

Simcenter Nastran Basic Nonlinear Analysis User's Guide

Contents

Proprietary & Restricted Rights Notice	7
Introduction to Nonlinear Analysis	1-1
Overview of Nonlinear Analysis	1-1
Creating an Input File for a Basic Nonlinear Analysis	1-2
Case Control Section	1-4
Bulk Data Section	1-5
Parameters	1-7
Nonlinear Element Output Codes for Plotting	1-10
Nonlinear Characteristics and General Recommendations	1-10
Elements for Nonlinear Analysis	2-1
Overview of Nonlinear Elements	2-1
Physical Elements	2-1
Contact Elements	2-9
Small Strain Elements	2-10
User Interface	2-11
Output	2-12
Co-rotational Formulation in Geometric Nonlinear Analysis	2-12
Guidelines	2-13
Limitations	2-13
Fully Nonlinear Hyperelastic Elements	2-14
User Interface	2-20
Output Description	2-20
Guidelines	2-23
Limitations	2-23
Example 1: Rubber Bushing Problem	2-25
Example 2: Pressure of a Circular Plate	2-28
Bushings Elements	2-31
Swinging Pendulum	2-32
Adaptive Gap Element	2-35
Usage	2-35
Limitations	2-35
How to Use the CGAP and PGAP Bulk Data Entries	2-35
Output	2-36
Example: Hertz Contact Stress	2-37
Material and Geometric Nonlinearity	3-1
Overview of Material Nonlinearity	3-1
Nonlinear Elastic Material	3-1
Defining Nonlinear Elastic Materials in Simcenter Nastran	3-2
Hyperelastic Material	3-2

Special Cases of the Material Law	3-4
Defining Hyperelastic Materials in Simcenter Nastran	3-4
Using the Curve Fitting Algorithm to Determine Material Constants	3-5
Guidelines for Using Hyperelastic Materials	3-6
Limitations of Using Hyperelastic Materials	3-7
Example: Curve-Fitting Algorithm	3-7
Elastic-Plastic Material	3-10
Yield Function	3-11
Defining Elastic-Plastic Materials in Simcenter Nastran	3-15
Grid Point Stresses in Solid Elements	3-16
Viscoelastic Material in Nonlinear Analysis	3-16
Temperature-Dependent Material	3-17
Understanding Geometric Nonlinearity	3-18
Loads for Nonlinear Analysis	4-1
Overview of Nonlinear Static Loads	4-1
Concentrated Loads	4-1
Distributed Loads	4-1
Mass Related Loads (GRAV and RFORCE Entries)	4-2
Thermal Loads	4-2
Load Combinations	4-3
Overview of Nonlinear Transient Loads	4-4
NLRGAP	4-7
Nonlinear Analysis Types	5-1
Overview of Nonlinear Analysis Types	5-1
3-D Slide Line Contact Analysis	5-1
Diagnostic Output	5-5
Limitations	5-5
Defining 3D Slideline Contact in the Input File	5-5
3D Slideline Contact Example	5-6
Nonlinear Static Analysis	5-11
Case Control	5-11
Restrictions and Limitations in SOL 106	5-12
Controlling Iterations with the NLPARM Entry	5-13
Examples of Convergence Tolerances	5-16
Nonlinear Buckling Analysis	5-25
Example (NLSOLIDB.DAT)	5-27
Nonlinear Modal Analysis	5-30
Example (TURBINE1.DAT)	5-30
Restarting from SOL 106 into SOL 103 or into Another Linear Solution Sequence	5-31
Differential Stiffness and Follower Force Stiffness in Linear and Nonlinear Solution Sequences	5-32
Nonlinear Transient Response Analysis	5-33
Case Control	5-33
Limitations of Nonlinear Transient Response Analysis	5-34
Direct Time Integration	5-34
Implicit Integration Control: TSTEPNL Data Entry	5-35
Iteration Related Output Data	5-39

Restarts	5-41
Nonlinear Analysis With Superelements	5-46
Grid Point Force Balance and Element Strain Energy in Nonlinear Static Analysis	5-46

Proprietary & Restricted Rights Notice

© 2019 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. Simcenter 3D 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. Simcenter 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.

HDF5 (Hierarchical Data Format 5) Software Library and Utilities Copyright 2006-2016 by The HDF Group

NCSA HDF5 (Hierarchical Data Format 5) Software Library and Utilities Copyright 1998-2006 by the Board of Trustees of the University of Illinois. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted for any purpose (including commercial purposes) provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or materials provided with the distribution.
3. In addition, redistributions of modified forms of the source or binary code must carry prominent notices stating that the original code was changed and the date of the change.
4. All publications or advertising materials mentioning features or use of this software are asked, but not required, to acknowledge that it was developed by The HDF Group and by the National Center for Supercomputing Applications at the University of Illinois at Urbana-Champaign and credit the contributors.
5. Neither the name of The HDF Group, the name of the University, nor the name of any Contributor may be used to endorse or promote products derived from this software without specific prior written permission from The HDF Group, the University, or the Contributor, respectively.

DISCLAIMER: THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS "AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED.

In no event shall The HDF Group or the Contributors be liable for any damages suffered by the users arising out of the use of this software, even if advised of the possibility of such damage

Chapter 1: Introduction to Nonlinear Analysis

1.1 Overview of Nonlinear Analysis

This book describes the basic nonlinear capabilities which are available in Solution Sequences 106 and 129.

- For theoretical information on basic nonlinear capabilities, see the *Simcenter Nastran Handbook of Nonlinear Analysis*.
- For information on nonlinear heat transfer capabilities (Solution Sequences 153 and 159), see the *Simcenter Nastran Thermal Analysis User's Guide*.
- For information on advanced nonlinear capabilities (Solution Sequence 601), see the *Simcenter Nastran Solution 601 Theory and Modeling Guide*.

Nonlinear effects in structures occur mainly due to nonlinear materials and large deformations (rotations and/or strains). Contact problems exhibit nonlinear effects due to changes in boundary conditions. All of these attributes may be represented by Simcenter Nastran nonlinear elements.

The nonlinear element library consists of:

- CBUSH and CBUSH1D for generalized nonlinear springs and dampers.
- CROD, CONROD, and CTUBE for unidirectional truss members.
- CBEAM for axially and laterally deforming line members.
- CQUAD4 and CTRIA3 for membrane, plate and shell modeling and hyperelastic plane strain.
- CQUAD8, CQUAD, and CTRIA6 for hyperelastic plane strain.
- CQUADX4, CQUADX8, CTRAX3, and CTRAX6 for hyperelastic axisymmetric modeling.
- CHEXA, CPENTA, CPYRAM, and CTETRA for solid modeling.
- CGAP and slideline contact (BCONP, BLSEG, BFRIC, BWIDTH) for contact and friction modeling.

Nonlinear elements may be combined with linear elements for computational efficiency if the nonlinear effects can be localized. Primary operations for nonlinear elements are updating element coordinates and applying loads for large displacements, and the internal relaxation iteration for material nonlinearity. Refer to the table [Physical Elements for Nonlinear Analysis –Small Strain Elements](#) for a summary of the nonlinear element properties.

The geometric nonlinearity becomes discernible when the structure is subjected to large displacement and rotation. Geometric nonlinear effects are prominent in two different aspects: geometric stiffening due to initial displacements and stresses, and follower forces due to a change in loads as a function of displacements. These effects are included, but the large deformation effect resulting in large

strains is implemented only for hyperelastic materials. Large strain effects are not available for elastic-plastic, creep, or any other material types.

Material nonlinearity is an inherent property of any engineering material. Material nonlinear effects may be classified into many categories and those included in Simcenter Nastran are plasticity, nonlinear elasticity, hyperelasticity, creep, and viscoelasticity. Creep and viscoelasticity, implemented as a generalized nonlinear viscoelastic capability, may be coupled with plasticity. Many options are available for yield criteria and hardening behavior in plasticity.

The primary solution operations are gradual load or time increments, iterations with convergence tests for acceptable equilibrium error, and stiffness matrix updates. The iterative process is based on the modified-Newton's method combined with optional expeditious methods such as the quasi-Newton (BFGS) update and the line search. The stiffness matrix updates are performed occasionally to improve the computational efficiency, and may be overridden at your discretion. A number of options for arc-length methods are also available for snap-through or post-buckling analysis of static problems. For transient response analysis, a number of options are available for implicit direct time integration, combined with adaptive and expedient iteration strategies similar to those implemented for static analysis.

Solution Sequences 106, 129, 153, and 159 consolidate all the nonlinear features described above. SOLs 106 and 153 are applicable to static, quasi-static, and nonlinear buckling analyses. SOLs 129 and 159 are primarily applicable to dynamic transient response analysis with some limited static analysis capability. SOLs 153 and 159 may also be used to perform a nonlinear heat transfer analysis. See the *Simcenter Nastran Thermal Analysis User's Guide*. These solution sequences can accommodate superelements and provide easy restarts from the database.

