TABLED1,10
,0.,1.,360.,1.,ENDT
FORCE,11,381,1,1388.889,,,-1.
$
PSHELL      1      1      .03      1      1
CQUAD4      1      1      1      2      21      20      0.      0.
CQUAD4      19      1      20      21      40      39      0.      0.
CQUAD4      37      1      39      40      59      58      0.      0.
CQUAD4      55      1      58      59      78      77      0.      0.
CQUAD4      73      1      77      78      97      96      0.      0.
CQUAD4      91      1      96      97      116      115      0.      0.
CQUAD4      109      1      115      116      135      134      0.      0.
CQUAD4      127      1      134      135      154      153      0.      0.
CQUAD4      145      1      153      154      173      172      0.      0.
CQUAD4      163      1      172      173      192      191      0.      0.
CQUAD4      181      1      191      192      211      210      0.      0.
CQUAD4      199      1      210      211      230      229      0.      0.
CQUAD4      217      1      229      230      249      248      0.      0.
CQUAD4      235      1      248      249      268      267      0.      0.
CQUAD4      253      1      267      268      287      286      0.      0.
CQUAD4      271      1      286      287      306      305      0.      0.
CQUAD4      289      1      305      306      325      324      0.      0.
CQUAD4      307      1      324      325      344      343      0.      0.
```

CQUAD4	325	1	343	344	363	362	0.	0.
CQUAD4	343	1	362	363	382	381	0.	0.
\$								

Listing 26-3. Input File for Five Degrees Strip of Cylinder (Continued)

```

MAT1      1      1.+7      3.84+6      .3      0.      0.      0.      0.
$ Nodes of Group : group5
GRID      1      1      10.      0.      0.1
GRID      2      1      10.      5.      0.1
GRID     20      1      10.      0.      1.1
GRID     21      1      10.      5.      1.1
GRID     39      1      10.      0.      2.1
GRID     40      1      10.      5.      2.1
GRID     58      1      10.      0.      3.1
GRID     59      1      10.      5.      3.1
GRID     77      1      10.      0.      4.1
GRID     78      1      10.      5.      4.1
GRID     96      1      10.      0.      5.1
GRID     97      1      10.      5.      5.1
GRID    115      1      10.      0.      6.1
GRID    116      1      10.      5.      6.1
GRID    134      1      10.      0.      7.1
GRID    135      1      10.      5.      7.1
GRID    153      1      10.      0.      8.1
GRID    154      1      10.      5.      8.1
GRID    172      1      10.      0.      9.1
GRID    173      1      10.      5.      9.1
GRID    191      1      10.      0.     10.1
GRID    192      1      10.      5.     10.1
GRID    210      1      10.      0.     11.1
GRID    211      1      10.      5.     11.1
GRID    229      1      10.      0.     12.1
GRID    230      1      10.      5.     12.1
GRID    248      1      10.      0.     13.1
GRID    249      1      10.      5.     13.1
GRID    267      1      10.      0.     14.1
GRID    268      1      10.      5.     14.1
GRID    286      1      10.      0.     15.1
GRID    287      1      10.      5.     15.1
GRID    305      1      10.      0.     16.1
GRID    306      1      10.      5.     16.1
GRID    324      1      10.      0.     17.1
GRID    325      1      10.      5.     17.1
GRID    343      1      10.      0.     18.1
GRID    344      1      10.      5.     18.1
GRID    362      1      10.      0.     19.1
GRID    363      1      10.      5.     19.1
GRID    381      1      10.      0.     20.1
GRID    382      1      10.      5.     20.1
$
CORD2C     1      0.      0.      0.      0.      0.      1.
+          A
+          A 1.      0.      0.
ENDDATA

```

Listing 26-3. Input File for Five Degrees Strip of Cylinder

0	SUBCASE = 1										SEGMENT = 1		
D I S P L A C E M E N T V E C T O R													
POINT ID.		TYPE	T1	T2	T3	R1	R2	R3					
381		G	.0	.0	-1.058190E-01	2.021944E-16	-2.435790E-02	5.526086E-19					
382		G	.0	.0	-1.058190E-01	1.009687E-16	-2.435790E-02	1.583684E-17					
0												HARMO	
1	A 5 DEG STRIP					FEBRUARY	4,	2004 NX NASTRAN	1/23/04	PAGE	49		
0	SUBCASE = 1										SEGMENT = 1		
S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)													
ELEMENT	FIBRE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)								
ID.	DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES					
0 1	-1.500000E-02	-7.481326E+03	-5.592213E+04	2.894851E-10	.0000	-7.481326E+03	-5.592213E+04						
5.258216E+04	1.500000E-02	-5.769141E+03	-5.021485E+04	2.903178E-10	.0000	-5.769141E+03	-5.021485E+04						
4.759325E+04	0 343	-1.500000E-02	-7.481326E+03	-5.592213E+04	-5.481948E-10	.0000	-7.481326E+03	-5.592213E+04					
5.258216E+04	1.500000E-02	-5.769141E+03	-5.021485E+04	-5.431988E-10	.0000	-5.769141E+03	-5.021485E+04						
4.759325E+04													

Figure 26-13. Abridged Output Using Axisymmetry

D I S P L A C E M E N T V E C T O R									
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
381	G	.0	.0	-1.058189E-01	.0	-2.433120E-02	3.729655E-16		
382	G	.0	.0	-1.058189E-01	.0	-2.442414E-02	-5.255163E-16		
S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)									
ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			VON	
		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR		
MISES									
0 1	-1.500000E-02	-7.482788E+03	-5.592399E+04	-1.636876E-10	.0000	-7.482788E+03	-5.592399E+04		
5.258343E+04	1.500000E-02	-5.769486E+03	-5.021298E+04	-1.697938E-10	.0000	-5.769486E+03	-5.021298E+04		
4.759125E+04	0 343	-1.500000E-02	-7.482788E+03	-5.592399E+04	-4.949596E-10	.0000	-7.482788E+03	-5.592399E+04	
5.258343E+04	1.500000E-02	-5.769486E+03	-5.021298E+04	-2.932691E-10	.0000	-5.769486E+03	-5.021298E+04		
4.759125E+04									

Figure 26-14. Abridged Output Using Full Model

Chapter 27: p-Elements

- *Introduction to p-Elements*
- *p-Version Capabilities*
- *p-Element Loads and Constraints*
- *Line and Pressure Loads in p-Element Analysis*
- *p-Element Example Problems*
- *Square Plate with a Circular Hole*
- *Spherical Structure Under Internal Pressure*
- *Motor Blades — Normal Modes*
- *Torsion of Elliptical Bar*
- *Circular Solid Model*
- *Clamped Rhombic Plate*
- *Helix Modeled with CBEAM p-Elements*
- *Plate with Stiffener*
- *Solid p-Elements in Linear Dynamic Solution Sequences*
- *p-Element Thermal Analysis*

27.1 Introduction to p-Elements

One of the main applications of p-version elements is detailed stress analysis. The p-elements have higher-order polynomials, which provide better representation of complex stress fields. For these complex stress fields, the geometry, loads, and boundary conditions must be represented accurately. This includes modeling fillets instead of sharp corners, distributed loads and constraints instead of point loads and constraints, etc. A more detailed model leads to more detailed results.

p-Elements have several modeling advantages. With the higher-order polynomials, the geometry and loads can be represented more accurately. Generally, fewer p-version elements with curvature than h-version elements with straight edges are required.

The accuracy of the analysis is controlled primarily by the polynomial level, not by the element size. Generally the user needs to only use the minimum number of elements necessary to adequately model the geometry, independent of expected result characteristics. Polynomials levels can then be assigned based on areas of the model in which the user is interested, and areas of the model in which the user is not. (Of course the same polynomial level can be assigned everywhere for a minimum of user involvement, but this is not as efficient.) If a more accurate answer is necessary, the polynomial levels can be increased, either manually by the user or automatically by the program, until the answers reach the specified accuracy, without changing the mesh. Adding degrees-of-freedom until the error decreases to a specified level is known as adaptivity. By adding higher-order polynomials instead of refining the mesh, a faster rate of convergence, and therefore fewer iterations, can be achieved.

p-Elements in NX Nastran

P-elements allow different polynomial orders in the three coordinate directions, and use bubble functions at $p=1$ and optimum integration algorithms at $p \geq 2$ to achieve better accuracy. The elements will also pass the patch tests and rigid body motion at any allowable p-level. These additional features improve the accuracy over conventional p-elements.

Compatibility with elements of different p-orders, and with the existing solid elements, is automatically enforced. The user does not have to write constraint equations or use special elements to provide displacement continuity. Loads and boundary conditions can be applied to the elements themselves, or to the geometry on which they are defined.

The definition of geometry has been added to NX Nastran. This geometry can be defined with midside points, as in the existing elements; with rational parametric cubics or equations; or even with any external geometry. A facility has been set up to invoke remote procedure calls to any independent program and database that provides a toolkit for geometry queries, which would allow proprietary geometries of other pre-processors or CAD packages to be used. Associativity has been added so that the elements are defined on the geometry, and the element geometry calculated accordingly.

The error estimator requires only a single analysis, not the difference between two analyses. For linear elements, it is based on the grid-point stress discontinuity; for higher polynomial orders, it considers the contribution of the additional terms for the individual edge, face, and body functions. Therefore it provides efficient information because of the directional sensitivity within the elements.

The adaptivity method is very flexible, working on an element-by-element basis. Using the error estimator, the polynomial level within each element can be varied independently in the three directions. At the most basic level, the edges are varied, since they determine the face and body functions. Different error criteria may be used in different regions, allowing areas of primary interest to have lower error tolerances. Multiple subcases may be used.

Combining p- and h-Elements

The main advantage of implementing p-elements in NX Nastran is the ability to combine them with the existing h-elements for global/local analysis. The mesh could consist mostly of h-version elements, with p-version elements in the areas of interest. This method of global/local interface is the most accurate, because it provides both stiffness and loads information, rather than transferring just boundary displacements or tractions into the local model.

h-Version elements still have several advantages. They are better for global behavior, such as loads analysis, where the exact geometry is not used. They are better for strongly singular problems, such as nonlinear problems. Approximations, such as sharp corners, point loads, point constraints, MPCs, etc. may be used. For a given iteration, they tend to have shorter solution times and use less disk space. Finally, h-elements are a very mature technology.

Putting both approaches in the same program combines the advantages for general problems. The p-elements may be connected directly to the existing h-elements, and continuity is automatically enforced. This provides global/local analysis in a single run.

Most general problems have both some global and some local aspects; i.e., regions where h-elements and p-elements, respectively, are appropriate. Therefore the best approach is combined h- and p-adaptivity, with different criteria in different parts of the model.

27.2 p-Version Capabilities

Following is a limited description of the Bulk Data entries and Case Control commands for the p-version elements. [Table 27-2](#) contains a comparison with the conventional h-version elements. A full description of the entries may be found in the *NX Nastran Quick Reference Guide*.

Solution Sequences Supported

Solution Sequences 101, 103, 107, 108, 109, 110, 111, 112, and 101 (Heat Transfer).

Geometry

With the h-version elements, geometry is determined by the addition of midside grid points to the element definition. With the p-version elements, you can define the edges and faces of elements to lie on curves and surfaces, which can span an arbitrary number of elements. You don't have to define midside grid points.

The Bulk Data entries to define the associativity between the finite element model and the geometrical model are the FEEDGE and FEFACE entries for the edges and faces, respectively. These entries reference the grids that make up the edges and faces, not the elements themselves, and therefore do not have to be repeated for each element that shares the edge or face. The old method of midside grid points is still available for the FEEDGE entry, which would reference one or two GRIDs or POINTs. The POINT entry is very similar to the GRID entry, except that it defines a location in space with no associated degrees-of-freedom.

The more useful way of defining the FEEDGES and FEFACES is to have them reference geometrical curves and surfaces, defined by the GMCURV and GMSURF entries. GMCURVs and GMSURFs are functions of one or two parametric variables, respectively, and can be defined as rational parametric cubics or general equations. For instance, a hole could be defined with the parametric equations for a cylinder, which all the element faces on the hole would then reference. The GMCURVs and

GMSURFs could also reference external geometry from a CAD or other database, using remote procedure calls with an external programming interface. When geometry has been specified, it is approximated as cubics for the element-level computations. Using cubics ensures that the elements pass the rigid-body and patch tests.

The two different methods of defining the geometry are shown in **Figure 27-1**.

Convective coordinate systems may be defined on the FEEDGEs, FEFACEs, GMCURVs, and GMSURFs. The directions of the systems are normal and tangent to the entities at every point. These coordinate systems can then be used for material properties, loads, boundary conditions, etc.

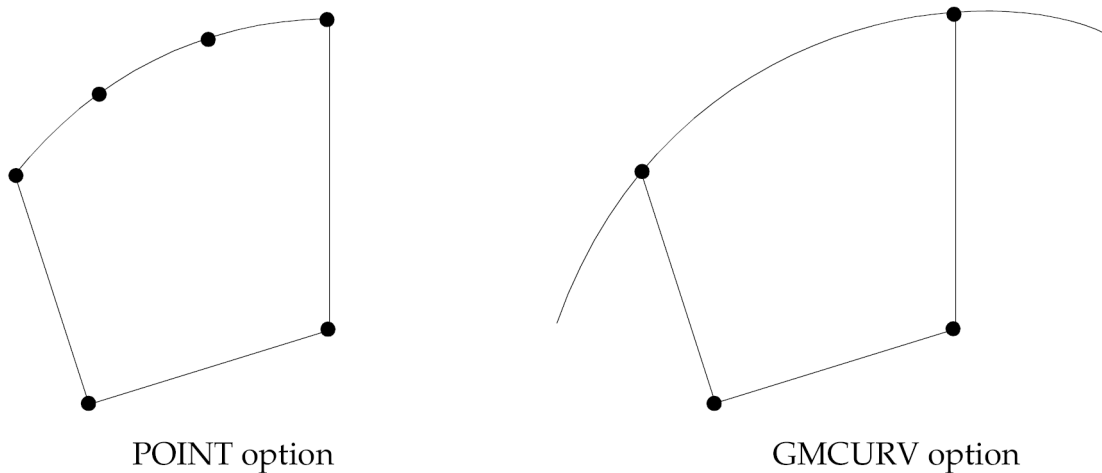


Figure 27-1. Methods of Defining Element Geometry

Elements

Only the CHEXA, CPENTA, CTETRA, CQUAD, CTRIA, and CBEAM elements may be defined as p-elements. Midside grid points may not be specified on p-version elements; curvature is defined by the FEEDGE or FEFACE entries, as previously described.

The standard Bulk Data entries, PSOLID, PSHELL, and PBEAM are used to define the element properties. The p-version properties are defined in the PVAL and ADAPT entries, which are described later.

The p-version elements may be joined to existing h-version elements with a few considerations. FEEDGEs and FEFACEs may not be defined on the interface, since the h-elements can not accept them. Midside grid points may not be defined on the interface, since the p-elements can not accept them. The polynomial levels of the p-elements will be lowered automatically so that displacement continuity is enforced.

Materials

The standard isotropic, orthotropic, and anisotropic materials are available for p-elements. Orthotropic and anisotropic material properties may be defined in a curvilinear material system.

27.3 Defining p-Element Loads

For p-version elements in detailed stress analysis, it is important to use distributed loads. Concentrated forces cause singularities in the stress field, and therefore should be avoided.

27.4 Line and Pressure Loads in p-Element Analysis

The GMLOAD Bulk Data entry has been defined for the p-version elements. You can use the GMLOAD entry to specify line and pressure loads by:

- Applying directly to the finite elements entities, FEEDGES and FEFACEs.
- Applying to the geometry entities GMCURV and GMSURF.

The load can be distributed as constant, linear, quadratic, cubic, equation, or table, and can be given in any coordinate system.

Thermal Loads

For thermal loads, the TEMPF entry allows specification of an equation or table to describe the variation of the temperature field over the elements. The TEMP and TEMPD entries may also be used, but will result in linear interpolation of the temperature from the grids. The TEMPP1 entry defines a constant temperature and temperature gradient distribution in a shell element. The TEMPRB entry defines a linear temperature and temperature gradient distribution in a beam element.

Gravity Loads

You can use the GRAV entry to define gravity loads for p-elements.

Pressure Loads

You can use the PLOAD4 entry to define a pressure load on a p-element.

Point Loads

You can use the FORCE entry to define point loads applied to grid points. No point loads can be applied to finite element edges or faces. You should only use FORCE where the adjacent elements have a fixed p-level and the results aren't of interest.

27.5 Defining p-Element Constraints

As with the loads, it is important to use distributed boundary conditions for p-version elements in detailed stress analysis. Concentrated boundary conditions also cause singularities in the stress field and should be avoided.

The GMSPC and GMBC Bulk Data entries have been defined for the p-version elements. These can apply distributed boundary conditions to GRIDs, FEEDGES, FEFACEs, GMCURVs, or GMSURFs. Note that they should be applied to GRIDs only to prevent rigid-body motion. The imposed boundary conditions can be distributed as constant, linear, quadratic, cubic, equation, or table, and can be

given in any coordinate system. The current SPC and SPC1 entries can be used, but should only be used to prevent rigid-body motion or used where the adjacent elements have a fixed p-level and the results are not of interest.

Boundary Conditions

The SPC and SPCD entries are used for the point constraints which are allowed only on corner GRID points.

The GMSPC entry is used to define zero constraints for FEEDGEs, FEFACEs, GMCURVs, and GMSURFs.

For non-zero constraints, the GMBC entry is used.

Resolution of Conflicting Zero Constraints

In general, the hierarchy set to resolve multiple input data (constraints and enforced displacements) for GRID, FEEDGE, FEFACE, GMCURV, and GMSURF entities is:

1. GRIDs
2. FEEDGEs
3. GMCURVs
4. FEFACEs
5. GMSURFs

The rules for nonzero boundary constraints are:

- Displacement values specified for each component of a given GMSURF entry are applied to the same component of all GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the GMSURF.
- Displacement values specified for each component of different GMSURF entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom which are shared by (common to) the multiple GMSURFs.
- Displacement values specified for a given FEFACE entry are applied to all GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the FEFACE. This data overrides the data that is specified for all the components of the given GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the FEFACE by using GMSURF entries.
- Displacement values specified for each component of different FEFACE entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom which are shared by (common to) the multiple FEFACEs. This data overrides the data that is specified for all the components of the given FEEDGE and GRIDs by using GMSURF entries.
- Displacement values specified for each component of a given GMCURV entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the GMCURV. This data overrides the data for all components that is specified for the given FEEDGE and GRIDs by using GMSURF and FEFACE entries.

- Displacement values specified for each component of different CURV entries are averaged and applied to the same component of all GRID degrees-of-freedom which are shared by (common to) the multiple GMCURVs. This data overrides the data for all the components that is specified for the given FEEDGE and GRIDS by using GMSURF and FEFACE entries.
- Displacement values specified for each component of a given FEEDGE entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the FEEDGE. This data overrides the data for all components that is specified for the given FEEDGE by using GMCURV or FEFACE or GMSURF entries.
- Displacement values specified for each component of different FEEDGE entries are averaged and applied to the same component of all GRID degrees-of-freedom which are shared by (common to) the multiple FEEDGES. This data overrides the data for all the components that is specified for the given FEEDGE by using GMCURV or FEFACE or GMSURF entries.
- Grids have the highest priority, i.e., any value/property specified using a GRID entry overrides all other information associated with that GRID. If multiple entries are used for a given GRID, e.g., multiple SPCs, then the existing rules of NX Nastran govern (SPCs are combined, FORCE are added, SPCDs for the same component are not allowed).
- It is important to recall that these displacements are assumed to be in the global coordinate system and that the interconsistency of the output coordinate systems of the various GRIDs, FEEDGES, FEFACEs is not checked.

27.6 p-Version Adaptivity

In order for an element to be recognized as a p-version element, it must be assigned the polynomial values in the PVAL Bulk Data entry, which is referenced in the ADAPT Bulk Data entry. In order for the p-element to be adaptive, the adaptivity parameters must be assigned in the ADAPT entry.

The PVAL Bulk Data entry assigns the polynomial levels for the three directions to a single element or a set of elements, and is referenced in the ADAPT Bulk Data entry. By default, the three directions are along the element edges. If a coordinate system is specified, the polynomial level of each edge is set to a weighted combination of the coordinate directions. There may be multiple PVALs with the same ID; all of the elements do not have to use the same entry. Each p-version element must have a starting, minimum, and maximum PVAL ID; however, these do not need to be unique. Since the polynomial levels can not decrease, the starting and minimum PVAL IDs can be the same. If the starting and maximum PVAL IDs are the same, the element will have a fixed p-level.

After the p-value distribution has been defined, all of the values will be resolved to ensure displacement continuity. This includes selecting the highest p-level specified for elements with common edges, decreasing the p-level for p-elements adjacent to h-elements, and increasing the p-level on curved edges to adequately map them.

The ADAPT Bulk Data entry controls the p-adaptivity. The ADAPT Bulk Data entry is referenced with the ADAPT Case Control command, and there may only be one unique ADAPT command in the Bulk Data file. The ADAPT entry contains the IDs for the starting, minimum, and maximum p-value distributions and the maximum number of iterations. It also contains the adaptivity parameters, which may be assigned differently to different sets of elements. These sets of elements do not have to correspond with any sets used in the PVAL entries. The adaptivity type may be element-by-element p-adaptive, uniform p-adaptive, no change, or list of p-distributions. For each set in the ADAPT entry,

the error estimator method, error tolerance, and stress and strain tolerances are specified. If a p-version element is not included in one of the sets, it is not adaptive.

For a non-adaptive p-version solution, the ADAPT Bulk Data entry may specify a maximum of one iteration, or may specify an adaptivity type of no change for all the elements. A third way is to use the PSET Bulk Data entries, which have the same format as the PVAL entry, and reference them with the ADAPT Case Control command.

27.7 Data Recovery

Since the p-version analysis may have multiple iterations, the output control must have the capability of differing among the iterations. For this reason, the OUTPUT Bulk Data entry was added. For a given set of elements, the conventional displacements, stresses, and strains, as well as the element errors and polynomial values, may be printed, plotted, or punched for the given iteration. The OUTPUT entry is referenced by the DATAREC Case Control command. The conventional Case Control commands still apply to all iterations, but the OUTPUT entry has precedence.

The p-version elements also tend to be larger than h-version elements, and have higher-order distributions of displacements, stresses, and strains. Therefore results only at the grid points are not adequate, especially for most plotting packages, which use linear interpolation. For data recovery, the p-version elements are divided into view-grids and view-elements in order to better visualize the results. The OUTRCV Bulk Data entry defines the number of view-elements in each element direction and the output coordinate system for the view-grids and view-elements. The OUTRCV entry is referenced by the OUTRCV Case Control command. The coordinates of the view-grids may be output using the VUGRID Case Control command.

At the beginning of a P-element analysis, output recovery elements are generated in order to post process displacements, stresses and strains of P-elements. The output resolution does not affect the solution P order. The definition of the output recovery “display mesh” is defined by the OUTRCV bulk data card. The default interval (i.e. the number of subdivisions) is 3*3*3. For a CQUAD4 element, nine output elements are generated as well as 16 grid points. These internally generated elements and grids lds have starting and ending values that are defined via a number of system cells.

Table 27-1. System cells to control output recovery grid and element IDs

System cell number	System cell name	Default	Description
178	GNSTRT	101000000	Adaptive Grid Start ID
179	GNMAX	110000000	Adaptive Grid Maximum ID
180	VGSTRT	111001001	VU-Grid Start ID
181	VGMAX	120000000	VU-Grid Maximum ID
182	VESTRT	100001001	VU-Element Start ID
183	VEMAX	200000000	VU-Element Maximum ID

The internally generated grids start at 111,001,001 while the internally generated elements start at 100,001,001. So, for the first CQUAD4, the output grid lds range from 111001001 to 111001016 and the element lds range from 100001001 to 100001009. However, the internally generated grids and elements lds for the second element do not directly follow those of the first element. Instead they are offset by a value defined by the PARAMETERS VUELJUMP and VUGJUMP which have a default of 1000. So, for the second CQUAD4, the output grid lds range from 111002001 to 111002016 while the

element ids range from 100002001 to 100002009. The default VUELJUMP and VUGJUMP values are set to accommodate a 9*9*9 display element mesh. Unfortunately, these default values limit the number of p-elements to fewer than 9,000 (VUGJUMP is the limiting number). In order to be able to define a larger number of p-elements, the values of VUGJUMP (and perhaps VUELJUMP) need to be adjusted downward.

To maximize the number of p-elements, the value of VUGJUMP should to be set to the maximum number of internally generated grids for all element types. The number of internally generated grids is a function of the VIEW entry on the OUTRCV card and the element type. See remark (5) on the OUTRCV card.

For example, using the default setting on the OUTRCV card (3*3*3 elements), the CQUAD4 will generate 16 grids (9 elements), the CHEXA will generate 64 grids (27 elements), the CPENTA will generate 40 grids (27 elements), the CTETRA will generate 27 grids (27 elements), CTRIA3 will generate 10 grids (9 elements), and CBEAM will generate 4 grids (3 elements). So, for a pure CQUAD4 element mesh, a value of 16 for VUGJUMP will allow over 562,000 elements instead of fewer than 9,000. The default VUELJUMP (1000) will accommodate just under 100,000 elements. Adjusting VUELJUMP downward will allow more P-elements to be defined.

For post-processors that do not support vu-elements (p-element display mesh), NX Nastran supports the ability to automatically change the name to something recognizable. See PARAMs VUBEAM, VUHEXA, VUPENTA, VUQUAD4, VUTETRA, VUTRIA3. For example, PARAM,VUHEXA,CHEXA renames the display element VUHEXA to "CHEXA" in the output files.

Table 27-2. Bulk Data Entries for h-Elements and p-Elements

	h-Version Elements	p-Version Elements
Geometry		GMCURV GMSURF
Geometrical Associativity		FEEDGE FEFACE
Coordinate Systems	CORDij	CORDij GMCORD
GridsPoints	GRID	GRID POINT
Elements	CHEXA CPENTA CTETRA CQUAD CTRIA CBEAM	CHEXA CPENTA CTETRA CQUAD CTRIA CBEAM

Table 27-2. Bulk Data Entries for h-Elements and p-Elements

	h-Version Elements	p-Version Elements
Element Properties	PSOLID PSHELL PBEAM	PSOLID PSHELL PBEAM
Material Properties	MATi	MATi
Loads	PLOAD4 FORCE	GMLOAD PLOAD4 FORCE*
Thermal Loads	TEMP, TEMPD TEMPP1 TEMPRB	TEMPF TEMP, TEMPD TEMPP1 TEMPRB
Gravity Loads	GRAV	GRAV
Constraints	SPC, SPC1	GMSPC SPC, SPC1*
Boundary Conditions	SPCD	GMBC SPCD*
Polynomial Values		PVAL PSET
Adaptivity Control		ADAPT
Data Recovery		OUTPUT OUTRCV
Miscellaneous		DEQATN TABLE3D
Sets Definition		SETS DEFINITION
* Should only be used under specific conditions.		

27.8 p-Element Example Problems

This section demonstrates the p-element and p-adaptivity capabilities. Several examples are included in this section to show how p-elements are used. Most of the input files of these examples have been

annotated and are included in the Test Problem Library (TLP) directory. The detailed descriptions of the input commands and the interpretation of the results can be found in the *NX Nastran Quick Reference Guide*.

Examples

Each example contains the following parts:

Objectives	Outlines the problem objectives.
Keywords	Serves as index and show what p-element related Bulk Data entries being used.
Problem Description	Describes the geometry and data of the problem.
Theoretical Results	Shows the closed form solution, if available.
Setup	Describes what and how to use the Bulk Data entries.
Results	Compares the calculated results with theory.
References	

Square Plate with a Circular Hole

Objectives

This example computes the stress concentration factor of a square plate with a circular hole under axial loads. The objective of this problem is to validate the stress concentration factor for a square plate of 2.73 around the hole.

Problem Description

Plate size	= 10 in x 10 in
Thickness	= 0.1 in
Hole radius	= 2.0 in
Modulus of Elasticity	= 1.0E7 psi
Poisson's ratio	= 0.3

Figure 27-2 shows the quarter model that requires only two CHEXA solid p-elements.

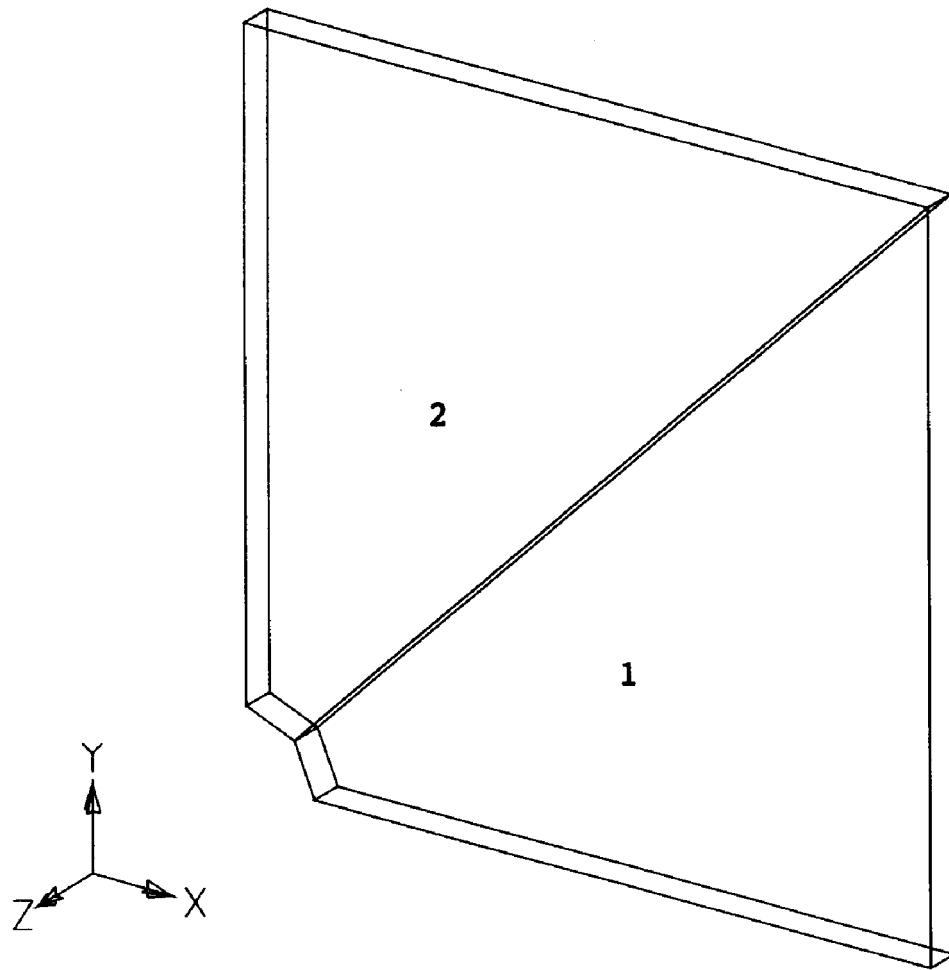


Figure 27-2. Geometry of a Square Plate with a Circular Hole

A portion of the file which has been annotated is shown in [Listing 27-1](#). The complete input file, PEV0101.dat, can be found in the TPL directory.

Theoretical Results

Stress concentration factor of 2.73 around the hole as described in Reference [1].

Setup

The Bulk Data entry FEEDGE is used to describe the circular arc of the hole. The quadratic order is used for demonstration purposes; a cubic order curve can be easily accommodated with an additional point for each curve. The load and boundary conditions are also specified using FEFACE.

Since the file has been heavily annotated, the description of the Bulk Data entries is not repeated here. Please refer either to the following partial list or the complete input file in the TPL directory for detailed descriptions.