1.2 Creating an Input File for a Basic Nonlinear Analysis

The input file consists of an optional NASTRAN statement, Executive Control Section, Case Control Section, and Bulk Data Section. All the features and principles for the user interface are common in Simcenter Nastran; thus, all the features of the user interface for nonlinear analysis are compatible with those for the linear analysis. Any exceptions for nonlinear analysis will be explained in the relevant sections that follow.

Mechanical design is dictated by the strength, dynamic, and stability characteristics of the structure. Simcenter Nastran provides the analysis capabilities of these characteristics with solution sequences, each of which is designed for specific applications. The type of desired analysis is specified in the Executive Control Section by using a solution sequence identification. SOLs 106 and 153 are designed for static, quasi-static, and buckling analyses. SOLs 129 and 159 provide nonlinear transient response analysis.

The basic input data required for a finite element analysis may be classified as follows:

- Geometric data
- Element data
- Material data
- Boundary conditions and constraints
- Loads and enforced motions

- Solution methods

The first three classes of data may not be changed during the course of an analysis whereas the last three classes of data may be changed during the analysis as defined in the Case Control Section. Examples of input data are given in [Listing 1-1](#) and [Listing 1-2](#) for nonlinear static and dynamic analysis, respectively.

```

SOL 106                $ NONLINEAR STATIC ANALYSIS
DIAG 8,50              $ DIAGNOSTIC PRINTOUT
CEND                   $ END OF EXECUTIVE CONTROL DATA
TITLE = TEST OF CTETRA ELEMENT (CUBE SUBJECT TO UNIAXIAL LOADING)
      DISP = ALL
      STRESS = ALL
      SPC = 100
SUBCASE 1
  SUBTITLE = ELASTIC -- LOAD TO 850. PSI
  LABEL = LOAD TO YIELD
  LOAD = 50
  NLPARM = 50
SUBCASE 2
  SUBTITLE = PLASTIC -- LOAD TO 1000. PSI
  LABEL = LOAD BEYOND YIELD
  LOAD = 100
  NLPARM = 100
SUBCASE 4
  SUBTITLE = ELASTIC -- UNLOAD COMPLETELY TO 0. PSI
  LABEL = FULL UNLOAD
  LOAD = 200
  NLPARM = 200
$OUTPUT(PLOT) $ PLOT DATA ADDED FOLLOWING THIS LINE IF NEEDED
$ END OF CASE CONTROL DATA
BEGIN BULK
$ PARAMETERS FOR NONLINEAR ITERATION
NLPARM      50      1      AUTO      UPW      NO
NLPARM      100     8      SEMI      UPW      NO
NLPARM      200     2      AUTO      UPW      NO
.
.
.
ENDDATA

```

Listing 1-1. Example Input Data for Nonlinear Static Analysis

```

SOL 129                $ NONLINEAR TRANSIENT ANALYSIS
DIAG 8,50              $ DIAGNOSTIC PRINTOUT
CEND                   $ END OF EXECUTIVE CONTROL DATA
TITLE = GAP VERIFICATION BY NONLINEAR TRANSIENT ANALYSIS SEALL = ALL $ FOR ALL MATRIC
      SET 1 = 201      $ ,202,200
      SET 3 = 22      $ ,23
      SET 4 = 20      $ CONROD
      DISP = 1
      STRESS = 3      $ FOR GAP FORCES
      ELFORCE = 4
LOADSET = 20
`SPC = 100

```

```

SUBCASE 1                $ UP TO 0.5 SECONDS
  DLOAD = 210
  TSTEPNL = 22
SUBCASE 2                $ UP TO 1 SECOND
  DLOAD = 210
  TSTEPNL = 21
OUTPUT (XYPLOT)
  CSCALE = 1.3
  XTITLE = TIME IN SECONDS
  YTITLE = DISPLACEMENT IN INCHES
  YMIN = 0.
  YMAX = 15.
  XYPLOT DISP RESP/201 (T1) , 202 (T1) , 200 (T1)
$ END OF CASE CONTROL DATA
BEGIN BULK
TSTEPNL      21      200      .0025      2      AUTO      10
.
.
.
ENDDATA

```

Listing 1-2. Example Input Data for Nonlinear Dynamic Analysis

1.3 Case Control Section

The Case Control Section is used to define input data sets, output features, and subcases. The subcase structure in Simcenter Nastran provides a unique means of changing loads, boundary conditions, and solution methods by making selections from the Bulk Data.

In SOL 106 and SOL 129, loads and solution methods may change from subcase to subcase on an incremental basis. However, constraints can be changed from subcase to subcase only in the static solution sequence. As a result, the subcase structure determines a sequence of loading and constraint paths in a nonlinear analysis. The subcase structure also lets you select and change output requests for printout, plot, etc., by specifying set numbers with keywords. Any selections made above the subcase specifications are applicable to all the subcases. Selections made in an individual subcase supersede the selections made above the subcases. [Table 1-1](#) summarizes the Case Control commands for nonlinear analysis.

Table 1-1. Summary of Nonlinear Case Control Commands	
Load Selection	
LOAD	Selects static loading condition.
CLOAD	Selects static load combination for superelements.
DLOAD	Selects dynamic loading conditions.
LOADSET	Selects static load sets defined on the Bulk data LSEQ.
NONLINEAR	Selects nonlinear loading (NOLINi) for transient response.
IC	Selects initial conditions for transient response.
Solution Method Selection	
METHOD	Selects methods for eigenvalue analysis.

Table 1-1. Summary of Nonlinear Case Control Commands

NLPARM	Selects iteration methods for nonlinear static analysis.
TSTEPNL	Selects iteration methods for nonlinear transient analysis.
Output Requests	
DISPLACEMENT	Requests output for displacements of physical points.
VELOCITY	Requests output for velocities of physical points.
ACCELERATION	Requests output for acceleration of physical points.
ELFORCE	Requests output for element forces.
STRESS	Requests output for element stresses.
BOUTPUT	Requests output for sideline contact.
SPCFORCES	Requests output for constraint forces of SPC points.
NLOAD	Requests output for NOLINI in transient response.
OUTPUT(PLOT)	Requests the beginning of the plotter output.
Superelement Control	
SUPER	Specifies the superelement identification number and the load sequence number.
SEALL	Combines the functions of SEMG, SELG, SEKR, SEMR, and SELR.
SEKR	Specifies the superelement identification numbers for which stiffness matrices will be assembled and reduced.
SELG	Specifies the superelement identification numbers for which load vectors will be generated.
SELR	Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.
SEMG	Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.
SEMR	Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced.

1.4 Bulk Data Section

All details of the input data are specified in the Bulk Data Section. Constraint, load, and solution method Bulk Data entries are not used unless they are selected in the Case Control Section. The nonlinear Bulk Data entries are summarized in [Table 1-2](#). All the input data designed specially for nonlinear analysis will be described in the following sections.

Table 1-2. Summary of Nonlinear Bulk Data Entries

Element Connectivity	
CBEAM	Defines connection for beam element.
CBUSH	Defines connections for generalized spring and damper.
CBUSH1D	Defines connections for rod type spring and damper.
CGAP	Defines connection for gap or frictional element.
CHEXA	Defines connection for six-sided solid element.

Table 1-2. Summary of Nonlinear Bulk Data Entries

CONROD	Defines connection and properties for rod.
CPENTA	Defines connection for five-sided solid element.
CPYRAM	Defines connection for five-sided solid element.
CQUAD4	Defines connection for quadrilateral element with plane strain or bending and membrane stiffness.
CQUAD8 and CQUAD	Defines connection for plane strain hyperelastic quadrilateral element.
CROD	Defines connection for rod with axial and torsional stiffness.
CTETRA	Defines connection for four-sided solid element.
CTRIA3	Defines connection for triangular element with plane strain or bending and membrane stiffness.
CTRIA6	Defines connection for hyperelastic plane strain triangular element.
CQUADX4 and CQUADX8	Defines connection for axisymmetric hyperelastic quadrilateral element.
CTRAX3 and CTRAX6	Defines connection for axisymmetric hyperelastic triangular element.
CTUBE	Defines connection for a tube.
Element Properties	
PBCOMP	Defines properties for composite CBEAM.
PBEAM	Defines properties for CBEAM.
PBUSH and PBUSHT	Defines properties for CBUSH.
PBUSH1D	Defines properties for CBUSH1D.
PCOMP	Defines properties for composite material laminate.
PGAP	Defines properties for CGAP.
PROD	Defines properties for CROD.
PSHELL	Defines properties for CTRIA3 and CQUAD4.
PSOLID	Defines properties for CHEXA, CPENTA, CPYRAM, and CTETRA.
PTUBE	Defines properties for CTUBE.
PLPLANE	Defines properties for hyperelastic plane elements CTRIA3, CTRIA6, CQUAD4, CQUAD8, CQUAD, CQUADX and CTRIAX.
PLSOLID	Defines properties for hyperelastic solid elements CHEXA, CPENTA, CPYRAM, and CTETRA.
Material Properties	
CREEP	Defines creep material properties.
MAT2	Defines anisotropic material properties for shell elements.
MAT8	Defines orthotropic material properties for shell elements.
MAT9	Defines anisotropic material properties for solid elements.
MATS1	Defines properties for plastic and nonlinear elastic material.
MATHP	Defines properties for hyperelastic material.
TABLEDi	Defines a function for internal load vs. disp or velocity for bush element.
TABLES1	Defines a function for stress-dependent material properties or experimental data for hyperelastic material.
TABLEST	Combines many TABLES1 entries for temperature dependent material properties.
Slideline Contact	

Table 1-2. Summary of Nonlinear Bulk Data Entries

BCONP	Defines contact parameters.
BLSEG	Defines boundary line segments.
BFRIC	Defines frictional properties for slideline contact.
BWIDTH	Defines boundary line segments width/thickness.
BOUTPUT	Defines grid points where contact output is requested.
Constraints	
SPC	Defines single-point constraints and enforced displacements.
SPC1	Defines single-point constraints.
MPC	Defines a linear relationship for two or more degrees-of-freedom.
Loads	
CLOAD	Defines a static load combination for superelement loads.
FORCEi	Defines concentrated load at grid point.
LSEQ	Defines static load sets for dynamic analysis.
MOMENTi	Defines moment at a grid point.
NOLINi	Defines nonlinear transient load.
PLOAD	Defines pressure loads on CQUAD4, CTRIA3, CHEXA, CPENTA, CPYRAM, and CTETRA. Should not be used for hyperelastic plane elements CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6 or for hyperelastic CHEXA, CPENTA, CTETRA with midside nodes.
PLOAD2	Defines pressure loads on shell elements, CQUAD4 and CTRIA3. Not available for hyperelastic elements.
PLOAD4	Defines pressure loads on surfaces of CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3 and CQUAD4 elements. Not available for hyperelastic plane elements CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6.
PLOADX1	Defines pressure loads on axisymmetric elements CQUADX4, CQUADX8, CTRAX3, and CTRAX6.
RFORCE	Defines load due to centrifugal force field.
TEMP	Defines temperature at grid points.
TEMPPi	Defines temperature field for surface elements.
TEMPRB	Defines temperature field for line elements.
TIC	Specifies initial values for displacement and velocity.
TLOADi	Defines loads as a function of time.
Solution Methods	
NLPARM	Defines iteration methods for nonlinear static analysis.
NLPCI	Defines arc-length methods for nonlinear static analysis.
TSTEPNL	Specifies integration and iteration methods for nonlinear transient analysis.
EIGB or EIGRL	Defines eigenvalue extraction method for buckling analysis.

1.5 Parameters

Parameters constitute an important part of the input data because they maintain communications among the user, DMAP, and modules. Parameters are single-valued input data used for requesting special features or specifying miscellaneous data.

You can change the initial values of parameters by specifying the PARAM entry in the Bulk Data Section or in the Case Control Section. Many options available in Simcenter Nastran are controlled by user-specified parameters. Table 1-3 lists the parameters used in nonlinear Solution Sequences 106 and 129

Table 1-3. PARAMeters Used in Nonlinear Solution Sequences

ParameterName	Solution Sequence			Default	Description
	64	106	129		
ADPCON		E	E	1.0	Adjusts automatic penalty values for sideline contact regions.
AUTOSPC	B	E	E	NO	Specifies automatic single-point-constraints.
AUTOSPCR		E	B	NO	Specifies automatic SPCs for residual structure.
BAILOUT		E	E	0	Terminates superelement program with singularities.
BETA			B	0.33333	Specifies control factor for Newmark-Beta method.
BUCKLE	E			-1	Selects nonlinear buckling analysis for restarts.
CB1,2		E		1.0	Selects superelement damping matrix coefficients.
CK1,2,3	E	E		1.0	Selects superelement stiffness matrix coefficients.
CM1,2	E	E		1.0	Selects superelement mass matrix coefficients.
CNSTRT	B			1	Controls restarts for constraint change.
COUPMASS	B	E	E	-1	Generates coupled rather than lumped mass matrices.
CURV	E			-1	Computes stress data in a material coordinate system.
CURVPLOT	E			-1	Requests x-y (curve) plots.
DBDICT	B	B	B	-1	Prints database dictionary at the end and beginning of the run.
DBDRNL		E	E	0	Prints database dictionary in the nonlinear loop (SOLs 66 and 99 only).
DLOAD	B	B	B	0	Indicates no other changes except for loads (SOLs 64 through 99 only).
DOPT	E			0	Controls x spacing of curves for CURVPLOT.
ERROR		E	E	-1	Terminates run for superelement errors.
EST	B	E	E	2	Requests element measure and volume computation.
FACTOR	B	B	B	10,000	Generates SEQID on SEQGP entry.
FKSYMFAC		B		1.0	Controls symmetrization of stiffness.
FOLLOWK		E	E	YES	Include follower force stiffness.
GPECT	B	E	E	-1	Prints all elements connected to each grid point.
GRDPNT	B	E	E	-1	Executes Grid Point Weight Generator.
G			E	0.0	Specifies uniform structural damping coefficient.
INRLM			E	0	Requests appending of inertia relief modes.
KDIAG	E			-1.0	Applies value of a spring constant to nonlinear stiffness matrix diagonals.
K6ROT	E	E	E	100.0	Assigns stiffness to normal rotation of CQUAD4 and CTRIA3 elements.
LANGLE		E	E	1	Selects gimbal angle approach for large rotations if 1 and rotation vector approach if 2.
LGDISP		E	E	-1	Selects large displacement effects.
LOOPID		E	E	0	Specifies loop ID in the database for restarts.
MAXLP			E	5	Maximum number of iterations for internal loop.

Table 1-3. PARAMETERS Used in Nonlinear Solution Sequences

ParameterName	Solution Sequence			Default	Description
	64	106	129		
MAXRATIO	E	E	E	1 .0E + 7	Determines the singularity of the stiffness matrix.
MODEL	B	B	B	0	Stores several models in the database.
MPCX	B	B	B	0	Controls MPC and rigid element processing on restarts.
NDAMP			E	0.025	Specifies numerical damping in ADAPT method.
NINTPTS	E			10	Requests CURV to interpolate over all elements.
NLAYERS		B	B	5	Specifies number of integration points through thickness for CQUAD4 and CTRIA3.
NLTOL		B		2	Sets accuracy tolerances
NMLOOP		E		0	Saves tangent stiffness in SOL 106 for normal modes and other linear solution sequences.
NOCOMPS		E		1	Suppresses composite stress output.
NODATA	B	B	B	0	Indicates no changes in Bulk Data (SOLs 64 through 99 only).
NOTRED			E	0	Controls t-set reduction (SOL 99 only).
OG	E			0	Calculates stress/strain data at grid points for CURV.
OLDELM	B			1	Computes element forces using old method.
OUTOPT	E			0	Sets format for output quantities computed by CURV.
OUTPUT	B			0	Jumps to data recovery.
PDRMSG			E	1	Suppresses data recovery messages.
PLOT	B	E	E	1	Jumps to undeformed plot.
PLOTSUP		E	E	0	Makes undeformed plots for superelements.
PLTMSG		E	E	1	Suppresses undeformed plot messages.
POST		E	E	1	Stores post-processing data blocks in database.
PRGPST	B	E	E	NO	Suppresses singularity printout.
PROUT	B	E	E	-1	Suppresses execution and printout from ELTPRT.
RESDUAL		E	E	1	Skips to nonlinear operations (Phase 2) (SOLs 66 through 99 only).
SDATA			E	1	Obtains additional solution set output (SOL 99 only).
SEFINAL		B	B	-1	Indicates no data changes if change SEFINAL.
SEMAP		B	B	SEMAP	Determines partitioning of superelements.
SEMAPPRT		B	B	3	Prints SEMAP table.
SENAME		E	E	SENAME	Identifies superelement by name.
SEQOUT	B	B	B	0	Controls output options for OLDSEQ.
SKPLOAD	E			1	Skips load vector calculation.
SKPMTRX	E			1	Skips matrix reduction and decomposition.
SLOOPID			B	0	Specifies loop ID from SOL66 database for restart.
SOLID	B	B	B	0	Stores several solutions in same database.
SPCGEN	B	E	E	0	Places automatic SPCs in SPC1 format.
SSG3	B			0	Skips operations prior to FBS.
START	B	B	B	0	Defines number of grid points for OLDSEQ.
STIME			E	0.0	Specifies time step for restarts.