```

TITLE = PLATE WITH HOLE -- QUARTER MODEL (p-element model; Point option)
SUBTITLE = p-adaptivity; maximum allowable p-value 5X5X3
$
$   p-elements Case Control commands
$
$   Select Bulk Data ADAPT ID for adaptivity control
$
ADAPT = 150
$
$   Request output destination(s) of locations (i.e. coordinates)
$   for p-element data recovery points
$
VUGRID(PLOT,PUNCH)=all
$
$   Request output (STRESS, STRAIN, DISP, PVAL) for p-elements only
$   DATAEC = n, where n refers to a Bulk Data OUTPUT ID
$
DATAEC = 301
$
$   Select output options for p-elements
$   OUTRCV = n, where n defines an OUTRCV ID in Bulk Data
$
OUTRCV = 401
$
$   Select set(s) of elements for adaptivity and stress output
$   SETS DEFINITION must be defined at the end of the Case Control
$   SET 501 defines a set of all elements
$   referenced by ELSETID in ADAPT entry for ERRTOL of 10%
$
SETS DEFINITION
set 501 = all
$
BEGIN BULK
$
$   p-elements Bulk Data
$
$   Adaptivity control
$   PSTRTID specifies the PVAL ID 120 for the starting pvalues p1, p2 and p3
$   for the element SET 501 (all elements)
$   PMINID specifies the PVAL ID 120 for the minimum pvalues p1, p2 and p3
$   PMAXID specifies the PVAL ID 150 for the maximum pvalues p1, p2 and p3
$
$ADAPT  SID          PSTRTID PMINID  PMAXID
$      PART    ELSETID
$
ADAPT   150          120      120    150
PART=ALL,ELSET=501
$

```

Listing 27-1. Partial Bulk Data Entries (Continued)

```

$ Specify p-levels for element SET 501 (all elements)
$ POLY1, POLY2 and POLY3 correspond to p1, p2 and p3 respectively and
$ can have different p-values
$ p-values specified by PMINID should be = PSTRTID < PMAXID
$ starting p-level p1=p2=p3=2
$ minimum p-level p1=p2=p3=2
$ maximum p-level p1=p2=5 p3=3
$
$PVAL ID POLY1 POLY2 POLY3 SETTYP ID
PVAL 120 2 2 2 SET 501
PVAL 150 5 5 3 SET 501
$
$ Select various output options;
$ Displacements, Stresses, Strains and p-values
$
$ SID
$ ELSET=n, DISP=(PRINT,PLOT,PUNCH),
$ STRESS=(PRINT,PLOT,PUNCH), STRAIN=(PRINT,PLOT,PUNCH),
$ PVAL=(PRINT,PLOT,PUNCH), BY=n
$
OUTPUT 301
ELSET=501, DISP=PRINT, STRESS=PRINT
STRAIN=PLOT, PVAL=PRINT, BY=1
$
$ Define various output options;
$ for Displacements, Stresses and Strains
$
$ SID SETID
$ CID= 0 or n (default 0; basic coordinate system)
$ VIEW=i*j*k (default 3*3*3)
$ For less output, reduce i,j and k to 2 or 1
$
OUTRCV 401 501
VIEW=2*2*2
$
CHEXA 1 1 1 25 28 4 101 125
+ 128 104
CHEXA 2 1 32 4 28 56 132 104
+ 128 156
$

```

Listing 27-1. Partial Bulk Data Entries (Continued)

```

$ The following geometry specifications demonstrate the use of
$ different options on specifying actual geometry
$ to achieve more accurate results
$
$ Define FEEDGES and FEFACES to specify
$ geometry, loading and constraints
$
$ Use POINT, FEEDGE to describe curves 104, 108, 204 and 208
$
POINT 3 1.7305341.0026220.0
POINT 30 1.0026221.7305340.0
POINT 103 1.7305341.0026221.0
POINT 130 1.0026221.7305341.0
$
$ EDGEID G1 G2 GEOMIN ID1
FEEDGE 104 4 1 POINT 3
FEEDGE 204 4 32 POINT 30
FEEDGE 108 104 101 POINT 103
FEEDGE 208 104 132 POINT 130
$
$ Specify FEFACE for BC and loads
$
$ FACEID G1 G2 G3 G4
FEFACE 1001 1 25 125 101
FEFACE 1002 25 28 128 125
FEFACE 2002 56 156 132 32
$
$ Use GMSPC, GMLOAD and FEFACE to specify BC and loads
$ on geometry instead of element data
$
$ Boundary Conditions
$
$ SID C ENTITY ID
GMSPC 999 2 FEFACE 1001
GMSPC 999 1 FEFACE 2002
$
$ LOADS
$
$ LID N1 N2 N3 ENTITY ID METHOD
GMLOAD 998 1.0 0.0 0.0 FEFACE 1002 CONSTANT
90.
$
ENDDATA

```

Listing 27-1. Partial Bulk Data Entries

Results

Table 27-3 compares NX Nastran results with the theory, and they agree very well with only two p-elements. Closer results could have been obtained with more elements or higher p-values.

Table 27-3. Comparison of NX Nastran and Theoretical Results		
Stress Concentration Factor		Percent Difference
Theory	NX Nastran	
2.73	2.77	1.5%

References

[1] Roark, R. J. and Young, W. C. *Formulas for Stress and Strain*. 5th edition. McGraw-Hill, Inc., 1975, p. 594.

Spherical Structure Under Internal Pressure

Objectives

This example computes the effects of internal pressure on a spherical structure. The deflections at the inner and the outer surfaces are compared with the theoretical results.

Problem Description

Internal pressure (P)	= 3000.0 psi
Inner radius (b)	= 5.0 in
Outer radius (a)	= 10.0 in
Modulus of Elasticity (E)	= 1.0E7 psi
Poisson ratio (ν)	= 0.3

Theoretical Results

The theoretical results on radial deflections can be found in reference 1 and are listed as follows:

$$\text{Inner radial deflection} = \frac{Pb}{E} \left[\frac{(1 - \nu)(a^3 + 2b^3)}{2(a^3 - b^3)} + \nu \right]$$

$$\text{Outer radial deflection} = \frac{Pa}{E} \frac{3(1 - \nu)b^3}{2(a^3 - b^3)}$$

Setup

This example demonstrates how the higher cubic order of the geometry for a spherical structure can be specified using the EQUATION option of the GMCURV and GMSURF Bulk Data entries. An octant model is analyzed with four CPENTA elements. The boundaries of the CPENTA elements are specified along the great circles. **Figure 27-3** shows the geometry and the element boundaries.

Partial Bulk Data entries are given in **Listing 27-2**. Equations of the inner circle along the sphere are described using Bulk Data entries GMCURV and DEQATN. The equations (IDs 1, 2, and 3) are the spatial locations of the curve along the great circle (circumference of a sphere) expressed in terms of the radius and an angle (U). The first and second derivatives of the equations are described by DEQATN entries 4 to 9. The first and second derivatives are optional inputs. If they are missing, they will be calculated by the program.

The GMCURV entry specifies the great circle geometry on the x-y plane with equations 1 through 9 for the curve ID 1, and ranges from 0 to 90 degrees (0 to 1.57 radians). Similarly, the geometry on the y-z and x-z planes, curves 2 and 3, can be specified using the same equations with different orders. Note that the labels and arguments of the equations may be chosen for user convenience.

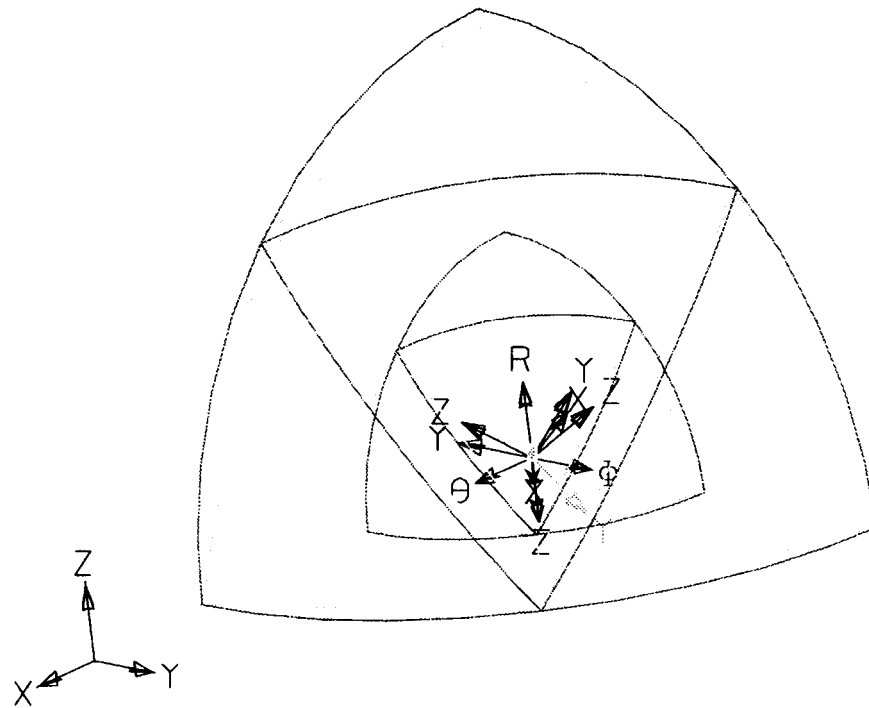


Figure 27-3. Spherical Structure