Table 1-3. PARAMETERS Used in Nonlinear Solution Sequences

ParameterName	Solution Sequence			Default	Description
	64	106	129		
SUBID	B	E		1	Specifies subcase ID for restarts.
SUBSKP	B			0	Skips a number of subcases for restarts.
SUPER	B	B	B	0	Deletes ungrouped grid points.
TABID			E	2	Controls punch for response spectra.
TABS		E		0.0	Converts units of temperature input to absolute.
TESTNEG	E	E		-1/-2/1	Tests for negative terms on factor diagonal of matrix.
TESTSE	E			1.0E + 36	Tests for strain energy for iteration control.
TSTATIC			E	-1	Selects static solution with TSTATIC=1 using ADAPT method in TSTEPNL.
USETPRT	B	E	E	-1	Selects output type for degrees-of-freedom.
USETSEL	B	E	E	0	Controls sets for row list output option.
WTMASS	B	E	E	1.0	Multiplies terms of structural mass matrix by value.
W3,W4			E	0.0	Selects frequency for conversion of damping.

Note

B = Must be specified in the Bulk Data Section only.

E = May be specified in either the Bulk Data or Case Control Section.

1.6 Nonlinear Element Output Codes for Plotting

Nonlinear elements (which reference nonlinear geometry and/or material) have different plot codes than linear elements.

See Also

- “Item Codes” in the *Simcenter Nastran Quick Reference Guide*

1.7 Nonlinear Characteristics and General Recommendations

Modeling for nonlinear analysis should follow the guidelines for good modeling practice pertaining to linear analysis, which are summarized as follows:

- The analyst should have some insight into the behavior of the structure to be modeled; otherwise, a simple model should be the starting point.
- Substructuring should be considered for the modularity of the model and/or synergism between projects and agencies involved.
- The size of the model should be determined based on the purpose of the analysis, the trade-offs between accuracy and efficiency, and the scheduled deadline.

- Prior contemplation of the geometric modeling will increase efficiency in the long run. Factors to be considered include selection of coordinate systems, symmetric considerations for simplification, and systematic numbering of nodal points and elements for easy classification of locality.
- Discretization should be based on the anticipated stress gradient, i.e., a finer mesh in the area of stress concentrations.
- Element types and the mesh size should be carefully chosen. For example, avoid highly distorted and/or stretched elements (with high aspect ratio); use CTRIA3 and CTETRA only for geometric or topological reasons.
- The model should be verified prior to the analysis by some visual means, such as plots and graphic displays.

Nonlinear analysis requires better insight into structural behavior. First of all, the type of nonlinearities involved must be determined. If there is a change in constraints due to contact during loading, the problem may be classified as a boundary nonlinear problem and would require CGAP elements or slide line contact algorithm.

The material nonlinearity is characterized by material properties. However, the material nonlinear effects may or may not be significant depending on the magnitude and duration of the loading, and occasionally on environmental conditions. The anticipated stress level would be a key to this issue.

The geometric nonlinearity is characterized by large rotations which usually cause large displacements. Intuitively, geometric nonlinear effects should be significant if the deformed shape of the structure appears distinctive from the original geometry without amplifying the displacements. There is no distinct limit for large displacements because geometric nonlinear effects are related to the dimensions of the structure and the boundary conditions. The key to this issue is to know where the loading point is in the load-deflection curve of the critical area.

SOL 106 or SOL 129 must be used if the model contains any of the following:

- A parameter LGDISP (for geometric nonlinearity).
- Any hyperelastic element (PLPLANE or PLSOLID property entry, pointing to a MATHP material).
- Any CGAP element (CGAP and PGAP).
- Slideline contact (BCONP, BLSEG, and/or BFRIC).
- Any active nonlinear material data (specified on MATS1 and/or CREEP).
- Any combination of the above.

The model may consist of superelements, but only the residual structure (superelement 0) may consist of nonlinear elements mixed with any type of linear elements. All the gaps, slideline contact regions, and hyperelastic elements are always actively nonlinear if included in the residual structure. However, other potentially nonlinear elements in the residual structure become actively nonlinear only if a parameter, LGDISP, is used and/or if they use the nonlinear material data specified on the MATS1 and/or CREEP data entries.

The subcase structure of Simcenter Nastran may be used in SOLs 106 and 129 to characterize a nonproportional loading path. Cyclic loading is one example. Some nonlinear problems,

such as plastic deformation, creep deformation, and contact problems with frictional forces, are path-dependent; others, such as hyperelastic or purely geometric nonlinear, are path-independent. An incremental process may not be required for a purely geometric nonlinear problem. For computational efficiency, however, the analysis of such a problem does frequently need incremental solutions.

Note:

- Identify the type of nonlinearity and localize the nonlinear region for computational efficiency. If unsure, perform a linear analysis by SOL 101 or SOL 109 prior to the nonlinear analysis by SOL 106 or SOL 129, respectively.
- Segregate the linear region by using superelements and/or linear elements if possible. Notice that the potentially nonlinear elements can be used as linear elements.
- The nonlinear region usually requires a finer mesh. Use a finer mesh if severe element distortions or stress concentrations are anticipated.
- Be prepared for restarts with the database properly stored in cases of divergence and changing constraints or loading paths via subcases.
- The subcase structure should be utilized properly to divide the load or time history for convenience in restarts, data recovery, and database storage control, not to mention changing constraints and loading paths.
- The load or time for the subcase should then be further divided into increments, not to exceed 20 load steps (for SOL 106) or 200 time steps (for SOL 129) in each subcase.
- Many options are available in solution methods to be specified on NLPARM (for SOL 106) or TSTEPNL (for SOL 129) data entries. The defaults should be used on all options before gaining experience.
- Use caution when specifying CGAP element properties. In particular, the closed gap stiffness shouldn't exceed the stiffness of the adjacent degree-of-freedom by 1000 times.
- Normal rotation for CQUAD4 and CTRIA3 elements should be restrained by a parameter K6ROT when the geometric nonlinearity is involved. Default value for parameter K6ROT is 100.
- Understand the basic theory of plasticity or creep before using these capabilities.
- Use caution when preparing input data for creep, because it is unit dependent.
- For the transient analysis by SOL 129, some damping is desirable and the massless degree-of-freedom should be avoided.
- The time step size for a transient response analysis should be carefully determined based on the highest natural frequency of interest because it has significant effects on the efficiency as well as the accuracy.

Chapter 2: Elements for Nonlinear Analysis

2.1 Overview of Nonlinear Elements

The nonlinear elements can be divided into two categories:

- Physical elements, which have associated material properties.
- Contact elements, which don't have any associated material properties.

Physical Elements

The nonlinear physical elements are listed in [Table 2-1](#) and [Table 2-2](#).

The CBUSH element may have nonlinear force-deflection properties defined on the PBUSH and PBUSHT property entries. The CBUSH element is geometric linear; the orientation is not updated in case of large deformation. The CBUSH1D element is both geometric and material nonlinear.

The elements CROD, CONROD, and CTUBE may have material nonlinear extensional properties, with linear torsion. You may supply plastic or nonlinear elastic material properties. Since the stress-strain curve for compression need not be the same as for tension, this element can, for example, be used to model cables which cannot carry compression.

The CBEAM element has been modified to provide plastic hinges at the ends of an otherwise elastic element. This element is intended for collapse analysis in frameworks with loads at the joints, and materials with small work hardening. You don't need to specify the cross-section axis about which the yielding occurs, since the implementation allows for combinations of bending moments in two directions plus an axial load. The flexibility of the plastic hinge is based upon eight idealized rods at each end, chosen to match the total area, center of gravity and moments of inertia of the cross-section. As is shown in [Table 2-3](#), the calculated ultimate moment agrees reasonably well with the theoretical value for many cross-sections. The material specified on a MATS1 Bulk Data entry should be elastic-perfectly plastic. Use of any nonlinear material other than elastic-perfectly plastic will give inaccurate results.

The CQUAD4 and CTRIA3 plate elements are available for combined bending and membrane strains found in shell structures as well as for plane strain applications. For shell type problems the state of stress is assumed to be constant in each of a variable number of equal layers. One layer should be used for plane strain. Your input is the one-dimensional stress-strain curve for tension. Higher order CQUAD8, CQUAD, and CTRIA6 elements exist for fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain analysis.

The CHEXA, CPENTA, CPYRAM, and CTETRA elements are typically used to model thick-walled structures like castings and forgings. Each of these elements has full nonlinear capability except for the CPYRAM element which cannot be used for hyperelasticity.

Axisymmetric elements CQUADX4, CQUADX8, CTRAX3, and CTRAX6 are available for fully nonlinear analysis with the hyperelastic material.