```

$
$ ***** CURVE AND SURFACE GEOMETRY *****
$
$ Inner circle equations
$
$
DEQATN 1      X(U) = 5.0*COS(U)
DEQATN 2      Y(U) = 5.0*SIN(U)
DEQATN 3      Z(U) = 0.0
DEQATN 4      XP(U) = -5.0*SIN(U)
DEQATN 5      YP(U) = 5.0*COS(U)
DEQATN 6      ZP(U) = 0.0
DEQATN 7      X2P(U) = -5.0*COS(U)
DEQATN 8      Y2P(U) = -5.0*SIN(U)
DEQATN 9      Z2P(U) = 0.0
$
$ Inner circle on x-y plane
$
GMCURV 1      MSCGRP1 0      0      +
+      EQUATION 0.0 1.570796327 +
+      1 2 3 4 5 6 7 8 9
$
$ Inner circle on y-z plane
$
GMCURV 2      MSCGRP1 0      0      +
+      EQUATION 0.0 1.570796327 +
+      3 2 1 6 5 4 9 8 7
$
$ Inner circle on x-z plane
$
GMCURV 3      MSCGRP1 0      0      +
+      EQUATION 0.0 1.570796327 +
+      2 3 1 5 6 4 8 9 7
$

```

Listing 27-2. Partial Bulk Data Entries

Geometry of the inner surface is described using **GMSURF** and **DEQATN** entries. The equations (IDs 41, 42, and 43) are the spatial locations of the surface using a spherical coordinate system expressed in terms of radius and two angles (U and V). The first and second derivatives of the equations are described by **DEQATNs** 44 to 59. Again, the first and second derivatives are optional inputs.

The **GMSURF** entry specifies the surface geometry with equations 41 through 59 for the surface ID 1, and ranges from 0 to 90 degrees in two directions. Besides describing geometry, the specification of this surface facilitates the application of internal pressure with the **GMLOAD** command. The **GMLOAD** specifies a constant pressure load of 3000 psi on the surface **GMSURF** 1. The commands are shown below.


```

$
$ Inner sphere
$
DEQATN 41      SX(U,V) = 5.0*SIN(U)*COS(V)
DEQATN 42      SY(U,V) = 5.0*SIN(U)*SIN(V)
DEQATN 43      SZ(U,V) = 5.0*COS(U)
DEQATN 44      SXU(U,V) = 5.0*COS(U)*COS(V)
DEQATN 45      SYU(U,V) = 5.0*COS(U)*SIN(V)
DEQATN 46      SZU(U,V) = -5.0*SIN(U)
DEQATN 47      SXV(U,V) = -5.0*SIN(U)*SIN(V)
DEQATN 48      SYV(U,V) = 5.0*SIN(U)*COS(V)
DEQATN 49      SZV(U,V) = 0.
DEQATN 51      SXUU(U,V) = -5.0*SIN(U)*COS(V)
DEQATN 52      SYUU(U,V) = -5.0*SIN(U)*SIN(V)
DEQATN 53      SZUU(U,V) = -5.0*COS(U)
DEQATN 54      SXVV(U,V) = -5.0*SIN(U)*COS(V)
DEQATN 55      SYVV(U,V) = -5.0*SIN(U)*SIN(V)
DEQATN 56      SZVV(U,V) = 0.
DEQATN 57      SXUV(U,V) = -5.0*COS(U)*SIN(V)
DEQATN 58      SYUV(U,V) = 5.0*COS(U)*COS(V)
DEQATN 59      SZUV(U,V) = 0.
$
GMSURF 1      MSCGRP1 0      0      +
+      EQUATION 0.0 1.570796327 0.0 1.570796327 +
+      41 42 43 44 45 46 47 48 49 +
+      51 52 53 54 55 56 57 58 59 +
$
$ THIS SECTION CONTAINS LOADS
$
GMLOAD 1      1      1.      0.      0.      GMSURF 1      CONST +
+      3000.
$

```

Since only an octant model is analyzed, the symmetric boundary conditions are required. The following partial commands demonstrate how GMSPC, GMSURF, and DEQATN work together. The GMSURF 201 is expressed in terms of radius (R) and an angle (U), ranging from 5 to 10 inches and 0 to 1.57 radians. GMSPC specifies the constraint in the basic z-direction.

```

$
$ Surface on x-y plane
$
DEQATN 61      SX(R,U) =  R*COS(U)
DEQATN 62      SY(R,U) =  R*SIN(U)
DEQATN 63      SZ(R,U) =  0.
DEQATN 64      SXU(R,U) =  COS(U)
DEQATN 65      SYU(R,U) =  SIN(U)
DEQATN 66      SZU(R,U) =  0.
DEQATN 67      SXV(R,U) = -R*SIN(U)
DEQATN 68      SYV(R,U) =  R*COS(U)
DEQATN 69      SZV(R,U) =  0.
DEQATN 71      SXUU(R,U) = 0.
DEQATN 72      SYUU(R,U) = 0.
DEQATN 73      SZUU(R,U) = 0.
DEQATN 74      SXVV(R,U) = -R*COS(U)
DEQATN 75      SYVV(R,U) = -R*SIN(U)
DEQATN 76      SZVV(R,U) = 0.
DEQATN 77      SXUV(R,U) = - SIN(U)
DEQATN 78      SYUV(R,U) =  COS(U)
DEQATN 79      SZUV(R,U) = 0.
$
GMSURF 201      MSCGRP1 0      0      +
+      EQUATION  5.0 10.0 0.0 1.570796327 +
+      61 62 63   64 65 66   67 68 69      +
+      71 72 73   74 75 76   77 78 79      +
$
$ x-y plane in xyz basic
$
GMSPC 1      3      GMSURF 201
$

```

Other inputs, such as FEEDGE and FEFACE, are used to specify the geometry, the element boundaries, loads, and boundary conditions. For example, the following FEEDGE command is used to specify that the element edge between grids 3 and 4 will follow the geometry of GMCURV ID 1. The complete input file has been annotated and stored in the TPL directory for review.

```

$
$ Inner radius x-y plane
$
FEEDGE 1      3      4      0      GMCURV 1
$

```

Results

The p-element model requires two adaptivity cycles to converge to the desired error tolerance of 10% at p-value of 4. Table 27-4 tabulates the results with the theory.

Table 27-4. Comparison of Displacements			
Results			Percent Difference
Deflections	Theory	NX Nastran	
Inner	1.20E-03	1.19E-03	1.0%
Outer	4.50E-04	4.45E-04	1.1%

References

[1] Roark, R. J. and Young, W. C. *Formulas for Stress and Strain*. 5th edition. McGraw-Hill, Inc., 1975, p. 506.

Motor Blade Normal Modes

Objectives

A normal mode analysis of a simplified motor blade is analyzed and compared using the h- and p-elements. The natural frequencies between 0 and 2000 Hz are computed.

Problem Description

Figure 27-4 shows the geometry of the motor blade consisting of a base and a blade. The whole assembly part is made of aluminum.

Modulus of Elasticity	= 1.0E7 psi
Poisson's ratio	= 0.3
Mass density	= 2.59E-4 lb/in ³

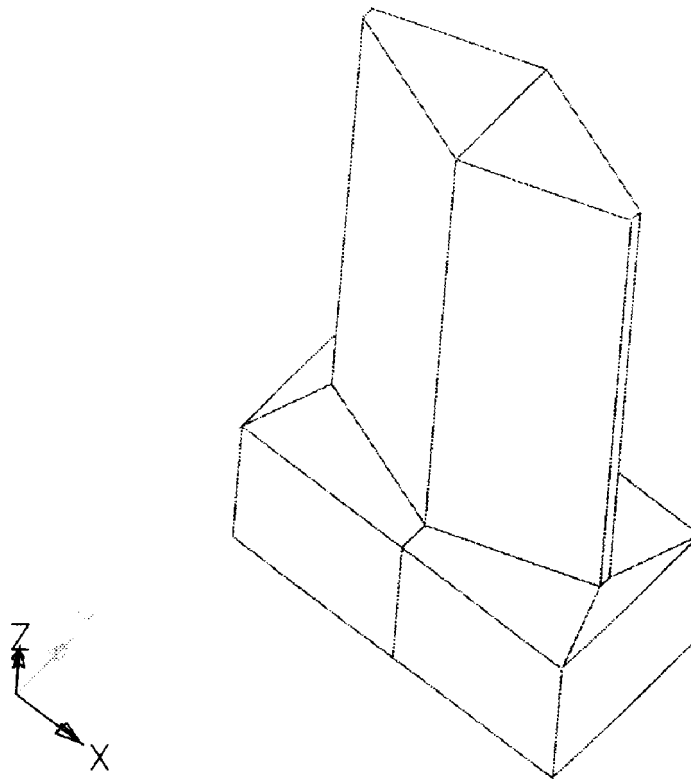


Figure 27-4. Motor Blade – p-Element Model

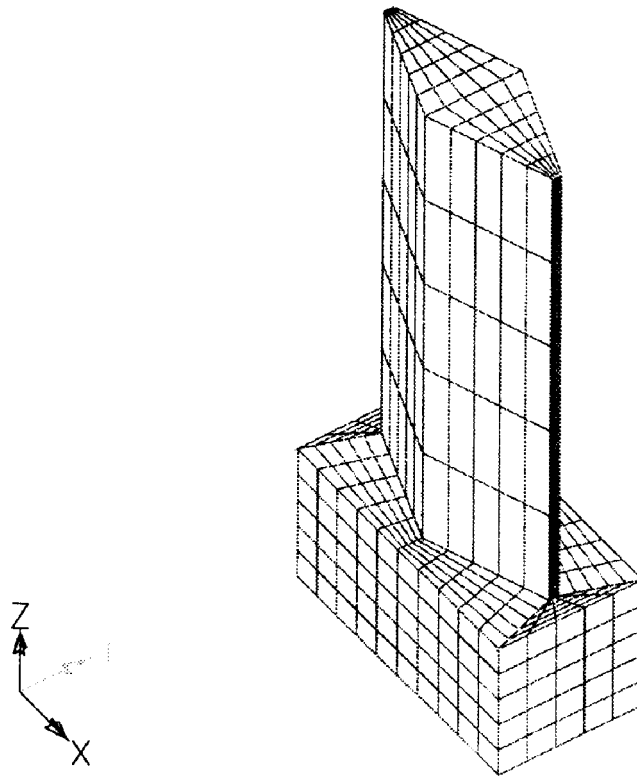


Figure 27-5. Motor Blade – Fine h-Model

Theoretical Results

There is no theoretical result. Results of three different meshes of h-elements—coarse, medium, and fine—are compared with a p-element model.

Setup

Figure 27-4 and Figure 27-5 show the p-model and the h-model with fine mesh respectively. Since only the natural frequencies are requested, the geometry of both the h- and p-meshes are simplified and the boundaries are modeled with straight edges. If a detailed stress analysis is required, the geometry should be specified more precisely with higher-order geometry.

Results

All models produce six rigid body modes, so they are not discussed here. Table 27-5 tabulates the computed elastic natural frequencies between 0.0 to 2000.0 Hz, the number of elements, and the number of degrees-of-freedom with respect to different models. The p-element results agree very well with the fine meshed h-model showing the power and usefulness of the p-elements.

Table 27-5. Comparison of Natural Frequencies of a Motor Blade							
Descriptions		No. of Elements	No. of DOFs	Elastic Modes			
	Mesh			1st	2nd	3rd	4th
h-model	Coarse	4	90	622.5	651.2	1204.9	–
	Medium	80	441	604.5	954.7	1289.6	1956.2

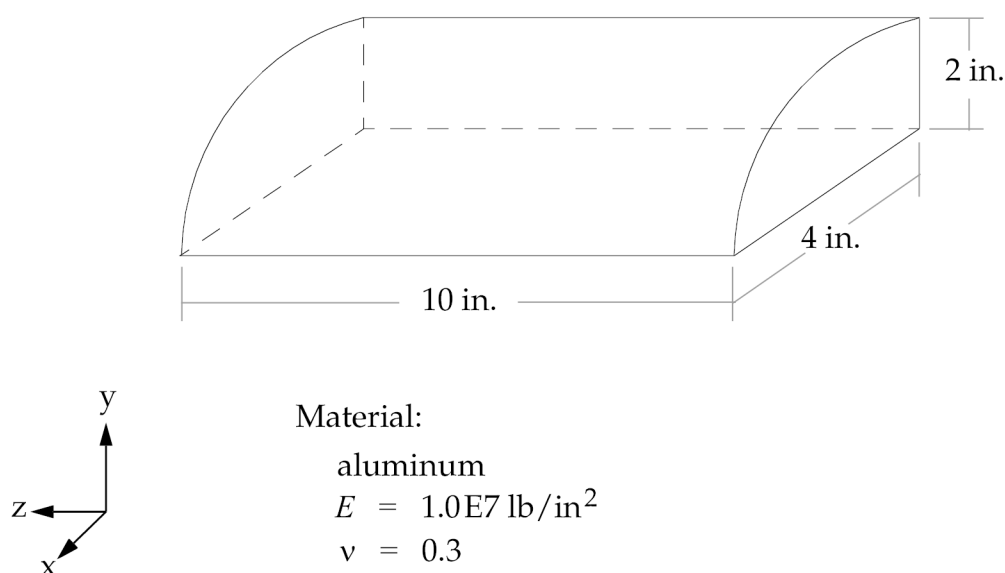
Table 27-5. Comparison of Natural Frequencies of a Motor Blade

Descriptions		No. of Elements	No. of DOFs	Elastic Modes			
	Mesh			1st	2nd	3rd	4th
	Fine	1250	4878	659.7	1051.5	1404.8	1744.7
p-adaptive	cycle=1	10	276	737.7	1064.8	1526.9	—
model	cycle=2		1161	675.8	1070.8	1426.9	1796.6
	cycle=3		2112	674.6	1071.2	1427.8	1765.2

Torsion of Elliptical Bar

Objectives and Problem Description

This example illustrates torsion of an elliptical bar.

**Figure 27-6. Octant of the Elliptical Bar Model**

Setup

The model is constructed from four CPENTA elements. There are two coordinate systems used: the cylindrical coordinate system (CORD2C entry) to define grids of the model, and the convective coordinate system (GMCORD entry) to define loads applied to the bar.

The elliptical cross section of the model is described using GMCURV and GMSURF entries, both with the equation method. For the GMCURV entry:

The general ellipse equation:

$$\frac{x^2}{a^2} + \frac{y^2}{b^2} = 1$$

as shown in **Figure 27-7**.

Define x and y in polar coordinates:

$$\begin{aligned} x &= r \cos \theta \\ y &= r \sin \theta \\ r &= \frac{ab}{\sqrt{a^2 \sin^2 \theta + b^2 \cos^2 \theta}} \\ &= \frac{a}{\sqrt{\cos^2 \theta + \frac{a^2}{b^2} \sin^2 \theta}} \end{aligned} \quad \begin{aligned} x &= \frac{a}{\sqrt{\cos^2 \theta + \frac{a^2}{b^2} \sin^2 \theta}} \cos \theta \\ y &= \frac{a}{\sqrt{\cos^2 \theta + \frac{a^2}{b^2} \sin^2 \theta}} \sin \theta \end{aligned}$$

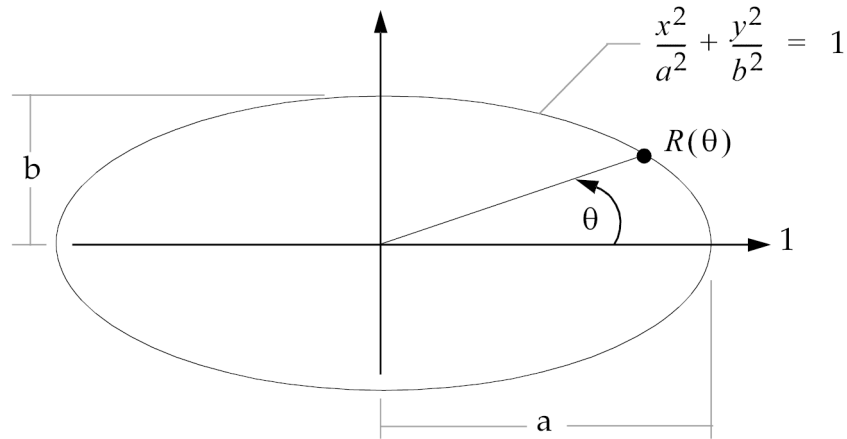


Figure 27-7.

For the GMSURF entry, three different types of parameterization were used. In the first method,

$$\begin{aligned} x_s &= \frac{v}{\sqrt{\cos^2 u + \frac{a^2}{b^2} \sin^2 u}} \cos u & 0.0 \leq u \leq \frac{\pi}{2} \\ y_s &= \frac{v}{\sqrt{\cos^2 u + \frac{a^2}{b^2} \sin^2 u}} \sin u & 0.0 \leq v \leq a \end{aligned}$$

as shown in **Figure 27-8**.

In the second method,

$$\begin{aligned}x_s &= av \cos u & 0.0 \leq u \leq \frac{\pi}{2} \\y_s &= bv \sin u & 0.0 \leq v \leq 1.0\end{aligned}$$

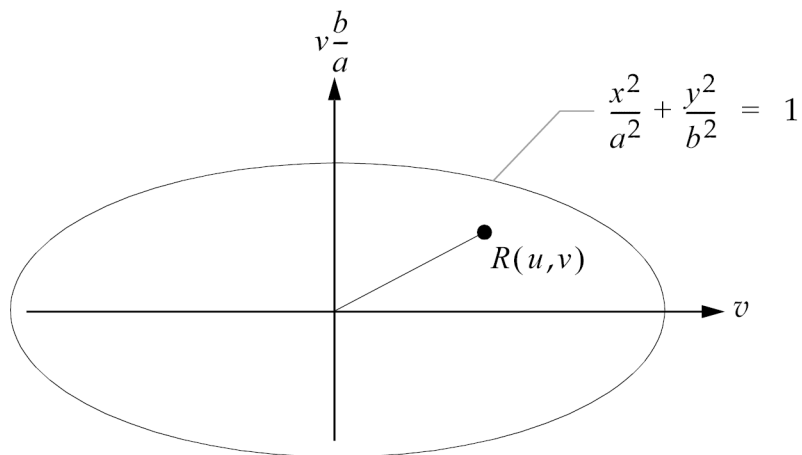


Figure 27-8.

The third method, using hyperbolic functions,

$$\begin{aligned}x + iy &= e(\cosh r \cos t + i \sinh r \sin t) \\&= e \cosh(r + it)\end{aligned}$$

$$x = e \cosh r \cos t \quad 0.0 < r < R$$

$$y = e \sinh r \sin t \quad 0.0 < t < \frac{\pi}{2}$$

$$e = \sqrt{a^2 - b^2} \quad R = \tanh^{-1}\left(\frac{b}{a}\right)$$

as shown in [Figure 27-9](#), with contours of t displayed.

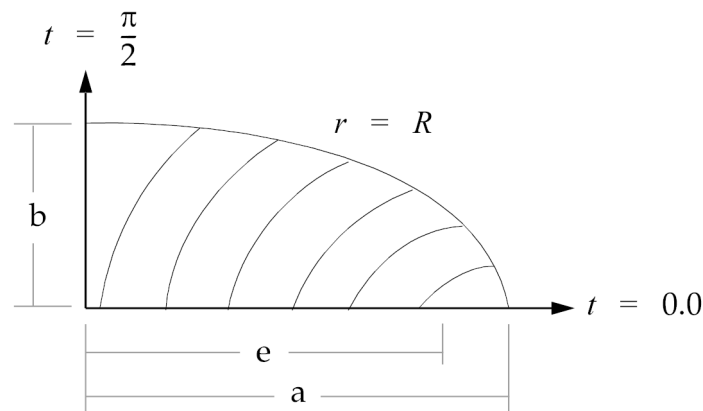


Figure 27-9.

For the GMSURF entry, the order of the input IDs of DEQATN entries for the surface is important. First, we input the IDs of DEQATN entries providing equations for the x , y , and z coordinates of the surface. Then, the IDs of DEQATN entries of the first derivatives of x , y , and z functions with respect to the surface parameter u . Then, the IDs of DEQATN entries of the first derivatives of x , y , and z functions with respect to the surface parameter v . Then, the IDs of DEQATN entries of the second derivatives of x , y , and z functions with respect to the surface parameter u . Then, the IDs of DEQATN entries of the second derivatives of x , y , and z functions with respect to the surface parameter v . Finally, the IDs of DEQATN of the mixed second derivatives of the x , y , and z functions with respect to both surface parameters u and v .

The torsion in this example is defined by applying an edge load (GMLOAD entry) using the convective coordinate system (GMCORD entry). The load is applied tangent to the GMCURV 3. This is illustrated in Figure 27-10. Note that an edge load on solid elements can cause singularities, but it is used here for demonstration purposes.

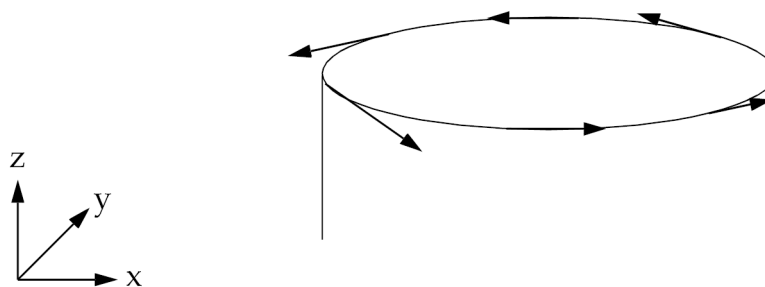


Figure 27-10.

The input file of this model is listed in Listing 27-3.