Table 2-1. Physical Elements for Nonlinear Analysis –Small Strain Elements

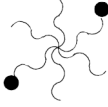
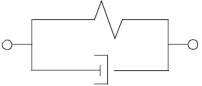

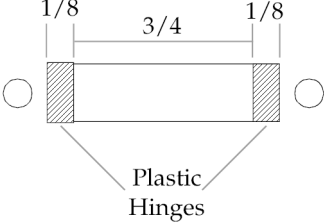
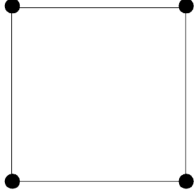
Name	Characteristics	Nonlinear Elastic	Elasto-Plastic	Geometric Nonlinear
CBUSH 	Generalized 3D spring damper with 6 stiffnesses for 3 translations and 3 rotations.	Yes	No	No
CBUSH1D 	Rod-like spring and damper with generalized force function $f(u, \dot{u})$ for axial displacement u and velocity \dot{u} .	Yes	No	Yes
CROD, CONROD, CTUBE 	Extension may be plastic or nonlinear elastic. Torsion is elastic (linear).	Yes	Yes	Yes
CBEAM 	Plastic hinge at each end, which couples axial motion and rotations. Linear material for center section, traverse shear, and torsion.	No	Yes (only elastic-perfectly plastic)	Yes
CQUAD4 	Shell and Plate Two-dimensional plasticity with FIVE layers for membrane and bending. 12 layers maximum; 1 layer minimum. Transverse shear is linear.	Yes	Yes	Yes
Plane Strain PSHELL with MID2=-1 and PARAM,NLAYER=1		Yes	Yes	Yes
Plane Stress MID2=0 NLAYER=1		Yes	Yes	Yes

Table 2-1. Physical Elements for Nonlinear Analysis –Small Strain Elements

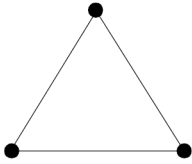
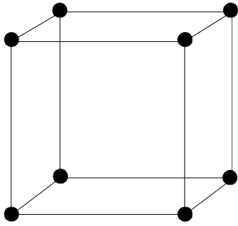
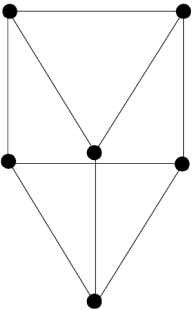
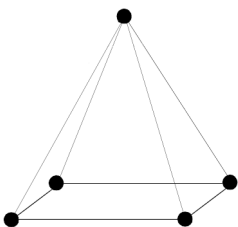
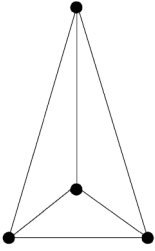
Name	Characteristics	Nonlinear Elastic	Elasto-Plastic	Geometric Nonlinear
<p>CTRIA3</p> 	<p>Shell and Plate</p> <p>Two-dimensional plasticity with FIVE layers for membrane and bending. 12 layers maximum; 1 layer minimum. Transverse shear is linear.</p>	Yes	Yes	Yes
	<p>Plane Strain</p> <p>PSHELL with MID2=-1 and PARAM,NLAYER=1</p>	Yes	Yes	Yes
	<p>Plane Stress</p> <p>PSHELL with MID2=0 NLAYER=1</p>	Yes	Yes	Yes
<p>CHEXA</p> 	<p>Mid-side nodes are allowed.</p> <p>For additional nonlinear characteristics, see PSOLID in the <i>Simcenter Nastran Quick Reference Guide</i>.</p>	Yes	Yes	Yes
<p>CPENTA</p> 	<p>Mid-side nodes are allowed.</p> <p>For additional nonlinear characteristics, see PSOLID in the <i>Simcenter Nastran Quick Reference Guide</i>.</p>	Yes	Yes	Yes
<p>CPYRAM</p> 	<p>Mid-side nodes are allowed.</p> <p>For additional nonlinear characteristics, see PSOLID in the <i>Simcenter Nastran Quick Reference Guide</i>.</p>	Yes	Yes	Yes

Table 2-1. Physical Elements for Nonlinear Analysis –Small Strain Elements

Name	Characteristics	Nonlinear Elastic	Elasto-Plastic	Geometric Nonlinear
CTETRA 	Mid-side nodes are allowed. For additional nonlinear characteristics, see PSOLID in the <i>Simcenter Nastran Quick Reference Guide</i> .	Yes	Yes	Yes

- One-dimensional stress-strain curves use the MAT1 entry.
- All other elements may be used for nonlinear analysis as long as they remain linear.

Table 2-2. Physical Elements for Nonlinear Analysis—Fully Nonlinear (Finite Deformation) Hyperelastic Elements

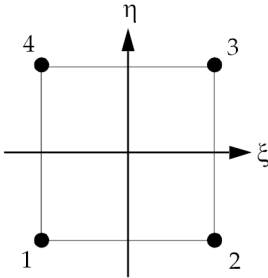
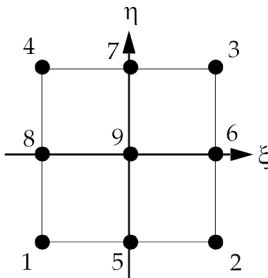
Name	Characteristics	Gauss Point	ξ	η	ζ
CQUAD, CQUAD4, CQUAD8, CQUADX4 	Plane strain or axisymmetric analysis. Linear isoparametric quadrilateral with constant pressure. Compressible and nearly incompressible applications. Four Gauss points.	1 2 3 4 $\alpha = 1/\sqrt{3}$	$-\alpha$ α $-\alpha$ α	$-\alpha$ $-\alpha$ α α	–
CQUAD, CQUAD8, CQUADX8 	Plane strain or axisymmetric analysis. Quadratic isoparametric quadrilateral with linear pressure. Compressible and nearly incompressible applications. Center node not available on the CQUAD8 connectivity. Nine Gauss points.	1 2 3 4 5 6 7 8 9 $\alpha = \sqrt{15}/5$	$-\alpha$ 0 α $-\alpha$ 0 α $-\alpha$ 0 α	$-\alpha$ $-\alpha$ $-\alpha$ 0 0 0 0 α α	–

Table 2-2. Physical Elements for Nonlinear Analysis—Fully Nonlinear (Finite Deformation) Hyperelastic Elements

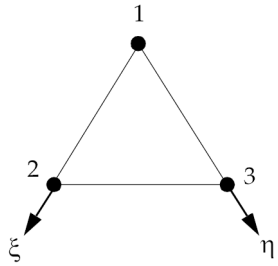
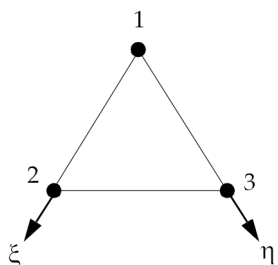
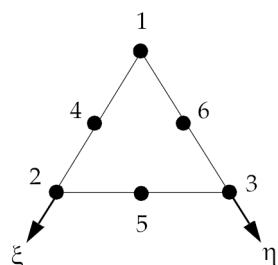
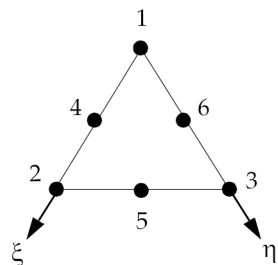
Name	Characteristics	Gauss Point	ξ	η	ζ
CTRIA3, CTRIA6 	Plane strain. Linear isoparametric triangle with constant pressure. Compressible and nearly incompressible applications. One Gauss point.	1	1/3	1/3	–
CTRAX3 	Axisymmetric analysis. Linear isoparametric triangle with constant pressure. Compressible and nearly incompressible applications. Three Gauss points.	1 2 3	1/6 2/3 1/6	1/6 1/6 2/3	–
CTRIA6 	Plane strain. Quadratic isoparametric triangle with constant pressure. Compressible and nearly incompressible applications. Three Gauss points.	1 2 3	1/6 2/3 1/6	1/6 1/6 2/3	–
CTRAX6 	Axisymmetric analysis. Quadratic isoparametric triangle with constant pressure. Compressible and nearly incompressible applications. Seven Gauss points.	1 2 3 4 5 6 7	0.333333 0.797427 0.101287 0.101287 0.470142 0.470142 0.059716	0.333333 0.101287 0.797427 0.101287 0.470142 0.059716 0.470142	–

Table 2-2. Physical Elements for Nonlinear Analysis—Fully Nonlinear (Finite Deformation) Hyperelastic Elements

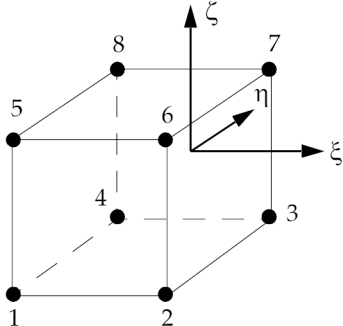
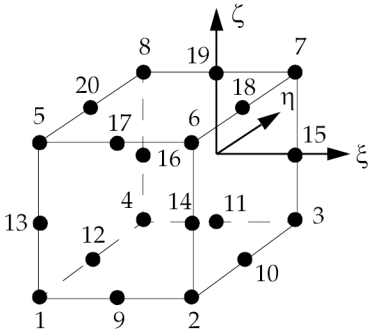
Name	Characteristics	Gauss Point	ξ	η	ζ
<p>CHEXA</p> 	<p>Linear isoparametric hexahedron with constant pressure. Compressible and nearly incompressible applications.</p> <p>Eight Gauss points.</p>	<p>1 2 3 4 5 6 7 8</p> <p>$\alpha = 1/\sqrt{3}$</p>	<p>α $-\alpha$ α $-\alpha$ α $-\alpha$ $+\alpha$ $-\alpha$</p>	<p>α α $-\alpha$ $-\alpha$ α α $-\alpha$ $-\alpha$</p>	<p>α α α α $-\alpha$ $-\alpha$ $-\alpha$ $-\alpha$</p>
<p>CHEXA</p> 	<p>Quadratic isoparametric hexahedron with linear pressure. Compressible and nearly incompressible applications.</p> <p>Twenty-seven Gauss points.</p>	<p>1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23</p>	<p>α 0 $-\alpha$ α 0 $-\alpha$ α 0 $-\alpha$ α 0 $-\alpha$ α 0 $-\alpha$ α 0 $-\alpha$ α 0 $-\alpha$ α 0</p>	<p>α α α 0 0 0 $-\alpha$ $-\alpha$ $-\alpha$ α α α α 0 0 0 $-\alpha$ $-\alpha$ $-\alpha$ α α α 0</p>	<p>α α α α α α α α α 0 0 0 0 0 0 0 0 0 $-\alpha$ $-\alpha$ $-\alpha$ $-\alpha$ $-\alpha$</p>

Table 2-2. Physical Elements for Nonlinear Analysis—Fully Nonlinear (Finite Deformation) Hyperelastic Elements

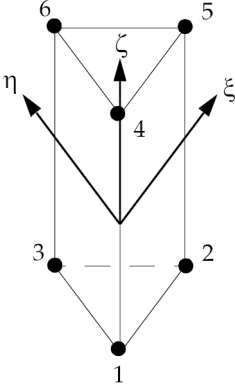
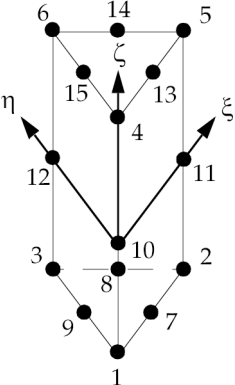
Name	Characteristics	Gauss Point	ξ	η	ζ
		24 25 26 27 $\alpha = (\sqrt{15})/5$	$-\alpha$ α 0 $-\alpha$	0 $-\alpha$ $-\alpha$ $-\alpha$	$-\alpha$ $-\alpha$ $-\alpha$ $-\alpha$
CPENTA 	Linear isoparametric pentahedron with constant pressure. Compressible and nearly incompressible applications. Six Gauss points.	1 2 3 4 5 6	$1/2$ $1/2$ 0 $1/2$ $1/2$ 0	0 $1/2$ $1/2$ 0 $1/2$ $1/2$	$1/\sqrt{3}$ $-1/\sqrt{3}$ $-1/\sqrt{3}$ $1/\sqrt{3}$ $1/\sqrt{3}$ $1/\sqrt{3}$
CPENTA 	Quadratic isoparametric pentahedron with linear pressure. Compressible and nearly incompressible applications. Twenty-one Gauss points. $\alpha_1 = 0.0597158717$ $\beta_1 = 0.4701420641$ $\alpha_2 = 0.7974269853$ $\beta_2 = 0.1012865073$	1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17	$1/3$ α_1 β_1 β_1 α_2 β_2 β_2 β_2 $1/3$ α_1 β_1 β_1 α_2 β_2 β_2 $1/3$ α_1 β_1	$1/3$ β_1 α_1 β_1 β_2 α_2 β_2 β_2 $1/3$ β_1 α_1 β_1 β_2 α_2 β_2 $1/3$ β_1 α_1	$-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ $-\sqrt{15}/5$ 0 0 0 0 0 0 0 0 0 0 $+\sqrt{15}/5$ $+\sqrt{15}/5$

Table 2-2. Physical Elements for Nonlinear Analysis—Fully Nonlinear (Finite Deformation) Hyperelastic Elements

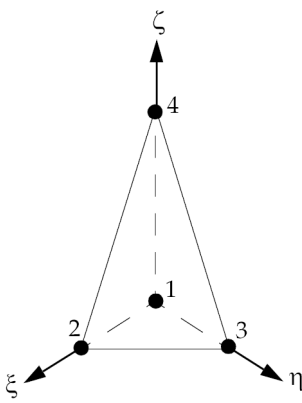
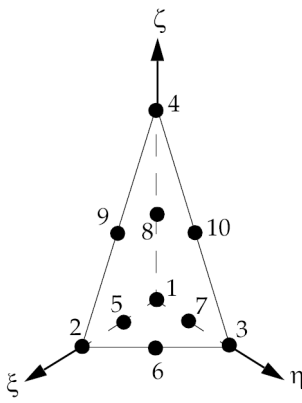
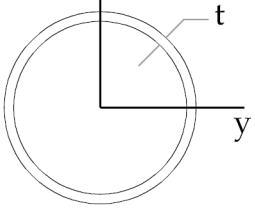
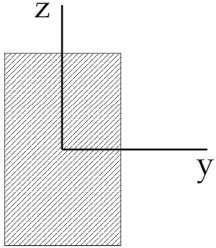
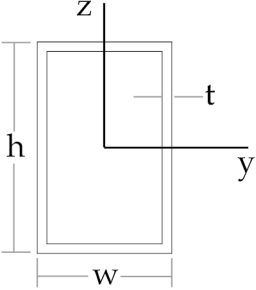
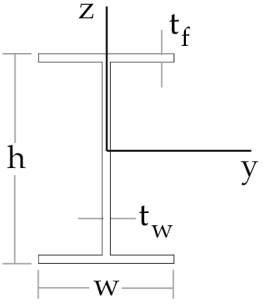
Name	Characteristics	Gauss Point	ξ	η	ζ
		18	β_1	β_1	$+\sqrt{15}/5$
		19	α_2	β_2	$+\sqrt{15}/5$
		20	β_2	α_2	$+\sqrt{15}/5$
		21	β_2	β_2	$+\sqrt{15}/5$
					$+\sqrt{15}/5$
CTETRA	Linear isoparametric tetrahedron with constant pressure. Compressible and nearly incompressible applications. One Gauss point.	1	1/4	1/4	1/4
					
CTETRA	Quadratic isoparametric tetrahedron with linear pressure. Compressible and nearly incompressible applications. Five Gauss points.	1	1/6	1/6	1/6
		2	1/2	1/6	1/6
		3	1/6	1/2	1/6
		4	1/6	1/6	1/2
		5	1/4	1/4	1/4
					

Table 2-3. Accuracy in Calculation of Ultimate Moment in Yielded State

Section	η		
	Moment Axis		
	Any multiple of 45° from y-axis	.9481	
	y or z	.9856	
 <p>$t \ll h, w$</p>	h/w	η for y-axis	η for z-axis
	0	.8536	.9856
	.5	.9031	.9543
	1.0	.9295	.9295
	2.0	.9543	.9031
	∞	.9856	.8536
 <p>$t_f, t_w \ll h, w$</p>	ht_w / wt_f	η for y-axis	η for z-axis
	0	.8536	.9856
	.5	.8826	1.102
	1.0	.9031	1.207
	2.0	.9295	1.394
	∞	.9856	

Contact Elements

The contact (interface) elements are listed in [Table 2-4](#).

The CGAP element is intended to model surfaces which may come into contact. When positive pressure exists, the gap can carry any transverse shear load which is less than the coefficient of friction times the normal load. The CGAP element connects two grid points which may be initially

coincident. There is no geometric nonlinear behavior, which implies that the orientation of the contact plane does not change during deflection. (The physical shape of the two contact surface would have to be specified and it would require the solution of a difficult analytic problem to determine the location of the actual contact point or points.)

Due to the requirements of the solution algorithm, there must be a finite compression stiffness for the CGAP element. The value must be carefully chosen, since a very large value may lead to numerical problems. A finite extensional stiffness may be used to prevent the drift of structures that are supported only by CGAP elements. You may also supply an initial opening of the gap. An orientation vector is required (see CGAP entry for details) to define the transverse axes, even if no shear is to be carried.