```

ID TORSION, STATIC
TIME 500
SOL 101 $ STATIC ANALYSIS
CEND
TITLE = TORSION, ELLIPSE CSA
SUBTITLE = METHOD 2
$
$ DEFINE ANALYSIS ENTRY
$
ADAPT = 100
$
LOAD = 1
SPC = 1
$
$ DEFINE OUTPUT REQUEST OPTION
$
DATAREC = 100
OUTRCV = 200
VUGRID (PUNCH) = ALL
$
$ DEFINE THE SETS OF ELEMENTS REQUIRED FOR THE ADAPTIVITY ANALYSIS
$ THIS IS REQUIRED WHEN SETTYP=SET IN PVAL ENTRY
$ (REFER TO PVAL ENTRY ALSO)
$
SETS DEFINITION
SET 120 = ALL
$
BEGIN BULK
PARAM    VUPENTA CPENTA
$
$ BOTH THE ADAPT AND PVAL ENTRY ARE USED TO
$ DEFINE ADAPTIVITY OPTION ENTRY FOR ALL ELEMENTS
$
$ FOR THE ADAPT ENTRY:
$ ID      = ADAPT ENTRY ID SELECTED IN CASE CONTROL
$ ADGEN   = ID OF THE FIRST PVAL ENTRY GENERATED IN THE ADAPTIVE PROCESS;
$          MUST BE LARGER THAN PSTRID, PMINID, AND PMAXID
$ MAXITER = NUMBER OF ITERATIONS REQUESTED BEFORE THE ADAPTIVE PROCESS STOPPED
$ PSTRTID = ID OF THE STARTING PVAL ENTRY
$ PMINID  = ID OF THE MINIMUM PVAL ENTRY
$ PMAXID  = ID OF THE MAXIMUM PVAL ENTRY
$ PART    = ANY NAME CHOSEN FOR THE ELEMENTS;
$          USED FOR ERROR ESTIMATOR REFERENCE
$ ELSETID = ID OF SETS OF ELEMENTS REQUESTED FOR ADAPTIVE PROCESS
$ TYPE    = EBEP : THE P-ORDER IS INCREASED ELEMENTS-BY-ELEMENT
$ ERREST  = ERROR ESTIMATOR METHOD USED FOR ADAPTIVE PROCESS
$ ERRTOL  = ERROR TOLERANCE (IN FRACTION), REQUIRED IF MAXITER IS NOT DEFINED
$ SIGTOL  = STRESS TOLERANCE
$

```

Listing 27-3. Input File for the Model (Continued)

```

$      SID      ADGEN      MAXITER PSTRID  PMINID  PMAXID
ADAPT  100      1001      10       10      20      30              +000001
++000001PART=ALL,ELSET=120,TYPE=EBEP,ERREST=1,ERRTOL=0.02,SIGTOL=5.0  +000002
$
$ DEFINE THE POLYNOMIAL ORDER DISTRIBUTION:
$ PVAL ID = ID OF PVAL DEFINED IN THE ADAPT ENTRY ABOVE
$ PX      = THE POLYNOMIAL ORDER DISTRIBUTION IN X-DIRECTION
$ PY      = THE POLYNOMIAL ORDER DISTRIBUTION IN Y-DIRECTION
$ PZ      = THE POLYNOMIAL ORDER DISTRIBUTION IN Z-DIRECTION
$ SETTYP  = TYPE OF SET PROVIDED. IT CAN BE EITHER SET OR ELID
$ ID      = SET ID OR ELEMENT ID, WHICHEVER IS APPROPRIATE
$
$      ID      Px      Py      Pz      SETTYP  ID
PVAL  10      2      2      2      SET      120
PVAL  20      2      2      2      SET      120
PVAL  30      4      4      4      SET      120
$
$ DEFINE THE OUTPUT REQUEST OPTION
$ REQUEST DISPLACEMENT AND STRESS OUTPUT FOR SET 120 FOR THE LAST
$ ADAPTIVITY ANALYSIS
$
OUTPUT  100              +000003
++000003ELSET=120,DISP=PRINT,STRESS=PRINT,STRAIN=NONE  +000004
++000004FIRST=NO, LAST=YES,BY=2  +000005
$
$ THE OUTPUT COORDINATE SYSTEM IS THE BASIC COORDINATE SYSTEM (CID=0)
$
OUTRCV  200      120              +000003
++000003CID=0,VIEW=2*2*2  +000004
$
$ DEFINE THE CYLINDRICAL COORDINATE SYSTEM FOR
$ GRID POINTS
$
CORD2C  1      0      0.0      0.0      0.0      0.0      0.0      1.0  +000005
++0000051.0      0.0      0.0              +000006
$

```

Listing 27-3. Input File for the Model (Continued)

```

$ DEFINE THE CONVECTIVE COORDINATE SYSTEM FOR THE APPLIED LOAD
$ IF GMCURV IS USED AS THE ENTITY TO DEFINE THE COORDINATE SYSTEM,
$ TWO ID'S ARE REQUIRED. THE FIRST ID IS THE GMCURV ID AND THE SECOND
$ ONE IS THE CORRESPONDING GMSURF ID
$
$      CID      ENTITY  ID 1    ID 2
GMCORD 10      GMCURV  3      2
$
$ DEFINE PENTA ELEMENTS
$
CPENTA 1      1      12      1      2      11      4      5
CPENTA 2      1      12      2      3      11      5      6
CPENTA 3      1      11      4      5      10      7      8
CPENTA 4      1      11      5      6      10      8      9
$
$ DEFINE GRID POINTS
$
GRID   1      1      4.      0.0    0.0
GRID   2      1      2.52982245.  0.
GRID   3      1      2.0      90.0    0.0
GRID   4      1      4.0      0.0     5.0
GRID   5      1      2.52982245.0  5.0
GRID   6      1      2.0      90.0    5.0
GRID   7      1      4.0      0.0    10.0
GRID   8      1      2.52982245.0  10.0
GRID   9      1      2.0      90.0    10.0
GRID  10      1      0.0      0.0    10.0
GRID  11      1      0.0      0.0     5.0
GRID  12      1      0.0      0.0     0.0
$

```

Listing 27-3. Input File for the Model (Continued)

```

$ DEFINE FEEDGES
$ FEEDGES BY DEFAULT ARE STRAIGHT LINES
$ GMCURVS ARE REQUIRED FOR FEEDGES THAT DEFINE CURVED EDGES
$
$ FEEDGE 18 AND FEEDGE 19 MAKE GMCURV 1
$ FEEDGE 20 AND FEEDGE 21 MAKE GMCURV 2
$ FEEDGE 22 AND FEEDGE 23 MAKE GMCURV 3
$
$      ID      GRID 1  GRID 2  CID      GEOMIN  ID
FEEDGE 1      12      1      0
FEEDGE 2      12      2      0
FEEDGE 3      12      3      0
FEEDGE 4      11      4      0
FEEDGE 5      11      5      0
FEEDGE 6      11      6      0
FEEDGE 7      10      7      0
FEEDGE 8      10      8      0
FEEDGE 9      10      9      0
FEEDGE 10     12      11     0
FEEDGE 11     1      4      0
FEEDGE 12     2      5      0
FEEDGE 13     3      6      0
FEEDGE 14     11     10     0
FEEDGE 15     4      7      0
FEEDGE 16     5      8      0
FEEDGE 17     6      9      0
FEEDGE 18     1      2      0      GMCURV 1
FEEDGE 19     2      3      0      GMCURV 1
FEEDGE 20     4      5      0      GMCURV 2
FEEDGE 21     5      6      0      GMCURV 2
FEEDGE 22     7      8      0      GMCURV 3
FEEDGE 23     8      9      0      GMCURV 3
$
$ DEFINE CURVE GEOMETRY USING EQUATION METHOD
$
DEQATN 1      X(U) = 4.0*COS(U) / (SQRT(1.0+3.0*SIN(U)*SIN(U)))
DEQATN 2      Y(U) = 4.0*SIN(U) / (SQRT(1.0+3.0*SIN(U)*SIN(U)))
DEQATN 3      Z(U) = 0.0
DEQATN 4      Z1(U) = 5.0
DEQATN 5      Z2(U) = 10.0
DEQATN 6      XP(U) = -16.0*SIN(U) / ((1.0+3.0*SIN(U)*SIN(U))**1.5)
DEQATN 7      YP(U) = 4.0*COS(U) / ((1.0+3.0*SIN(U)*SIN(U))**1.5)
DEQATN 8      ZP(U) = 0.0
DEQATN 9      XPP(U) = (-16.0*COS(U)+96.0*SIN(U)*SIN(U)*COS(U)) /      +000009
++000009      (1.0+3.0*SIN(U)*SIN(U))**2.5      +000010
DEQATN 10     YPP(U) = (-4.0*SIN(U)*(10.0-6.0*SIN(U)*SIN(U))) /      +000011
++000011      ((1.0+3.0*SIN(U)*SIN(U))**2.5)      +000012
DEQATN 11     ZPP(U) = 0.0
$
$
GMCURV 1      MSCGRP1 0      0      +000013
++000013EQUATION,0.0,1.5708,1,2,3,6,7,      +000014
++0000148      9      10      11      +000015
GMCURV 2      MSCGRP1 0      0      +000016
++000016EQUATION,0.0,1.5708,1,2,4,6,7,      +000017
++0000178      9      10      11      +000018
GMCURV 3      MSCGRP1 0      0      +000019
++000019EQUATION,0.0,1.5708,1,2,5,6,7,      +000020
++0000208      9      10      11      +000021
$

```

Listing 27-3. Input File for the Model (Continued)

```

$ DEFINE FEFACES
$
$      ID      GRID 1  GRID 2  GRID 3  GRID 4  CID      SURFID
FEFACE 1      12      1      2              0      1
FEFACE 2      12      2      3              0      1
FEFACE 3      11      4      5              0
FEFACE 4      11      5      6      0              0
FEFACE 5      10      7      8              0      2
FEFACE 6      10      8      9              0      2
FEFACE 7      12      1      4      11      0
FEFACE 8      12      3      6      11      0
FEFACE 9      11      4      7      10      0
FEFACE 10     11      6      9      10      0
$
$ DEFINE SURFACE GEOMETRY USING EQUATION METHOD
$
DEQATN 13      X(U,V)  = 4.0*V*COS(U)
DEQATN 14      Y(U,V)  = 2.0*V*SIN(U)
DEQATN 15      Z1(U,V) = 0.0
DEQATN 16      Z2(U,V) = 10.0
DEQATN 17      XU(U,V) = -4.0*V*SIN(U)
DEQATN 18      YU(U,V) = 2.0*V*COS(U)
DEQATN 19      ZU(U,V) = 0.0
DEQATN 20      XV(U,V) = 4.0*COS(U)
DEQATN 21      YV(U,V) = 2.0*SIN(U)
DEQATN 22      ZV(U,V) = 0.0
DEQATN 23      XUU(U,V) = -4.0*V*COS(U)
DEQATN 24      YUU(U,V) = -2.0*V*SIN(U)
DEQATN 25      ZUU(U,V) = 0.0
DEQATN 26      XVV(U,V) = 0.0
DEQATN 27      YVV(U,V) = 0.0
DEQATN 28      ZVV(U,V) = 0.0
DEQATN 29      XUV(U,V) = -4.0*SIN(U)
DEQATN 30      YUV(U,V) = 2.0*COS(U)
DEQATN 31      ZUV(U,V) = 0.0
$
$
GMSURF 1      MSCGRP1 0      0      +000022
++000022EQUATION,0.0,1.5708,0.0,1.0,13,14,15,      +000023
++00002317,18,19,20,21,22,23,24,      +000024
++00002425,26,27,28,29,30,31      +000025
GMSURF 2      MSCGRP1 0      0      +000026
++000026EQUATION,0.0,1.5708,0.0,1.0,13,14,16,      +000027
++00002717,18,19,20,21,22,23,24,      +000028
++00002825,26,27,28,29,30,31      +000029

```

Listing 27-3. Input File for the Model (Continued)

```

$ DEFINE CONSTRAINT IN THE X-DIRECTION
$
$      SID      C      ENTITY  ID
GMSPC  1        1      FEFACE  1
GMSPC  1        1      FEFACE  2
GMSPC  1        1      FEFACE  7
GMSPC  1        1      FEFACE  9
$
$ DEFINE CONSTRAINT IN THE Y-DIRECTION
$
$      SID      C      ENTITY  ID
GMSPC  1        2      FEFACE  1
GMSPC  1        2      FEFACE  2
GMSPC  1        2      FEFACE  8
GMSPC  1        2      FEFACE 10
$
$ DEFINE CONSTRAINT IN THE Z-DIRECTION
$
$      SID      C      ENTITY  ID
GMSPC  1        3      FEFACE  7
GMSPC  1        3      FEFACE  8
GMSPC  1        3      FEFACE  9
GMSPC  1        3      FEFACE 10
$
$ DEFINE THE MATERIAL PROPERTY (ALUMINUM)
$
MAT1    1        1.+7          0.3
PSOLID  1        1              GRID      SMECH
$
$ CONSTANT LOAD OF 100 LB/INCH IS APPLIED TANGENTIALLY TO
$ GMCURV 3
$
$ LID          = LOAD IDENTIFICATION NUMBER DEFINED IN CASE CONTROL SECTION
$ CID          = COORDINATE SYSTEM USED TO DEFINE THE APPLIED LOAD
$ N1,N2,N3    = DIRECTION OF THE LOAD VECTOR
$ ENTITY      = ENTITY AT WHICH LOAD IS APPLIED TO
$ ID          = ID OF THE ENTITY
$ METHOD       = METHOD USED TO DEFINE THE LOAD
$
$      LID      CID      N1      N2      N3      ENTITY  ID      METHOD
GMLOAD  1        10      1.0     0.0     0.0     GMCURV  3      CONSTANT +000007
$
$      VALUE OF LOAD DENSITY
++000007100.0                                +000008
$
ENDDATA

```

Listing 27-3. Input File for the Model

Results

The OUTPUT entry requests the displacements and stress output solely for the last adaptivity iteration. An abridged output for this model is shown in [Figure 27-11](#).