The slideline contact is capable of modeling nonlinear contact geometry and inelastic material behavior including large deformation. Slideline contact is useful for two dimensional geometries, for example plane stress, plane strain and axisymmetric. They can also be used for three dimensional geometries provided the contact between the two bodies can be defined in terms of parallel planes called the slideline planes. The sliding and separation of the two bodies is restricted to the slideline planes. The bodies can have large relative motions within the slideline plane. However, relative motions outside the slideline planes are ignored; therefore, they must be small compared to a typical slideline element.

Table 2-4. Contact Elements for Nonlinear Analysis

Name	Characteristics	Friction Law	Geometric Nonlinearity
CGAP	Contact between two points. Extension has different springs for compression and tension. Shear force is less than the friction coefficient times the compression. Transverse finite compression stiffness is required.	Coulomb's Law with:kinetic friction 0 static friction	No
Slideline	Contact between a point and a line.	Coulomb's Law with:kinetic friction = static friction	Yes

2.2 Small Strain Elements

A subset of Simcenter Nastran's elements for linear analysis is available for nonlinear structural analysis (see [Table 2-5](#)). The nonlinear elements for small strain analysis can handle material or geometric nonlinearities or both. In geometric nonlinear analysis, the elements may undergo large total displacements and rotations but the net deformation of each element has to remain small, therefore these elements are called “small strain” elements. For linear material and infinitesimal deformation, a nonlinear analysis with small strain elements gives results which are identical to a linear analysis. The small strain elements may be combined with any other type of elements, e.g., hyperelastic elements and/or linear elements.

Table 2-5. Small Strain Elements in Nonlinear Structural Analysis

Element	Property	Material ⁴
Line Elements		
CROD	PROD	MAT1 (MATT1), MATS1
CONROD	–	MAT1 (MATT1)
CTUBE	PTUBE	MAT1 (MATT1)
CBEAM	PBEAM	MAT1 (MATT1), MATS1
	PBCOMP ¹ (Composite Beam)	(only elastic-perfectly plastic)
2D ElementsPlane Strain, Plane Stress, Shells		
CQUAD4	PSHELL	MAT1 (MATT1), MAT2 (MATT2), MAT8 ³ , MATS1, CREEP
CTRIA3	PCOMP (Composite Shell) ²	
Solid Elements		
CHEXA	PSOLID	MAT1 (MATT1), MAT9 (MATT9), MAT11 (MATT11), MATS1, CREEP
CPENTA		
CPYRAM		
CTETRA		

For all elements, geometric nonlinearity is turned on with PARAM, LGDISP,1.

1. PBCOMP may refer to MAT1 only (linear material).
2. PCOMP must refer to MAT1, MAT2, MAT8 (only linear material).
3. MAT8 must not be referred by a MATS1 entry (only linear orthotropic material).

4. MAT1 isotropic material.

MAT2 anisotropic material for plates.

MAT8 orthotropic material for plates.

MAT9 anisotropic material for solids.

MAT11 orthotropic material for solids.

MATTi temperature dependency for MAT1, MAT2, MAT8, MAT9, MAT11.

MATS1 nonlinear material (piecewise linear, elasto-plastic) referred by MAT1, MAT2, MAT8, MAT9, MAT11.

CREEP creep characteristics, referred by MAT1, MAT2, MAT9.

User Interface

Element-related user input is summarized in the table **Small Strain Elements in Nonlinear Structural Analysis**. The nonlinear element calculations are activated if a nonlinear material exists (MATS1 or CREEP Bulk Data entries) or if a geometric nonlinear analysis of the whole model is requested with PARAM, LGDISP,1. The default is geometric linear analysis, PARAM, LGDISP,-1.

For elements with rotational degrees-of-freedom (CBEAM, CTRIA3, CQUAD4), two different definitions for large rotations are available, gimbal angles and rotation vector. The definitions are chosen with `PARAM,LANGLE,i`. The default is the gimbal angle definition, $i = 1$. The 3 gimbal angle components are the 3 subsequent rotations around the global axes. The second rotation component refers to the second global axis which has been rotated once by the first rotation. The third rotation refers to the third global axis which has been rotated twice by the first and second rotation. The rotation vector definition is chosen with $i = 2$. The components of the rotation vector refer to the fixed global axes. The direction of the rotation vector is the principal direction of the 3-D rotation. The magnitude of the rotation vector is the principal value of the 3-D rotation. The difference of the two angle definitions in the output of the elements appears only for large 3-D rotations.

Output

For this output discussion, an element is a nonlinear element if it supports geometric nonlinear and `PARAM,LGDISP,1` is defined, and/or material nonlinear (plasticity, creep or hyperelastic) is defined on this element.

Element stress and strain output is requested with the `STRESS` case control command. The stress and strain output for nonlinear elements is written into both a nonlinear and a linear format. The nonlinear format outputs stresses together with strains. Compared to the linear format, the nonlinear format provides more information with regard to nonlinear material laws (effective strain, equivalent stress etc.). The stresses and strains of the small strain elements refer to the undeformed area and length, respectively. The components of the stresses and strains are in the deformed element coordinate system which co-rotates with the elements' rigid body deformation.

Averaged grid point stresses are requested with the `GPSTRESS` case control command. Averaged grid point stresses for nonlinear elements is available in nonlinear static analysis but not in nonlinear transient analysis.

Element forces are requested with the `FORCE` case control command except for the `CBUSH` and `CBUSH1D` elements. The `STRESS` and `NLSTRESS` case control commands should be used to request the element force and stress output for the `CBUSH` and `CBUSH1D` elements. Element forces per unit length refer to the undeformed length. The components of the element forces are in the deformed element system. For nonlinear elements, element forces are available in nonlinear static analysis but not in nonlinear transient analysis. Element forces are not available for solid elements.

Deformations are requested with the `DISPLACEMENT` case control command. The output of displacements and rotations is available in nonlinear static and transient analysis.

Grid point forces are not available in nonlinear static or transient analysis.

Co-rotational Formulation in Geometric Nonlinear Analysis

For small strain elements, a large deformation is split into element rigid body deformation and element net distortion. The rigid body rotations of the points in the element are approximated by the rotation of the local element triad which follows the element deformation, as shown in [Figure 2-1](#). For the remaining element net distortions, a linear strain measure is used. The method is called co-rotational formulation. The formulation is capable to simulate large total deformations if a fine mesh is used and the stretches remain small.

In geometric nonlinear analysis with Simcenter Nastran, the tangent stiffness matrix is calculated in two steps. First, the linear stiffness or nonlinear material stiffness is calculated. The linear stiffness or nonlinear material stiffness is constructed from the variation of the stresses in the virtual work.

Second, the differential stiffness is calculated. The differential stiffness is constructed from the variation of the strains in the virtual work. The differential stiffness is also called geometric stiffness.

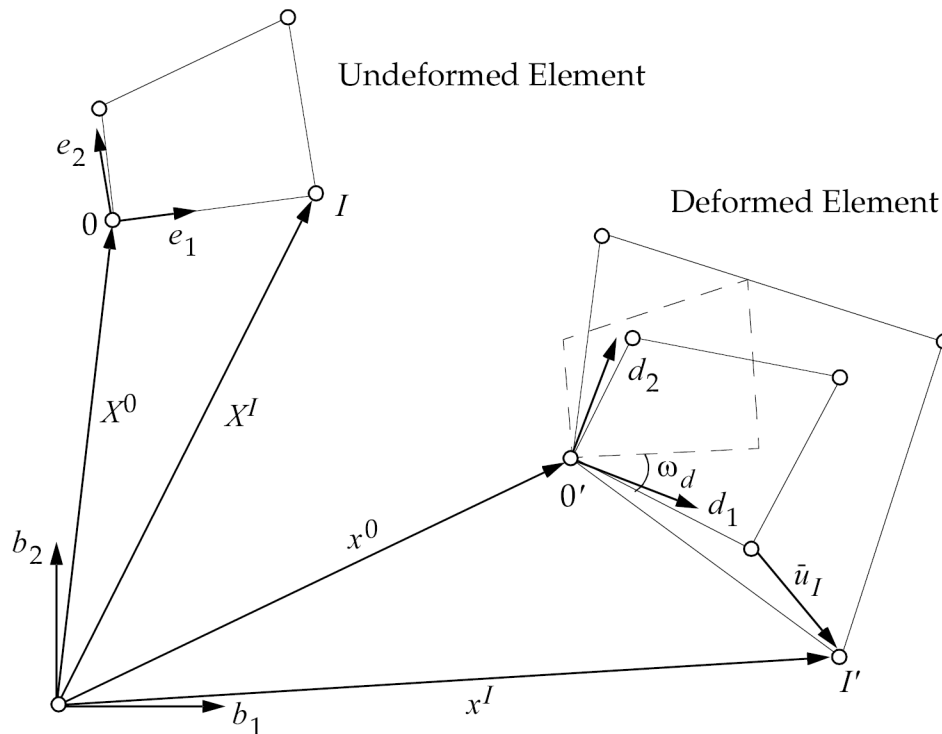


Figure 2-1. Co-rotational Concept for Small Strain Elements in Geometric Nonlinear Analysis

Guidelines

Large element net deformations should be avoided. In areas of the structure where large total deformations are expected, the mesh must be fine enough to keep the element net deformations small. The element net rotation should not exceed 20 degrees and the element should not be stretched by more than 10%. If stretches exceed 20%, it is recommended to use hyperelastic elements if applicable.