```

1  TORSION, ELLIPSE CSA                                JULY 23, 2003 NX NASTRAN 07/23/03 PAGE 73
METHOD 2
0*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL
LOAD SEQ. NO.      EPSILON      EXTERNAL WORK      EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
1                5.2309875E-15      1.6239751E-02
*** USER INFORMATION MESSAGE 6502 (ADAPT)
THE CURRENT SUPERELEMENT REQUIRES NO FURTHER P-ADAPTIVE ANALYSIS BECAUSE THERE ARE NO CHANGES IN P-VALUES.
0*** USER WARNING MESSAGE 4594. THE POSTCDB CONTAINS NO LEGITIMATE SURFACE OR VOLUME DEFINITIONS.
1  TORSION, ELLIPSE CSA                                JULY 23, 2003 NX NASTRAN 07/23/03 PAGE 74
METHOD 2                                ADAPTIVITY INDEX= 3
0

DISPLACEMENT VECTOR

POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
1          G          .0          .0          .0          .0          .0          .0
2          G          .0          .0      -8.692895E-07      .0          .0          .0
3          G          .0          .0          .0          .0          .0          .0
4          G          .0      1.357096E-05          .0          .0          .0          .0
5          G      -3.062882E-06      -2.173771E-06      -1.839771E-06      .0          .0          .0
6          G      1.593498E-05          .0          .0          .0          .0          .0
7          G          .0      1.513064E-04          .0          .0          .0          .0
8          G      1.887783E-06      -4.328616E-05      1.990558E-05      .0          .0          .0
9          G      1.121671E-04          .0          .0          .0          .0          .0
10         G          .0          .0          .0          .0          .0          .0
11         G          .0          .0          .0          .0          .0          .0
12         G          .0          .0          .0          .0          .0          .0
201001001  G          .0          .0      -6.008400E-23      .0          .0          .0
201001002  G          .0          .0      -3.919389E-23      .0          .0          .0
201001003  G          .0          .0      -1.764301E-22      .0          .0          .0
201001004  G          .0          .0      -4.865105E-07      .0          .0          .0
201001005  G          .0          .0      -8.088551E-07      .0          .0          .0
201001006  G          .0          .0      -8.692895E-07      .0          .0          .0
201001007  G      -3.961429E-23      1.151926E-22      6.248783E-24      .0          .0          .0
201001008  G      -1.317319E-22      2.320618E-06      -1.101103E-22      .0          .0          .0
201001009  G      -1.197587E-22      3.720583E-06      -2.934319E-24      .0          .0          .0
201001010  G      -7.882488E-07      1.256443E-06      -2.743236E-07      .0          .0          .0
201001011  G      -1.074418E-06      2.716110E-06      -1.095060E-06      .0          .0          .0
201001012  G      -1.967381E-06      1.468333E-06      -7.467912E-07      .0          .0          .0
201001013  G      -9.475001E-23      4.980112E-22      -3.150599E-23      .0          .0          .0
201001014  G      -3.235471E-22      3.547284E-06      -1.786648E-22      .0          .0          .0
201001015  G      -4.842875E-22      1.357096E-05      6.322074E-23      .0          .0          .0
201001016  G      -8.565316E-07      4.229675E-06      -7.164187E-07      .0          .0          .0
201001017  G      -3.000400E-06      4.619390E-06      -5.842737E-07      .0          .0          .0
201001018  G      -3.062882E-06      -2.173771E-06      -1.839771E-06      .0          .0          .0
201002001  G          .0          .0      3.004200E-23      .0          .0          .0
201002002  G          .0          .0      -4.865105E-07      .0          .0          .0
201002003  G          .0          .0      -8.692895E-07      .0          .0          .0
201002004  G          .0          .0      2.921973E-23      .0          .0          .0
201002005  G          .0          .0      5.445964E-07      .0          .0          .0
201002006  G          .0          .0      4.219457E-22      .0          .0          .0
201002007  G      -1.446019E-22      -5.887475E-23      -3.124391E-24      .0          .0          .0
201002008  G      -7.882488E-07      1.256443E-06      -2.743236E-07      .0          .0          .0
201002009  G      -1.967381E-06      1.468333E-06      -7.467912E-07      .0          .0          .0
201002010  G      -8.927043E-07      3.961755E-23      -6.052314E-24      .0          .0          .0
201002011  G      -2.660616E-06      7.911209E-07      5.446515E-08      .0          .0          .0
201002012  G      -7.240385E-08      1.330381E-22      -1.194728E-22      .0          .0          .0
201002013  G      -8.723581E-22      -3.184499E-22      1.575299E-23      .0          .0          .0
201002014  G      -8.565316E-07      4.229675E-06      -7.164187E-07      .0          .0          .0
201002015  G      -3.062882E-06      -2.173771E-06      -1.839771E-06      .0          .0          .0
201002016  G      -1.335821E-06      1.364713E-22      2.932544E-23      .0          .0          .0
201002017  G      -7.250181E-06      1.251031E-06      1.185184E-06      .0          .0          .0
201002018  G      1.593498E-05      2.834062E-22      4.533070E-22      .0          .0          .0
201003001  G      -9.475001E-23      4.980112E-22      -3.150599E-23      .0          .0          .0
201003002  G      -3.235471E-22      3.547284E-06      -1.786648E-22      .0          .0          .0

```

Figure 27-11. Abridged Output for the Model (Continued)

```

1  TORSION, ELLIPSE CSA                      JULY 23, 2003 NX NASTRAN 07/23/03  PAGE    77
   METHOD 2                                ADAPTIVITY INDEX=      3
0                                         PVAL ID=    1002 (P1,P2,P3=  3  2  3), SUBCASE    1
      STRESSES IN P - VERSION PENTAHEDRON SOLID ELEMENTS ( P E N T A )
VIEW-ELEMENT ID=100001001, P-ELEMENT ID =    1, OUTPUT COORD. ID=    0, P OF EDGES =  3  3  4  4  3  4  4  4
VUGRID  SIGMA-X/1      SIGMA-Y/2      SIGMA-Z/3      TAU-XY      TAU-YZ      TAU-XZ      MEAN-PRESSURE      VON-MISES
201001001  2.507561E-15  2.507561E-15  5.850975E-15  -2.450845E-31  -3.490772E+00  3.045360E-16
3.490772E+00  -3.490772E+00  2.881587E-15
201001002  -9.512720E-17  -6.704689E-16  3.533783E-16  7.719806E-32  7.080933E-01  -7.649862E-16
7.080933E-01  -7.080933E-01  -2.876032E-16
201001004  5.034403E+00  5.034403E+00  1.174694E+01  4.981801E-15  -1.109684E+01  -4.317494E+00
2.076182E+01  -3.980472E+00  5.034403E+00
201001007  1.006225E+01  2.347857E+01  1.006225E+01  -2.168649E+00  3.630430E-01  -1.325106E-16
2.382975E+01  9.711069E+00  1.006225E+01
201001008  -3.267787E+00  -7.624836E+00  -3.267787E+00  -1.073696E+00  -6.464741E-01  -1.551473E-15
-2.933002E+00  -7.959620E+00  -3.267787E+00
201001010  -1.527811E+00  -2.566048E+00  -9.243130E-01  -5.258331E-02  -3.754325E-01  -2.206367E+00
1.018136E+00  -3.538538E+00  -2.497769E+00
      STRESSES IN P - VERSION PENTAHEDRON SOLID ELEMENTS ( P E N T A )
VIEW-ELEMENT ID=100001002, P-ELEMENT ID =    1, OUTPUT COORD. ID=    0, P OF EDGES =  3  3  4  4  3  4  4  4
VUGRID  SIGMA-X/1      SIGMA-Y/2      SIGMA-Z/3      TAU-XY      TAU-YZ      TAU-XZ      MEAN-PRESSURE      VON-MISES
201001002  -9.512720E-17  -6.704689E-16  3.533783E-16  7.719806E-32  7.080933E-01  -7.649862E-16
7.080933E-01  -7.080933E-01  -2.876032E-16
201001003  2.474018E-14  5.246847E-14  2.999879E-14  -1.088513E-15  2.914146E+00  -2.396744E-15
2.914146E+00  -2.914146E+00  2.316292E-14
201001005  -7.025239E-01  -7.025239E-01  -1.639222E+00  3.874331E-15  1.205412E+01  -6.993091E+00
1.277274E+01  -1.511449E+01  -7.025239E-01
201001008  -3.267787E+00  -7.624836E+00  -3.267787E+00  -1.073696E+00  -6.464741E-01  -1.551473E-15
-2.933002E+00  -7.959620E+00  -3.267787E+00
201001009  6.025518E+00  1.405954E+01  6.025518E+00  7.571502E-01  6.719486E+00  7.331966E-16
1.790772E+01  2.177344E+00  6.025518E+00
201001011  2.021004E+00  -5.901449E+00  -1.849734E+00  2.802238E+00  2.882026E+00  -1.129690E+00
2.922884E+00  -8.277588E+00  -3.754748E-01
1.910060E+00  9.969281E+00
1  TORSION, ELLIPSE CSA                      JULY 23, 2003 NX NASTRAN 07/23/03  PAGE    78
   METHOD 2                                ADAPTIVITY INDEX=      3
0                                         PVAL ID=    1002 (P1,P2,P3=  3  2  3), SUBCASE    1
      STRESSES IN P - VERSION PENTAHEDRON SOLID ELEMENTS ( P E N T A )
VIEW-ELEMENT ID=100001003, P-ELEMENT ID =    1, OUTPUT COORD. ID=    0, P OF EDGES =  3  3  4  4  3  4  4  4
VUGRID  SIGMA-X/1      SIGMA-Y/2      SIGMA-Z/3      TAU-XY      TAU-YZ      TAU-XZ      MEAN-PRESSURE      VON-MISES
201001005  -7.025239E-01  -7.025239E-01  -1.639222E+00  3.874331E-15  1.205412E+01  -6.993091E+00
1.277274E+01  -1.511449E+01  -7.025239E-01
201001004  5.034403E+00  5.034403E+00  1.174694E+01  4.981801E-15  -1.109684E+01  -4.317494E+00
2.076182E+01  -3.980472E+00  5.034403E+00
201001002  -9.512720E-17  -6.704689E-16  3.533783E-16  7.719806E-32  7.080933E-01  -7.649862E-16
7.080933E-01  -7.080933E-01  -2.876032E-16
201001011  2.021004E+00  -5.901449E+00  -1.849734E+00  2.802238E+00  2.882026E+00  -1.129690E+00
2.922884E+00  -8.277588E+00  -3.754748E-01
201001010  -1.527811E+00  -2.566048E+00  -9.243130E-01  -5.258331E-02  -3.754325E-01  -2.206367E+00
1.018136E+00  -3.538538E+00  -2.497769E+00
201001008  -3.267787E+00  -7.624836E+00  -3.267787E+00  -1.073696E+00  -6.464741E-01  -1.551473E-15
-2.933002E+00  -7.959620E+00  -3.267787E+00
      STRESSES IN P - VERSION PENTAHEDRON SOLID ELEMENTS ( P E N T A )
VIEW-ELEMENT ID=100001004, P-ELEMENT ID =    1, OUTPUT COORD. ID=    0, P OF EDGES =  3  3  4  4  3  4  4  4
VUGRID  SIGMA-X/1      SIGMA-Y/2      SIGMA-Z/3      TAU-XY      TAU-YZ      TAU-XZ      MEAN-PRESSURE      VON-MISES
201001004  5.034403E+00  5.034403E+00  1.174694E+01  4.981801E-15  -1.109684E+01  -4.317494E+00
2.076182E+01  -3.980472E+00  5.034403E+00
201001005  -7.025239E-01  -7.025239E-01  -1.639222E+00  3.874331E-15  1.205412E+01  -6.993091E+00
1.277274E+01  -1.511449E+01  -7.025239E-01
201001006  -3.200166E-02  -3.200166E-02  -7.467054E-02  -8.733983E-14  1.493832E+01  -4.544772E+00
1.556104E+01  -1.566771E+01  -3.200166E-02
201001010  -1.527811E+00  -2.566048E+00  -9.243130E-01  -5.258331E-02  -3.754325E-01  -2.206367E+00
1.018136E+00  -3.538538E+00  -2.497769E+00
201001011  2.021004E+00  -5.901449E+00  -1.849734E+00  2.802238E+00  2.882026E+00  -1.129690E+00
2.922884E+00  -8.277588E+00  -3.754748E-01
201001012  2.319424E+00  -3.111391E+00  -1.438776E+00  -7.330344E+00  8.622318E+00  -9.914967E+00
1.681246E+01  -1.124202E+01  -7.801181E+00
7.435812E-01  2.650212E+01

```

Figure 27-11. Abridged Output for the Model

Circular Solid Model

Objectives and Problem Description

This example is a circular solid with a simply supported edge. It is made of aluminum and has the following properties:

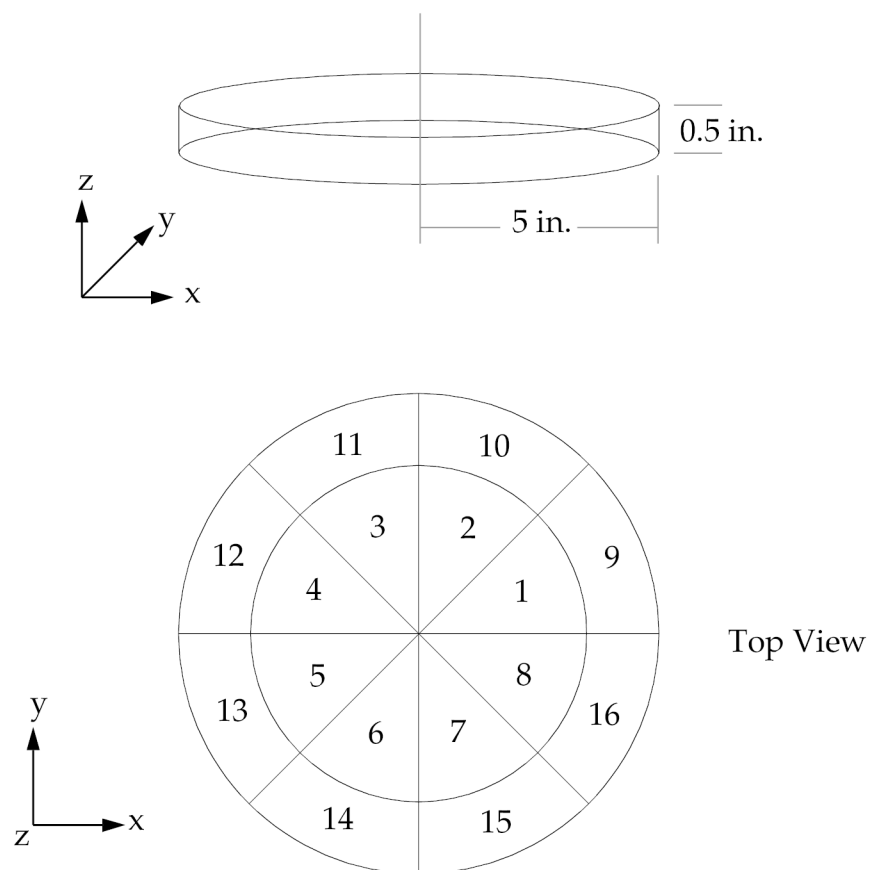
$$E = 1.0\text{E}7 \text{ lb/in}^2$$

$$\rho_w = 0.098 \text{ lbf/in}^3$$

$$\nu = 0.3$$

Equation 27-1.

It is illustrated in **Figure 27-12**.



element 1 thru 8: CPENTA elements
 element 9 thru 16: CHEXA elements (boundary elements)

Figure 27-12. The Circular Solid Model

Theoretical Results

The behavior of this circular solid model can be compared to the theoretical circular plate model described in Reference [1]. From Reference [1],

$$f = \frac{K_n}{2\pi} \sqrt{\frac{Dg}{wr^4}}$$

where:

$$\begin{aligned} D &= \frac{Et^3}{12(1-\nu^2)} \\ w &= \text{uniform load per unit area including own weight} \\ &= \frac{(\rho_w)(Volume)}{Area} \\ &= \frac{(\rho_w)(Area)(t)}{Area} \\ &= (\rho_m)(g)(t) \end{aligned}$$

$$f = \frac{K_n}{2\pi} \sqrt{\frac{Et^2}{12(1-\nu^2)(\rho_m)(r^4)}} \text{ cycles/sec}$$

where:

$$\begin{aligned} E &= \text{modulus of elasticity} \\ K_n &= \text{constant, } n \text{ refers to the mode of vibration} \\ t &= \text{thickness of the plate} \\ r &= \text{radius of the plate} \\ \nu &= \text{Poisson's ratio} \end{aligned}$$

For aluminum,

$$\begin{aligned} E &= 1.0\text{E}7 \text{ lb/in}^2 \\ \nu &= 0.3 \\ \rho_w &= 0.098 \text{ lbf/in}^3 \\ \rho_m &= 2.538\text{E-}4 \text{ lbf/in}^3 \end{aligned}$$

For the circular plate, radius (r) = 5 inches and thickness (t) = 0.5 inches. The theoretical natural frequencies for all modes:

$$f = \frac{K_n}{2\pi} \sqrt{\frac{(10^7)(0.5^2)}{12(1-0.3^2)(2.538 \times 10^{-4})(5^4)}} = 191.2 K_n \text{ cycles/sec}$$

The theoretical natural frequencies for the first several modes are calculated and tabulated in [Table 27-6](#).

Table 27-6. Theoretical Natural Frequency Values for Circular Plate		
Mode	K_n	Natural Frequency (cycles/sec)
Fundamental	4.99	954.1
One Nodal Diameter	13.9	2657.7
Two Nodal Diameters	25.7	4913.9
One Nodal Circle	29.8	5697.8

Setup

The model is constructed from 8 CPENTA elements and 8 CHEXA elements. The CHEXA elements are the boundary elements. The circular edge of the model is described using the circle equation (GMCURV entry, method=equation). Since the GMCURV can not be a closed curve, the full circle must be described with multiple GMCURV entries. In this model, one full circle is defined by two GMCURV entries. The first GMCURV entry has an input range from 0.0 to 3.1416 radians, and the second GMCURV entry has an input range from 3.1415 to 6.2832 radians. Note that there is some overlap, so that both segments include π . In addition, since the circle equation is a straightforward equation, the first and the second derivatives of the equation are not necessarily required. In this case, the derivatives are then computed numerically.

For the boundary conditions, GMSPC entries are used. Due to the hierarchy order which places grids as the highest priority, GMSPC entries that are imposed on grid 4 and 8 in the x-direction will override GMSPC entries on the adjacent FEFACES in the z-direction. Thus, it is necessary to add two more GMSPC entries for grids 4 and 8 in the z-direction to restore the constraints in the z-direction.

For the material property, the MAT1 entry is used. In the MAT1 entry, the weight density term ρ_w must be converted to the mass density ρ_m for consistency of units. Therefore,

$$\begin{aligned}\rho_m &= (0.098 \text{ lbf/in}^3)(1 / 32.174 \text{ ft/s}^2)(1 / 12 \text{ in/ft}) \\ &= 2.538\text{E-}4 \text{ lbm/in}^3\end{aligned}$$

The input file for this model is shown in [Listing 27-4](#).

```
ID NORMAL, MODES
TIME 200
SOL 103 $ NORMAL MODE ANALYSIS
CEND
TITLE = CIRCULAR PLATE
SUBTITLE = EDGE SIMPLY SUPPORTED
METHOD = 100
$
$ DEFINE ADAPTIVITY ANALYSIS ENTRY
$
ADAPT = 1
$
$ OUTPUT REQUEST
$
DISPLACEMENT = ALL
ECHO = BOTH
$
SPC =1
```

```

$
$ REQUEST FOR VIEW COORDINATES OF THE P-ELEMENTS TO BE
$ PRINTED IN THE PUNCH FILE. THE VIEW COORDINATES
$ ARE THE POINTS AT WHICH THE DISPLACEMENTS ARE
$ CALCULATED.
$
VUGRID(PUNCH) = ALL
$
$ DEFINE THE SETS OF ELEMENTS REQUIRED FOR THE ADAPTIVITY
$ ANALYSIS. THIS IS REQUIRED WHEN SETTYP=SET IN PVAL ENTRY
$ ( REFER TO PVAL ENTRY ALSO )
$
SETS DEFINITION
SET 10 = ALL
$
BEGIN BULK
PARAM    POST    0
$
PARAM    VUHEXA  CHEXA
PARAM    VUPENTA CPENTA
$
$ BOTH THE ADAPT AND PVAL ENTRY ARE USED TO
$ DEFINE ADAPTIVITY OPTION ENTRY FOR ALL ELEMENTS,
$
$ FOR THE ADAPT ENTRY:
$ ID      = ADAPT ENTRY ID SELECTED IN CASE CONTROL
$ ADGEN   = ID OF THE FIRST PVAL ENTRY GENERATED IN THE ADAPTIVE PROCESS;
$          MUST BE LARGER THAN PSTRID, PMINID, AND PMAXID
$ MAXITER = NUMBER OF ITERATIONS REQUESTED BEFORE THE ADAPTIVE PROCESS STOPPED
$ PSTRTID = ID OF THE STARTING PVAL ENTRY
$ PMINID  = ID OF THE MINIMUM PVAL ENTRY
$ PMAXID  = ID OF THE MAXIMUM PVAL ENTRY
$ PART    = ANY NAME CHOSEN FOR THE ELEMENTS;
$          USED FOR ERROR ESTIMATOR REFERENCE
$ ELSETID = ID OF SETS OF ELEMENTS REQUESTED FOR ADAPTIVE PROCESS
$ TYPE    = EBEP : THE P-ORDER WILL INCREASE ONLY IN THE ELEMENTS WHOSE ERROR
$          IS GREATER THAN ERROR TOLERANCE.
$ ERREST  = ERROR ESTIMATOR METHOD USED FOR ADAPTIVE PROCESS
$ ERRTOL  = ERROR TOLERANCE (IN FRACTION), REQUIRED IF MAXITER IS NOT DEFINED
$ SIGTOL  = STRESS TOLERANCE
$
$          ID      ADGEN   MAXITER PSTRTID PMINID  PMAXID
ADAPT    1        1001    5         10      20      30
$
++000001PART=ALL,ELSET=10,TYPE=EBEP,ERREST=1,ERRTOL=0.02,SIGTOL=0.0
$

```

Listing 27-4. Input File for the Circular Solid Model (Continued)

```

$ DEFINE THE POLYNOMIAL ORDER DISTRIBUTION:
$ PVAL ID = ID OF PVAL DEFINED IN THE ADAPT ENTRY ABOVE
$ Px      = THE POLYNOMIAL ORDER DISTRIBUTION IN X-DIRECTION
$ Py      = THE POLYNOMIAL ORDER DISTRIBUTION IN Y-DIRECTION
$ Pz      = THE POLYNOMIAL ORDER DISTRIBUTION IN Z-DIRECTION
$ SETTYP  = TYPE OF SET PROVIDED. IT CAN BE EITHER SET or ELID
$ ID      = SET ID or ELEMENT ID WHICHEVER IS APPROPRIATE
$
$      ID      Px      Py      Pz      SETTYP  ID
PVAL    10      2      2      2      SET      10
PVAL    20      4      4      2      SET      10
PVAL    30      6      6      2      SET      10
$
$ DEFINE A CYLINDRICAL COORDINATE SYSTEM
$ USED IN DEFINING THE GRIDS
$
CORD2C  1      0      0.0    0.0    0.0    0.0    0.0    1.0    +000003
++0000031.0    0.0    0.0
$
$ DEFINE SOLID ELEMENTS
$ HEXA ELEMENTS ARE THE BOUNDARY ELEMENTS
$
CPENTA  1      1      1      21      22      10      29      30
CPENTA  2      1      1      22      23      10      30      31
CPENTA  3      1      1      23      24      10      31      32
CPENTA  4      1      1      24      25      10      32      33
CPENTA  5      1      1      25      26      10      33      34
CPENTA  6      1      1      26      27      10      34      35
CPENTA  7      1      1      27      28      10      35      36
CPENTA  8      1      1      28      21      10      36      29
CHEXA   9      1      2      3      22      21      11      12      +000005
++00000530    29      +000006
CHEXA   10     1      3      4      23      22      12      13      +000007
++00000731    30      +000008
CHEXA   11     1      4      5      24      23      13      14      +000009
++00000932    31      +000010
CHEXA   12     1      5      6      25      24      14      15      +000011
++00001133    32      +000012
CHEXA   13     1      6      7      26      25      15      16      +000013
++00001334    33      +000014
CHEXA   14     1      7      8      27      26      16      17      +000015
++00001535    34      +000016
CHEXA   15     1      8      9      28      27      17      18      +000017
++00001736    35      +000018
CHEXA   16     1      9      2      21      28      18      11      +000019
++00001929    36
$
$DEFINE EIGRL CARD (LANCZOS METHOD)
$
EIGRL   100      10
$
$DEFINE THE CURVE GEOMETRY, METHOD = EQUATION, U IS THE CURVE PARAMETER
$
$DEFINE THE OUTER BOTTOM CIRCLE OF RADIUS=5.0 AT Z=0.0 USING DEQATN 1,2,AND 3.
$THE OUTER BOTTOM CIRCLE IS DEFINED BY TWO GMCURV ENTRIES:
$GMCURV 1 WITH THE RANGE OF CURVE PARAMETER,U = 0.0 TO 3.1416 RADIANS
$GMCURV 2 WITH THE RANGE OF CURVE PARAMETER,U = 3.1415 TO 6.2832 RADIANS
$
DEQATN  1      X(U) = 5.0*COS(U)
DEQATN  2      Y(U) = 5.0*SIN(U)
DEQATN  3      Z(U) = 0.0
$

```

Listing 27-4. Input File for the Circular Solid Model (Continued)

```

GMCURV 1      MSCGRP1 0      0      +000021
++000021EQUATION,0.0,3.1416,1,2,3      +000022
GMCURV 2      MSCGRP1 0      0      +000025
++000025EQUATION,3.1415,6.2832,1,2,3      +000026
$
$ DEFINE THE TOP OUTER CIRCLE USING THE SAME EQUATION, DEQATN 1 AND DEQATN 2,
$ BUT WITH Z=0.5 (DEQATN 4). THIS TOP OUTER CIRCLE IS DEFINED BY TWO GMCURV
$ ENTRIES:
$ GMCURV 3 WITH THE RANGE OF CURVE PARAMETER,U = 0.0      TO 3.1416 RADIANS
$ GMCURV 4 WITH THE RANGE OF CURVE PARAMETER,U = 3.1415 TO 6.2832 RADIANS
$
DEQATN 4      Z1(U) = 0.5
$
GMCURV 3      MSCGRP1 0      0      +000023
++000023EQUATION,0.0,3.1416,1,2,4      +000024
GMCURV 4      MSCGRP1 0      0      +000027
++000027EQUATION,3.1415,6.2832,1,2,4      +000028
$
$ DEFINE THE BOTTOM INNER CIRCLE OF RADIUS = 4.0 AT Z=0.0 USING DEQATN 5,6,
$ AND 7. THE BOTTOM INNER CIRCLE IS DEFINED BY TWO GMCURV ENTRIES:
$ GMCURV 5 WITH THE RANGE OF CURVE PARAMETER,U = 0.0      TO 3.1416 RADIANS
$ GMCURV 6 WITH THE RANGE OF CURVE PARAMETER U = 3.1415 TO 6.2832 RADIANS
$
DEQATN 5      X(U) = 4.0*COS(U)
DEQATN 6      Y(U) = 4.0*SIN(U)
DEQATN 7      Z(U) = 0.0
$
GMCURV 5      MSCGRP1 0      0      +000029
++000029EQUATION,0.0,3.1416,5,6,7      +000030
GMCURV 6      MSCGRP1 0      0      +000033
++000033EQUATION,3.1415,6.2832,5,6,7      +000034
$
$ DEFINE THE TOP INNER CIRCLE USING THE SAME EQUATION, DEQATN 5 AND DEQATN 6,
$ BUT WITH Z=0.5 (DEQATN 8). THIS TOP INNER CIRCLE IS DEFINED BY TWO GMCURV
$ ENTRIES:
$ GMCURV 7 WITH THE RANGE OF CURVE PARAMETER,U = 0.0      TO 3.1416 RADIANS
$ GMCURV 8 WITH THE RANGE OF CURVE PARAMETER U = 3.1415 TO 6.2832 RADIANS
$
DEQATN 8      Z1(U) = 0.5
$
GMCURV 7      MSCGRP1 0      0      +000031
++000031EQUATION,0.0,3.1416,5,6,8      +000032
GMCURV 8      MSCGRP1 0      0      +000035
++000035EQUATION,3.1415,6.2832,5,6,8      +000036
$
$ DEFINE FEEDGES
$ FEEDGE BY DEFAULT IS A STRAIGHT LINE.
$ GMCURVS ARE REQUIRED FOR FEEDGES THAT DEFINE CURVED EDGES
$
$ FEEDGES 1, 2, 3 AND 4 MAKE GMCURV 1
$ FEEDGES 5, 6, 7 AND 8 MAKE GMCURV 2
$ FEEDGES 9, 10, 11 AND 12 MAKE GMCURV 3
$ FEEDGES 13, 14, 15 AND 16 MAKE GMCURV 4
$ FEEDGES 17, 18, 19 AND 20 MAKE GMCURV 5
$ FEEDGES 21, 22, 23 AND 24 MAKE GMCURV 6
$ FEEDGES 25, 26, 27 AND 28 MAKE GMCURV 7
$ FEEDGES 29, 30, 31 AND 32 MAKE GMCURV 8
$

```

Listing 27-4. Input File for the Circular Solid Model (Continued)

```

$      ID      GRID 1  GRID 2  CID      GEOMIN ID
FEEDGE 1        2        3      0      GMCURV 1
FEEDGE 2        3        4      0      GMCURV 1
FEEDGE 3        4        5      0      GMCURV 1
FEEDGE 4        5        6      0      GMCURV 1
FEEDGE 5        6        7      0      GMCURV 2
FEEDGE 6        7        8      0      GMCURV 2
FEEDGE 7        8        9      0      GMCURV 2
FEEDGE 8        9        2      0      GMCURV 2
FEEDGE 9       11       12      0      GMCURV 3
FEEDGE 10      12      13      0      GMCURV 3
FEEDGE 11      13      14      0      GMCURV 3
FEEDGE 12      14      15      0      GMCURV 3
FEEDGE 13      15      16      0      GMCURV 4
FEEDGE 14      16      17      0      GMCURV 4
FEEDGE 15      17      18      0      GMCURV 4
FEEDGE 16      18      11      0      GMCURV 4
FEEDGE 17      21      22      0      GMCURV 5
FEEDGE 18      22      23      0      GMCURV 5
FEEDGE 19      23      24      0      GMCURV 5
FEEDGE 20      24      25      0      GMCURV 5
FEEDGE 21      25      26      0      GMCURV 6
FEEDGE 22      26      27      0      GMCURV 6
FEEDGE 23      27      28      0      GMCURV 6
FEEDGE 24      28      21      0      GMCURV 6
FEEDGE 25      29      30      0      GMCURV 7
FEEDGE 26      30      31      0      GMCURV 7
FEEDGE 27      31      32      0      GMCURV 7
FEEDGE 28      32      33      0      GMCURV 7
FEEDGE 29      33      34      0      GMCURV 8
FEEDGE 30      34      35      0      GMCURV 8
FEEDGE 31      35      36      0      GMCURV 8
FEEDGE 32      36      29      0      GMCURV 8
$
$ DEFINE FEFACES FOR BOUNDARY CONDITION
$
$      ID      GRID 1  GRID 2  GRID 3  GRID 4  CID
FEFACE 1        2        3      12      11      0
FEFACE .2      3        4      13      12      0
FEFACE 3        4        5      14      13      0
FEFACE 4        5        6      15      14      0
FEFACE 5        6        7      16      15      0
FEFACE 6        7        8      17      16      0
FEFACE 7        8        9      18      17      0
FEFACE 8        9        2      11      18      0
$
$ DEFINE CONSTRAINTS IN THE Z-DIRECTION
$
$ SID      = SPC ID THAT IS DEFINED IN CASE CONTROL
$ C        = CONSTRAINT COMPONENT NUMBER IN GLOBAL COORDINATE SYSTEM.
$           IT HAS TO BE A SINGLE INTEGER (1 OR 2 OR 3) .
$ ENTITY   = ENTITY AT WHICH THE CONSTRAINT IS APPLIED
$ ID       = ID OF THE ENTITY SELECTED ABOVE
$
$      SID      C      ENTITY  ID
GMSPC 1        3      FEFACE 1
GMSPC 1        3      FEFACE 2
GMSPC 1        3      FEFACE 3
GMSPC 1        3      FEFACE 4
GMSPC 1        3      FEFACE 5
GMSPC 1        3      FEFACE 6
GMSPC 1        3      FEFACE 7
GMSPC 1        3      FEFACE 8
$

```

Listing 27-4. Input File for the Circular Solid Model (Continued)

```

$ DEFINE CONSTRAINTS IN THE Y-DIRECTION
$
GMSPC    1          2          GRID    1
$
$ DEFINE CONSTRAINTS IN THE X-DIRECTION AND Z-DIRECTION
$ FOR BOTH GRID 4 AND GRID 8. MULTIPLE GMSPC ENTRIES ARE
$ REQUIRED FOR CONSTRAINING MULTIPLE COMPONENTS.
$
GMSPC    1          1          GRID    4
GMSPC    1          3          GRID    4
GMSPC    1          1          GRID    8
GMSPC    1          3          GRID    8
$
$ DEFINE CONSTRAINTS FOR ROTATION FOR ALL GRIDS
$
GRDSET          0                      0          456
$
$ DEFINE GRIDS
$
GRID      1          0.0      0.0      0.0
GRID      2          1          5.0      0.0      0.0
GRID      3          1          5.0      45.0      0.0
GRID      4          1          5.0      90.0      0.0
GRID      5          1          5.0     135.0      0.0
GRID      6          1          5.0     180.0      0.0
GRID      7          1          5.0     225.0      0.0
GRID      8          1          5.0     270.0      0.0
GRID      9          1          5.0     315.0      0.0
GRID     10          0.0      0.0      0.5
GRID     11          1          5.0      0.0      0.5
GRID     12          1          5.0     45.0      0.5
GRID     13          1          5.0     90.0      0.5
GRID     14          1          5.0    135.0      0.5
GRID     15          1          5.0    180.0      0.5
GRID     16          1          5.0    225.0      0.5
GRID     17          1          5.0    270.0      0.5
GRID     18          1          5.0    315.0      0.5
GRID     21          1          4.0      0.0      0.0
GRID     22          1          4.0     45.0      0.0
GRID     23          1          4.0     90.0      0.0
GRID     24          1          4.0    135.0      0.0
GRID     25          1          4.0    180.0      0.0
GRID     26          1          4.0    225.0      0.0
GRID     27          1          4.0    270.0      0.0
GRID     28          1          4.0    315.0      0.0
GRID     29          1          4.0      0.0      0.5
GRID     30          1          4.0     45.0      0.5
GRID     31          1          4.0     90.0      0.5
GRID     32          1          4.0    135.0      0.5
GRID     33          1          4.0    180.0      0.5
GRID     34          1          4.0    225.0      0.5
GRID     35          1          4.0    270.0      0.5
GRID     36          1          4.0    315.0      0.5
$
$ DEFINE THE MATERIAL PROPERTIES (ALUMINUM)
$
MAT1      1          1.+7          0.3      2.538-4
PSOLID   1          1          GRID      SMECH
$
ENDDATA

```

Listing 27-4. Input File for the Circular Solid Model

This example illustrates a normal modes analysis, SOL 103, using the Lanczos method (EIGRL entry). The output requests are eigenvectors for all modes (DISPLACEMENT = ALL). Adaptivity

analysis is required for all the elements. For this purpose, both the ADAPT and PVAL entries are used in conjunction with the ADAPT command in Case Control Section.

Results

```

1  CIRCULAR PLATE                                JULY 23, 2003 NX NASTRAN 07/23/03  PAGE  929
   EDGE SIMPLY SUPPORTED
0
0
      E I G E N V A L U E  A N A L Y S I S  S U M M A R Y  (LANCZOS ITERATION)
      BLOCK SIZE USED ..... 7
      NUMBER OF DECOMPOSITIONS ..... 2
      NUMBER OF ROOTS FOUND ..... 10
      NUMBER OF SOLVES REQUIRED ..... 10
      TERMINATION MESSAGE :  REQUIRED NUMBER OF EIGENVALUES FOUND.
1  CIRCULAR PLATE                                DECEMBER 13, 1993 NX NASTRAN 07/23/03  PAGE  930
   EDGE SIMPLY SUPPORTED
0
      R E A L  E I G E N V A L U E S
      RADIANS          CYCLES          GENERALIZED          GENERALIZED
      MODE NO.        ORDER          MASS          STIFFNESS
1      1      1      3.465334E+07      5.886709E+03      9.368988E+02      1.000000E+00      3.465334E+07
2      2      2      1.182291E+08      1.087332E+04      1.730543E+03      1.000000E+00      1.182291E+08
3      3      3      1.483814E+08      1.218119E+04      1.938696E+03      1.000000E+00      1.483814E+08
4      4      4      2.502755E+08      1.582010E+04      2.517847E+03      1.000000E+00      2.502755E+08
5      5      5      2.654028E+08      1.629119E+04      2.592823E+03      1.000000E+00      2.654028E+08
6      6      6      3.027782E+08      1.740052E+04      2.769379E+03      1.000000E+00      3.027782E+08
7      7      7      8.642510E+08      2.939815E+04      4.678860E+03      1.000000E+00      8.642510E+08
8      8      8      8.645740E+08      2.940364E+04      4.679734E+03      1.000000E+00      8.645740E+08
9      9      9      1.166431E+09      3.415306E+04      5.435628E+03      1.000000E+00      1.166431E+09
10     10     10     2.009605E+09      4.482862E+04      7.134696E+03      1.000000E+00      2.009605E+09
1  CIRCULAR PLATE                                JULY 23, 2003 NX NASTRAN 07/23/03  PAGE  931
   EDGE SIMPLY SUPPORTED
0
*** USER INFORMATION MESSAGE 6502 (ADAPT)
THE CURRENT SUPERELEMENT REQUIRES NO FURTHER P-ADAPTIVE ANALYSIS BECAUSE ALL P-ELEMENTS REACHED MAXIMUM P-VALUES.
0*** USER WARNING MESSAGE 4594. THE POSTCDB CONTAINS NO LEGITIMATE SURFACE OR VOLUME DEFINITIONS.
1  CIRCULAR PLATE                                JULY 23, 2003 NX NASTRAN 07/23/03  PAGE  932
   EDGE SIMPLY SUPPORTED          ADAPTIVITY INDEX= 5
0
      EIGENVALUE =  3.465334E+07
      CYCLES =  9.368987E+02      R E A L  E I G E N V E C T O R  N O .      1
      POINT ID.    TYPE      T1      T2      T3      R1      R2      R3
1      1      G      6.661354E-06      .0      -1.868621E+01      .0      .0      .0
2      2      G      1.333745E+00      -2.678128E-05      .0      .0      .0      .0
3      3      G      9.431428E-01      9.430584E-01      .0      .0      .0      .0
4      4      G      .0      1.333743E+00      .0      .0      .0      .0
5      5      G      -9.430543E-01      9.431386E-01      .0      .0      .0      .0
6      6      G      -1.333741E+00      2.663938E-05      .0      .0      .0      .0
7      7      G      -9.431665E-01      -9.430221E-01      .0      .0      .0      .0
8      8      G      .0      -1.333743E+00      .0      .0      .0      .0
9      9      G      9.430305E-01      -9.431752E-01      .0      .0      .0      .0
10     10     G      6.681371E-06      -6.129073E-08      -1.868621E+01      .0      .0      .0
11     11     G      -1.333732E+00      -3.440074E-04      .0      .0      .0      .0
12     12     G      -9.428723E-01      -9.433200E-01      .0      .0      .0      .0
13     13     G      3.471338E-04      -1.333743E+00      .0      .0      .0      .0
14     14     G      9.433249E-01      -9.428771E-01      .0      .0      .0      .0
15     15     G      1.333754E+00      3.440111E-04      .0      .0      .0      .0
16     16     G      9.429217E-01      9.432840E-01      .0      .0      .0      .0
17     17     G      -3.258019E-04      1.333743E+00      .0      .0      .0      .0
18     18     G      -9.432753E-01      9.429132E-01      .0      .0      .0      .0
21     21     G      1.343029E+00      -8.329611E-05      -5.443803E+00      .0      .0      .0
22     22     G      9.497321E-01      9.495923E-01      -5.443810E+00      .0      .0      .0
23     23     G      8.442988E-05      1.343021E+00      -5.443810E+00      .0      .0      .0
24     24     G      -9.495862E-01      9.497259E-01      -5.443810E+00      .0      .0      .0
25     25     G      -1.343013E+00      8.315749E-05      -5.443817E+00      .0      .0      .0
26     26     G      -9.497337E-01      -9.495776E-01      -5.443811E+00      .0      .0      .0
27     27     G      -7.570248E-05      -1.343021E+00      -5.443810E+00      .0      .0      .0
28     28     G      9.495844E-01      -9.497408E-01      -5.443809E+00      .0      .0      .0
29     29     G      -1.343016E+00      -2.135180E-04      -5.443803E+00      .0      .0      .0
30     30     G      -9.495108E-01      -9.498014E-01      -5.443810E+00      .0      .0      .0
31     31     G      2.196022E-04      -1.343021E+00      -5.443810E+00      .0      .0      .0
32     32     G      9.498075E-01      -9.495170E-01      -5.443810E+00      .0      .0      .0
33     33     G      1.343026E+00      2.135190E-04      -5.443817E+00      .0      .0      .0
34     34     G      9.495386E-01      9.497868E-01      -5.443811E+00      .0      .0      .0
35     35     G      -2.027499E-04      1.343021E+00      -5.443810E+00      .0      .0      .0
36     36     G      -9.497795E-01      9.495314E-01      -5.443809E+00      .0      .0      .0

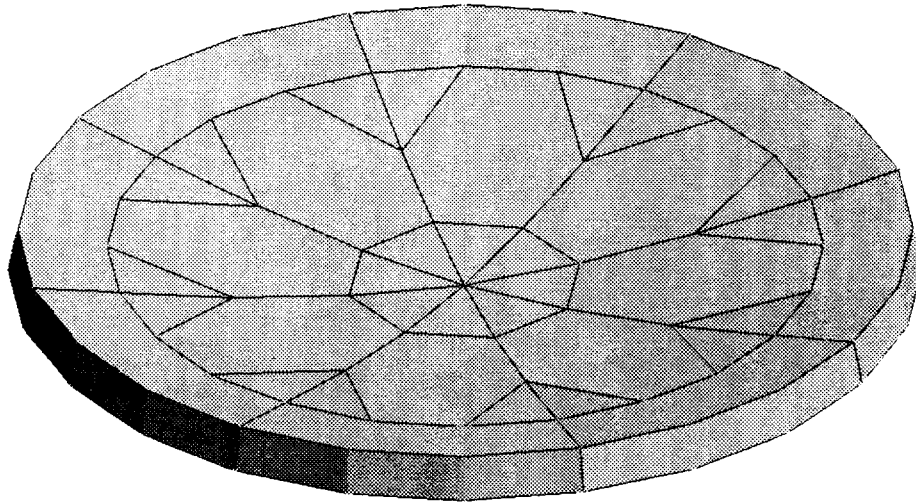
```

Figure 27-13. Abridged Output from the Circular Solid Model (Continued)

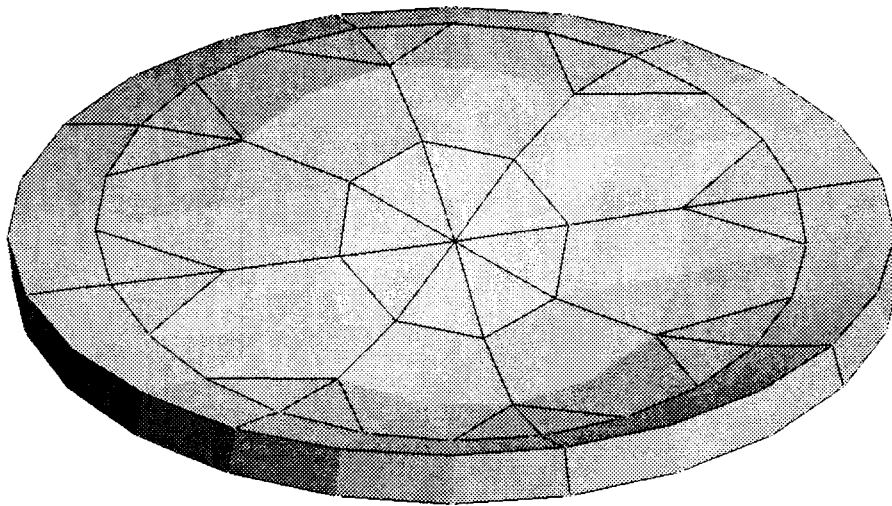
201001001	G	1.333745E+00	-2.678128E-05	.0	.0	.0	.0
201001002	G	1.288204E+00	3.459260E-01	.0	.0	.0	.0
201001003	G	1.155554E+00	6.657130E-01	.0	.0	.0	.0
201001004	G	9.431428E-01	9.430584E-01	.0	.0	.0	.0
201001005	G	1.354012E+00	-1.118739E-07	-1.801449E+00	.0	.0	.0
201001006	G	1.307760E+00	3.511871E-01	-1.801487E+00	.0	.0	.0
201001007	G	1.173140E+00	6.758680E-01	-1.801314E+00	.0	.0	.0
201001008	G	9.574580E-01	9.574031E-01	-1.801455E+00	.0	.0	.0
201001009	G	1.357469E+00	-5.671428E-05	-3.626504E+00	.0	.0	.0
201001010	G	1.311071E+00	3.521154E-01	-3.626570E+00	.0	.0	.0
201001011	G	1.176151E+00	6.776471E-01	-3.626329E+00	.0	.0	.0
201001012	G	9.599283E-01	9.598196E-01	-3.626513E+00	.0	.0	.0
201001013	G	1.343029E+00	-8.329611E-05	-5.443803E+00	.0	.0	.0
201001014	G	1.297096E+00	3.483891E-01	-5.443859E+00	.0	.0	.0
201001015	G	1.163644E+00	6.704805E-01	-5.443617E+00	.0	.0	.0
201001016	G	9.497321E-01	9.495923E-01	-5.443810E+00	.0	.0	.0

Figure 27-13. Abridged Output from the Circular Solid Model

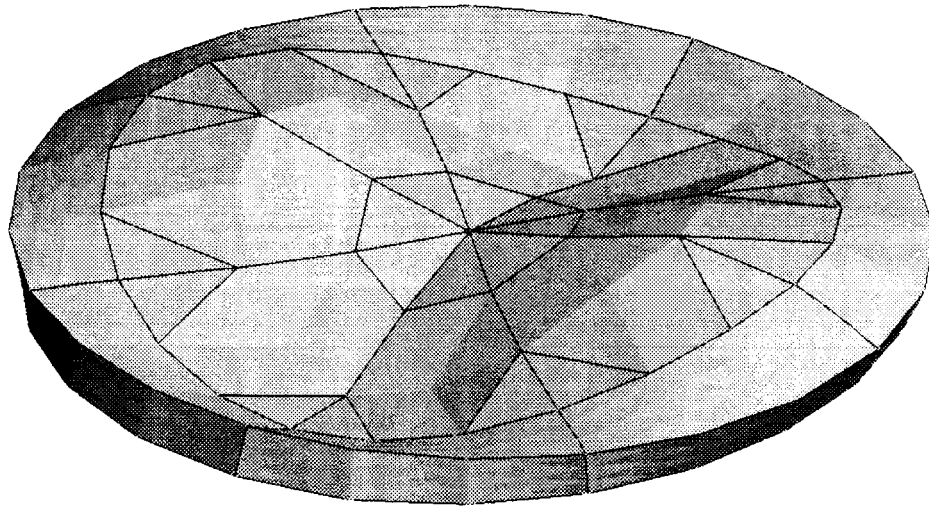
The mode shapes for modes 1, 5, 8, and 9 are shown in [Figure 27-14](#). Mode 1 is the fundamental mode, mode 5 is the one nodal diameter mode, mode 8 is the two nodal diameters mode, and mode 9 is one nodal circle mode.



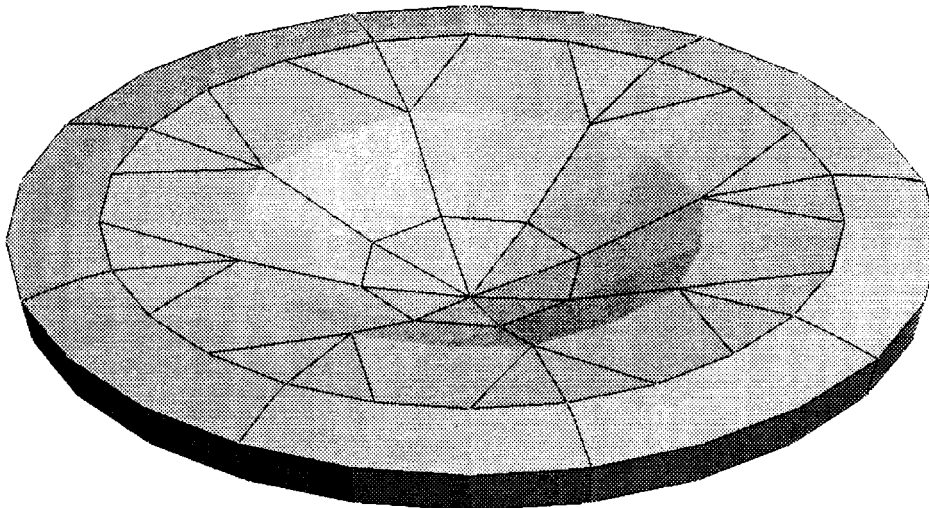
Mode 1: Fundamental Mode



Mode 5: One Nodal Diameter Mode



Mode 8: Two Nodal Diameter Mode



Mode 9: One Nodal Circle Mode

Figure 27-14. The Mode Shapes for Modes 1, 5, 8, and 9

Figure 27-14 The Mode Shapes for Modes 1, 5, 8, and 9 (continued)

Comparison of the natural frequencies obtained from the adaptivity analysis for the solid circular model to the theoretical natural frequencies for the circular plate is shown in **Table 27-7**. The discrepancies between the two values decrease as the p-order increases.

Table 27-7. Comparison of the Natural Frequencies		
Mode	Theoretical (Hertz)	Numerical (Hertz)
Fundamental	954.1	936.9
One Nodal Diameter	2657.7	2592.8
Two Nodal Diameters	4913.9	4679.7
One Nodal Circle	5697.8	5435.6

Further studies were performed in which the p-order was kept constant and the thickness of the solid model was varied. Results showed that as the thickness of the circular solid model decreases, the solid model behaves like a plate model, which is to be expected.

Reference

[1] Roark, R. J. and Young, W. C. *Formulas for Stress and Strain*. 5th edition. McGraw-Hill, Inc., 1975, p. 578.

Clamped Rhombic Plate

The first eigenfrequency of a rhombic plate is calculated using SOL 103. [Figure 27-15](#) shows a plate clamped at the left end and the first eigenmode. Compare the convergence behavior of three different models:

- One CHEXA p-element with p-orders varying from $p = 1$ to $p = 8$
- One CQUAD p-element with p-orders varying from $p = 1$ to $p = 8$
- Multiple CQUAD h-elements with varying meshes (1×2 , 2×2 , 4×4 , 8×8 , 16×16 , 32×32)

The convergence behavior of the three models is shown in [Figure 27-16](#). The first eigenfrequency is normalized with 2.932 rad/sec, which is the value from a series solution using Kirchhoff plate theory. All three models exhibit excellent convergence behavior.

The solid p-element and the shell p-element models converge when the p-level is increased. The CQUAD4 h-element model converges when the mesh is refined. If the error rate in the first eigenfrequency must be under 0.2%, then either: (a) a single shell or solid p-element with $p = 8$, or (b) 16×16 shell h-elements should be used. The shell p-element needs the smallest number of degrees-of-freedom to achieve the required accuracy. For a comparison of the degrees-of-freedom, see [Table 27-8](#).

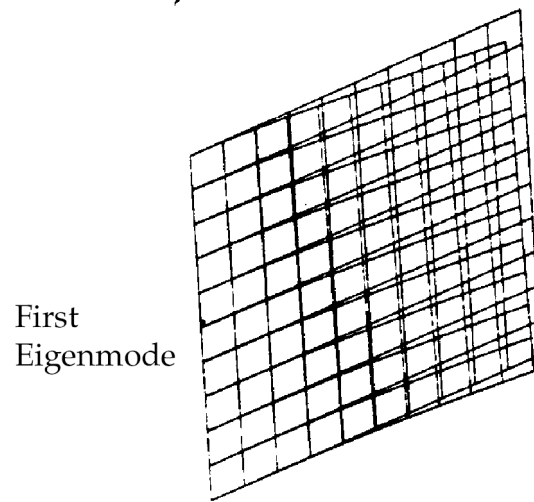
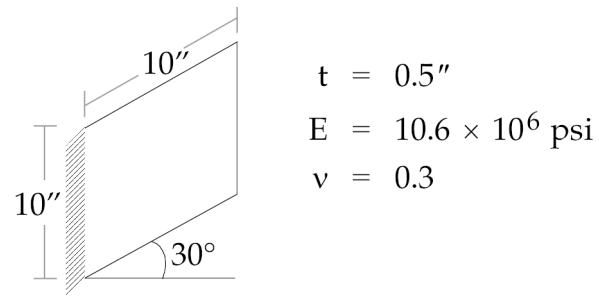


Figure 27-15. Rhombic Plate

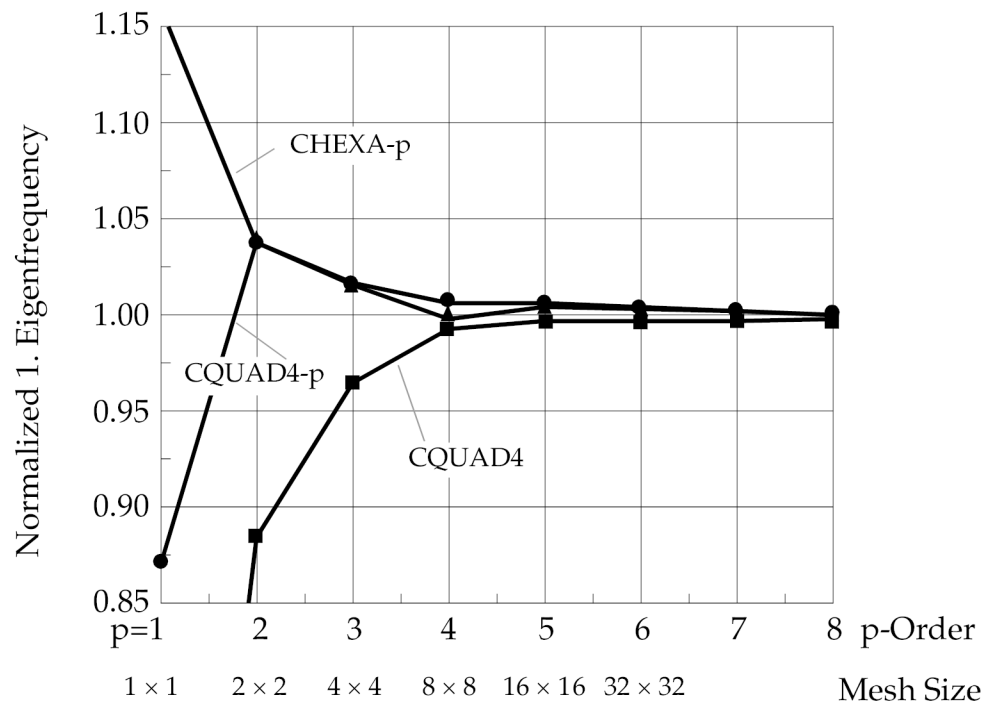


Figure 27-16. Normalized First Eigenfrequency for Different p-Levels and Mesh Sizes

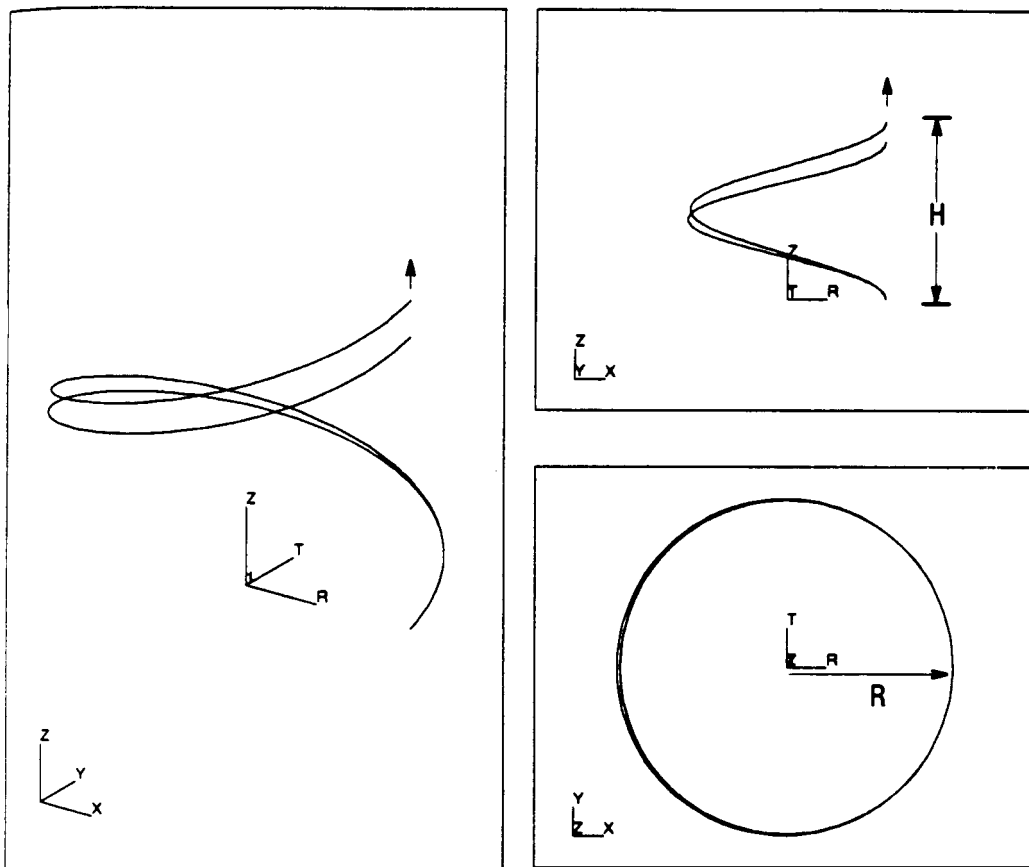
Table 27-8. Comparison of the Number of Degrees-of-Freedom for 0.2% Accuracy in the First Eigenfrequency

Element	Mesh Size and p-Level	Normalized Eigenfrequency	Degrees of Freedom
CQUAD4	16 × 16	0.9977	1445
CQUAD4-p	p = 8	1.0012	264
CHEXA-p	p = 8	1.0017	510

Helix Modeled with CBEAM p-Elements

A 360° helix is modeled with nonadaptive beam h-elements and with adaptive beam p-elements. The helix is clamped at the bottom and free at the top, with a single force applied at the top. The undeformed and deformed helix is shown in [Figure 27-17](#). The normalized deflection at the tip is shown in [Figure 27-18](#) for different mesh sizes.

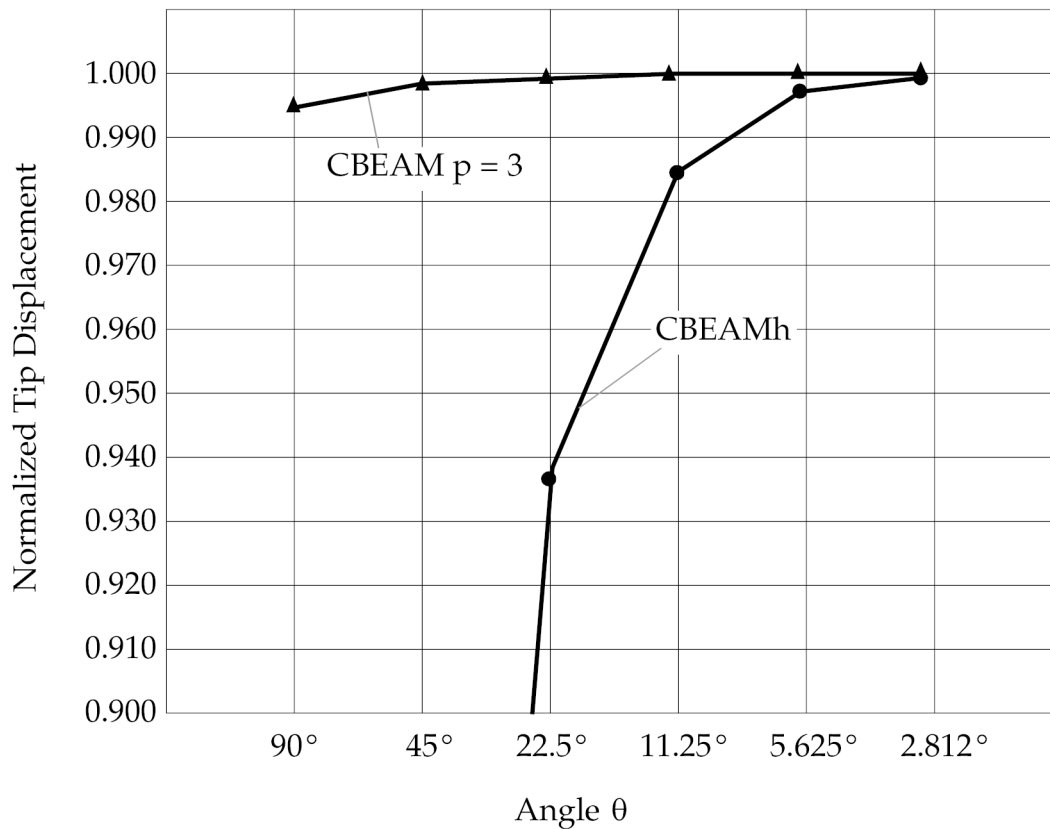
Both beam h- and p-elements converge well. If the error in the deflection must be under 0.2%, then 8 beam p-elements with $p = 3$ (45° angle for each element) or 64 beam h-elements should be used. A comparison of the degrees-of-freedom (see [Table 27-9](#)) shows that the beam p-elements require a smaller number of degrees-of-freedom than the number required by the beam h-elements in order to achieve the required accuracy.



$\theta = 360^\circ$
 $H = 9.$ height
 $R = 5.$ radius

CHAN1 cross-section
 $E = 1.5e+7$
 $\nu = 0.33$

Figure 27-17. Helix



Tip displacement normalized with $u_z = 5.930e-3$

Figure 27-18. Normalized Tip Displacement for Different Mesh Sizes

Table 27-9. Comparison of the Number of degrees-of-freedom for 0.2% Accuracy in the Tip Displacement

Element and Mesh	Normalized Tip Displacement	Degrees of Freedom
CBEAM h, $\theta = 5.6^\circ$; 64 Elements	0.9970	390
CBEAM p, $\theta = 45^\circ$; $p = 3$, 8 Elements	0.9978	150

Plate with Stiffener

In stiffened structures, an efficient practice is to model the stiffeners using beam elements with offsets. Typically, nonadaptive h-elements are used. The example below demonstrates the benefits of p-elements for this class of problems.

A plate with stiffener is loaded in the middle and simply supported at the ends (see [Figure 27-19](#)). The plate is modeled with 10×2 shell elements and the stiffener is modeled with 10 beam elements with offsets. Compare the stress results of two models, an h-element model and a p-element model, with a uniform p-order of $p = 2$ for shells and beams. The solution is compared with the solution derived from beam theory.

The nonadaptive h-elements produce very inaccurate stresses with errors around 30%, as shown in [Figure 27-20](#). The adaptive p-elements with a uniform p-order of $p = 2$ yield very accurate results—the error rate is only around 4%. In the h-element model, the normal forces in the shell and beam elements are constant within each element, which causes the high stress error. In the p-element model, the normal forces in each element are linear ($p = 2$), which improves the stress distribution in the cross-section of the beam.

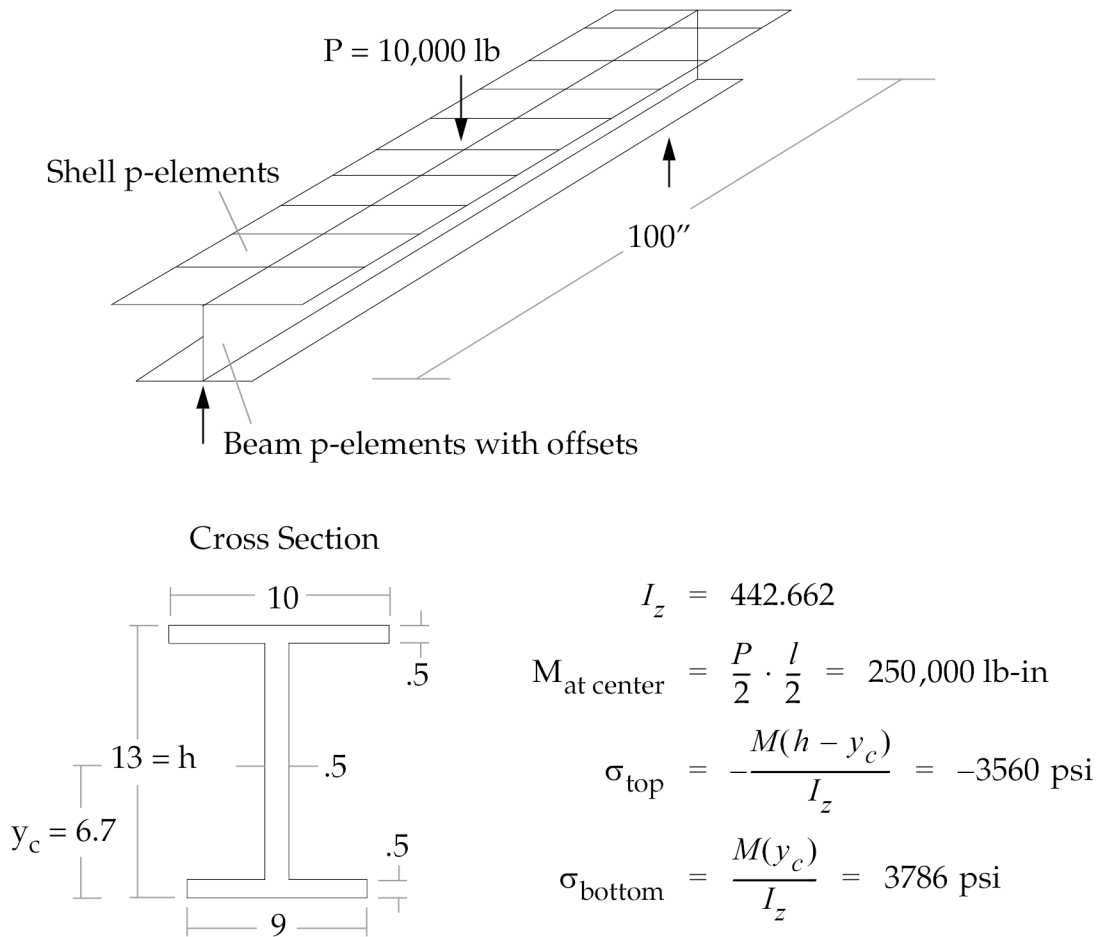
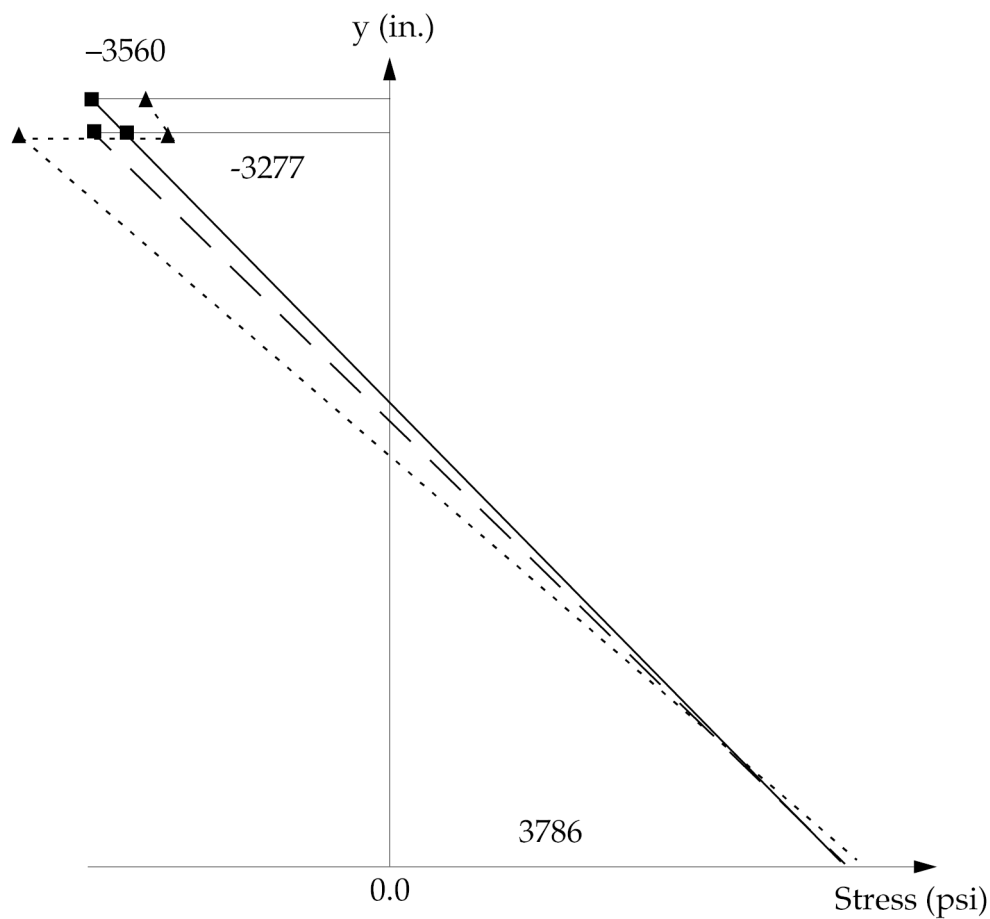


Figure 27-19. Plate with Stiffener



Stress [psi]	y [in]
3786.	0.
-3277.	12.5
-3560.	13.0

beam theory

—

Stress [psi]	y [in]
3978.	0.
-4311.	12.5
-2804.	12.5
-3082.	13.0

h-elements

▲ — — — — — ▲

Stress [psi]	y [in]
3834.	0.
-3545.	12.5
-3250.	12.5
-3590.	13.0

p=elements

p = 2

■ — — — — — ■

Figure 27-20. Stress Distribution in the Middle of the Beam

27.9 Solid p-Elements in Linear Dynamic Solution Sequences

Solid p-elements are available in the dynamic solution sequences SOL 107, to SOL 112. The p-elements do not adapt; the analysis is done for a fixed p-level. Displacements, velocities, accelerations, stresses and strains are output in SORT1 or SORT2 format. The complex output quantities are in rectangular format (real/imaginary) or in polar format (magnitude/phase).

User Interface in NX Nastran

Output for p-elements is requested with the Case Control command DATAREC, where SORT1 or SORT2 formats can be chosen. For complex eigenvalues (SOL 107, SOL 110) and frequency response (SOL 108, SOL 111), the default is SORT1. For transient response (SOL 109, SOL 112), the default is SORT2.

The output quantities are requested with DISP, VELO, ACCE, STRESS, or STRAIN in the Case Control, and/or in the OUTPUT Bulk Data entry. In the complex solution sequences, the formats are rectangular (REAL, IMAG) or polar (MAGNITUDE, PHASE); the rectangular format is the default. REAL, IMAG or PHASE is requested in the Case Control command or in the OUTPUT Bulk Data entry.

Limitations and Recommendations

The following limitations and recommendations apply to solid p-element usage in linear dynamic solution sequences:

- The output request must be all in SORT1 or all in SORT2 format for a single run.
- PARAM,CURVPLOT,1 must be specified to get SORT1 output in transient response (SOL 109 and SOL 112).
- If an ADAPT entry is specified with multiple p-levels, the analysis is done with the first p-level.

Example

The transient response of a cantilever subject to a constant step load is calculated in SOL 109 (see [Figure 27-21](#)). The following is a comparison of the results between the two different models:

- HEXA8 h-elements, $4 \times 4 \times 40$ element mesh.
- HEXA p-elements, $1 \times 1 \times 4$ element mesh with $p = 3$.

The plot displacement versus time graph shown in [Figure 27-22](#) illustrates that the results are in good agreement.

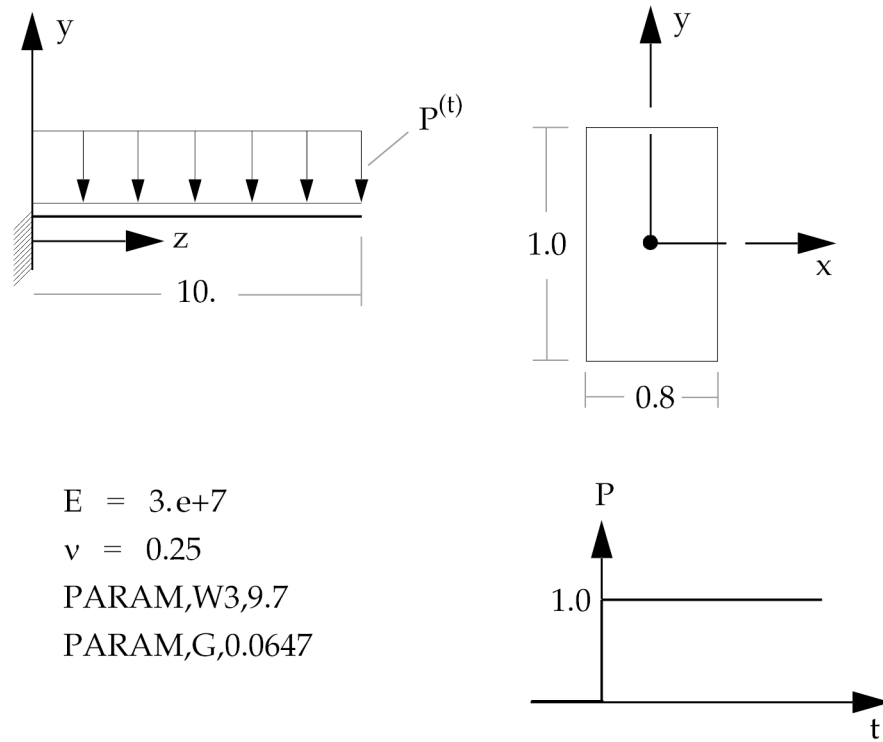


Figure 27-21. Cantilever with Step Load

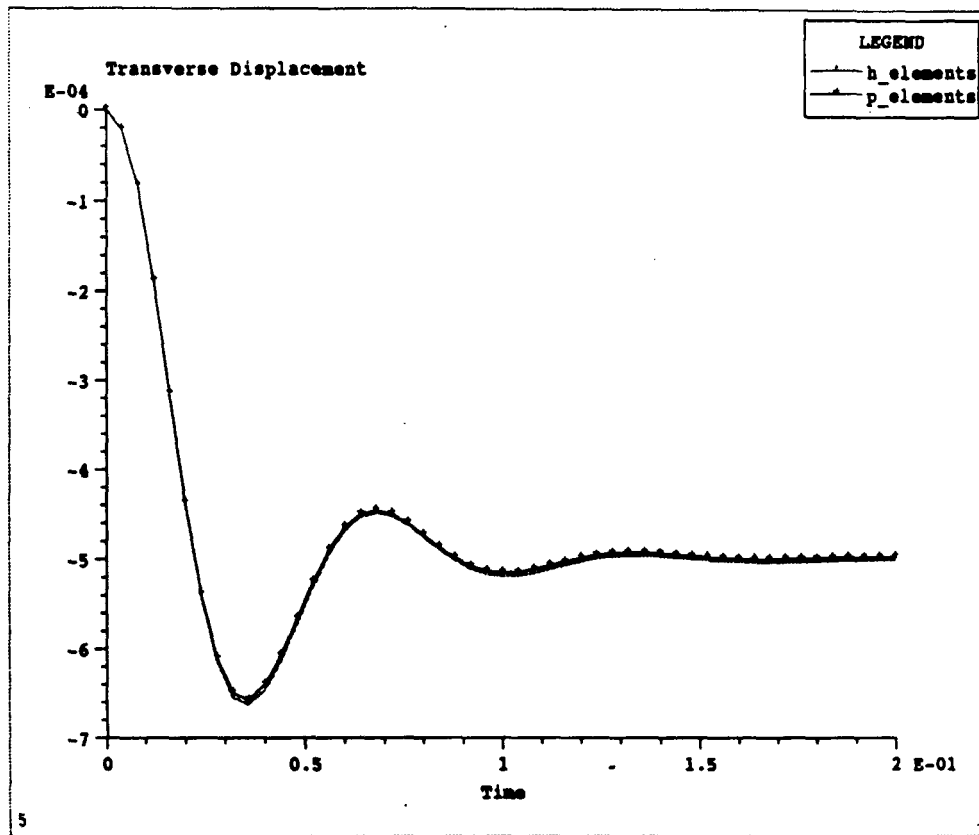


Figure 27-22. Tip Displacement of Cantilever

27.10 p-Element Thermal Analysis

Introduction

NX Nastran supports p-version elements for linear steady state thermal analysis (SOL 101). The definitions of model geometry, elements, p-values, and error estimation methods follow the general terminologies of the p-element structural analysis. A major difference exists in the specification of thermal loads and boundary conditions, which requires new or modified Bulk Data entries for thermal analysis.

The following sections provide usage guidelines and examples for the p-version thermal analysis.

Geometry

The geometry definitions are the same as those defined for the p-element structural analysis.

Elements

Six kinds of elements—CHEXA, CPENTA, CTETRA, CQUAD4, CTRIA3, and CBEAM—may be defined as p-elements for thermal analysis.

Materials

The standard isotropic, orthotropic, and anisotropic materials are available for p-version thermal elements. The only material property required is thermal conductivity which must be constant for linear steady state analysis.

Loads and Boundary Conditions

The following thermal loads and boundary conditions are supported in p-version thermal analysis.

- Prescribed temperatures applied on point, curve, 2D edge, surface, or 3D surface.
- Normal or directional heat fluxes applied on point, curve, 2D edge, surface, 3D surface, or solid.
- Volumetric heat generation within conduction elements.
- Free convection applied on point, curve, 2D edge, surface, or 3D surface.

Table 27-10 summarizes the Bulk Data entries for p-version thermal loads and boundary conditions. A full description of these entries may be found in the *NX Nastran Quick Reference Guide*.

Table 27-10. Bulk Data Entries for p-Version Thermal Loads and Boundary Conditions	
Load or Boundary Condition Type	Bulk Data
Prescribed Temperature	GMBC
Normal or Directional Heat Flux	GMLOAD
Volumetric Heat Generation	GMQVOL
Free Convection	GMCONV

p-Version Adaptivity

p-values and error estimation methods are the same as those for p-element structural analysis.

Data Recovery

Similar to structural output, the DATAREC and OUTRCV Case Control commands as well as the OUTPUT and OUTRCV Bulk Data entries define the output of p-version thermal analysis. Temperatures, temperature gradients, heat fluxes, polynomial values, and element errors may be printed, plotted, or punched. The TEMP output command of the OUTPUT Bulk Data entries requests for calculating temperatures while the FLUX output command of the OUTPUT Bulk Data entries requests for recovering temperature gradients and heat fluxes for conduction elements.

Thermal Stress Analysis

The interface to input temperature results as temperature loads for thermal stress analysis is described in the Case Control command “**TSTRU**” in the *NX Nastran Quick Reference Guide*.

Limitations and Recommendations

- Only Solution Sequence 101 (linear steady state) performs p-element thermal analysis.

- Temperature boundary conditions can be spatially varying, but must be independent of time.
- Heat fluxes and volumetric heat generation can be spatially varying, but must be independent of temperature or time.
- The convection heat transfer coefficients and ambient temperatures of free convection boundary conditions can be spatially varying, but must be independent of temperature or time.
- It is important to select the appropriate entity type while applying loads and boundary conditions. Concentrated loads or boundary conditions will cause singularities or inaccurate results and should be avoided. For example, the entity type FEFACE or GMSURF should be used to prescribe temperatures on a surface, rather than the entity type FEEDGE, GMCURVE, or GRID.

Siemens Industry Software

Headquarters

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe

Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific

Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

© 2014 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.