Limitations

- The element net distortions have to remain small.
- Rigid body elements (RBEi, RBAR, RROD entries, etc.) do not rotate in geometric nonlinear analysis.
- Multipoint constraints (MPCs) remain linear, the user-defined constraint equations don't change automatically in geometric nonlinear analysis. In nonlinear static analysis, you may change the MPCs from subcase to subcase. Then, the changes in the MPCs are accounted for incrementally.
- Offsets in the CBEAM, CTRIA3 and CQUAD4 elements are not allowed in combination with nonlinear material.

- The results in linear and nonlinear buckling with offsets may be incorrect.
- A temperature variation of the nonlinear material constants in plasticity is not available.
- For the CBEAM, the only nonlinear material available is elastic-perfectly plastic.

2.3 Fully Nonlinear Hyperelastic Elements

The hyperelastic elements are fully nonlinear finite deformation elements, including large strain and large rotation. The following elements are available: plane strain (CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6), solid (CHEXA, CPENTA, and CTETRA) and axisymmetric elements (CQUADX4, CQUADX8, CTRAX3, and CTRAX6).

The following tables summarize the supported properties for elements in a nonlinear analysis.

Table 2-6. Summary of Properties of the Nonlinear Elements				
Connectivity	Properties	No. of GRID Points	No. of Gauss Points	Geometric Nonlinearity
BCONP	BWIDTH	variable		X
BLSEG	BFRIC			
CBEAM	PBEAM	3	16*	X
	PBCOMP			
CBUSH	PBUSH	1 or 2		
	PBUSHT			
CBUSH1D	PBUSH1D	1 or 2		X
CGAP	PGAP	3	0	
CHEXA	PSOLID	8	8	X
	PLSOLID	8	8	X
		20‡	27	
CONROD	—	2	1	X
CPENTA	PSOLID	6	6	X
	PLSOLID	6	6	X
		15‡	21	
CQUAD4	PSHELL	4	5*	X**
	PCOMP			
	PLPLANE	4	4	X
CQUAD8	PLPLANE	8‡	9	X
CQUAD	PLPLANE	9‡	9	X
CROD	PROD	2	1	X
CTETRA	PSOLID	4,10	1	X
	PLSOLID	4	1	X
		10‡	5	

Table 2-6. Summary of Properties of the Nonlinear Elements

Connectivity	Properties	No. of GRID Points	No. of Gauss Points	Geometric Nonlinearity
CTRIA3	PSHELL	3	5*	X**
	PCOMP			
	PLPLANE	3	1	X
CTRIA6	PLPLANE	6‡	3	X
CTUBE	PTUBE	2	1	X
CQUADX4	PLSOLID	4	4	
CQUADX8	PLSOLID	8	9	
CTRAX3	PLSOLID	3	3	
CTRAX6	PLSOLID	6	7	

Note

Notes on Property Table

- * May be changed by you.
- ** Always in equilibrium in updated system.
- ‡ Center grid, if applicable, and midside grids may be omitted. For hyperelastic elements it is recommended that either all or no midside grids be omitted.
- Edge nodes are not applicable to nonlinear elements, except hyperelastic elements and the TET10 element.

Table 2-7. Summary of Material Properties of the Nonlinear Elements

Connectivity	Properties	Material Property		
		Nonlinear Isotropic	Anisotropic	Orthotropic
BCONP	BWIDTH			
BLSEG	BFRIC			
CBEAM	PBEAM	1†		
	PBCOMP			
CBUSH	PBUSH	X	X	
	PBUSHT			
CBUSH1D	PBUSH1D	X		
CGAP	PGAP			
CHEXA	PSOLID	1	9	
	PLSOLID	MATHP		
CONROD	—	1		
CPENTA	PSOLID	1	9	
	PLSOLID	MATHP		

Table 2-7. Summary of Material Properties of the Nonlinear Elements

Connectivity	Properties	Material Property		
		Nonlinear Isotropic	Anisotropic	Orthotropic
CQUAD4	PSHELL	1	2	8
	PCOMP			
	PLPLANE	MATHP		
CQUAD8	PLPLANE	MATHP		
CQUAD	PLPLANE	MATHP		
CROD	PROD	1		
CTETRA	PSOLID	1	9	
	PLSOLID	MATHP		
CTRIA3	PSHELL	1	2	8
	PCOMP			
	PLPLANE	MATHP		
CTRIA6	PLPLANE	MATHP		
CTUBE	PTUBE	1		
CQUADX4	PLSOLID	MATHP		
CQUADX8	PLSOLID	MATHP		
CTRAX3	PLSOLID	MATHP		
CTRAX6	PLSOLID	MATHP		

Note

Notes on Property Table

- * May be changed by you.
- † For the beam element, only elastic-perfectly plastic material is available.
- Integers listed under Material properties identify (i) on MATi Bulk Data entries.
- MATS1 and CREEP may be attached to MAT1, MAT2, and MAT9.

Table 2-8. Summary of Static Load Properties of the Nonlinear Elements

Connectivity	Properties	Static Loads		
		Thermal	Pressure	Gravity
BONP	BWIDTH			
BLSEG	BFRIC			
CBEAM	PBEAM	X		X
	PBCOMP			
CBUSH	PBUSH			
	PBUSHT			

Table 2-8. Summary of Static Load Properties of the Nonlinear Elements

Connectivity	Properties	Static Loads		
		Thermal	Pressure	Gravity
CBUSH1D	PBUSH1D			
CGAP	PGAP			
CHEXA	PSOLID	X	X	X
	PLSOLID	X	X	X
CONROD	—	X		X
CPENTA	PSOLID	X	X	X
	PLSOLID	X	X	X
CQUAD4	PSHELL	X	X	X
	PCOMP			
	PLPLANE	X		X
CQUAD8	PLPLANE	X		X
CQUAD	PLPLANE	X		X
CROD	PROD	X		X
CTETRA	PSOLID	X	X	X
	PLSOLID	X	X	X
CTRIA3	PSHELL	X	X	X
	PCOMP			
	PLPLANE	X		X
CTRIA6	PLPLANE	X		X
CTUBE	PTUBE	X		X
CQUADX4	PLSOLID	X	X	X
CQUADX8	PLSOLID	X	X	X
CTRAX3	PLSOLID	X	X	X
CTRAX6	PLSOLID	X	X	X

Table 2-9. Summary of Data Recovery Properties of the Nonlinear Elements

Connectivity	Properties	DATA Recovery				
		Stress	Grid Point Stress	Force		
BCONP	BWIDTH	X		X		
BLSEG	BFRIC					
CBEAM	PBEAM	X		X		
	PBCOMP					
CBUSH	PBUSH	X		X		
	PBUSHT					
CBUSH1D	PBUSH1D	X		X		
CGAP	PGAP	X		X		

Table 2-9. Summary of Data Recovery Properties of the Nonlinear Elements

Connectivity	Properties	DATA Recovery				
		Stress	Grid Point Stress	Force		
CHEXA	PSOLID	X	X			
	PLSOLID	X				
CONROD	—	X	X			
CPENTA	PSOLID	X	X			
	PLSOLID	X				
CQUAD4	PSHELL	X	X	X		
	PCOMP					
	PLPLANE	X				
CQUAD8	PLPLANE	X				
CQUAD	PLPLANE	X				
CROD	PROD	X		X		
CTETRA	PSOLID	X	X			
	PLSOLID	X				
CTRIA3	PSHELL	X	X	X		
	PCOMP					
	PLPLANE	X				
CTRIA6	PLPLANE	X				
CTUBE	PTUBE	X		X		
CQUADX4	PLSOLID	X				
CQUADX8	PLSOLID	X				
CTRAX3	PLSOLID	X				
CTRAX6	PLSOLID	X				

The element formulation is total Lagrangian in updated coordinates. See [Figure 2-2](#) for an illustration of the total Lagrangian concept. A nonlinear strain measure and pointwise rotation computed at each Gauss point are employed. Equilibrium is satisfied in the deformed configuration.

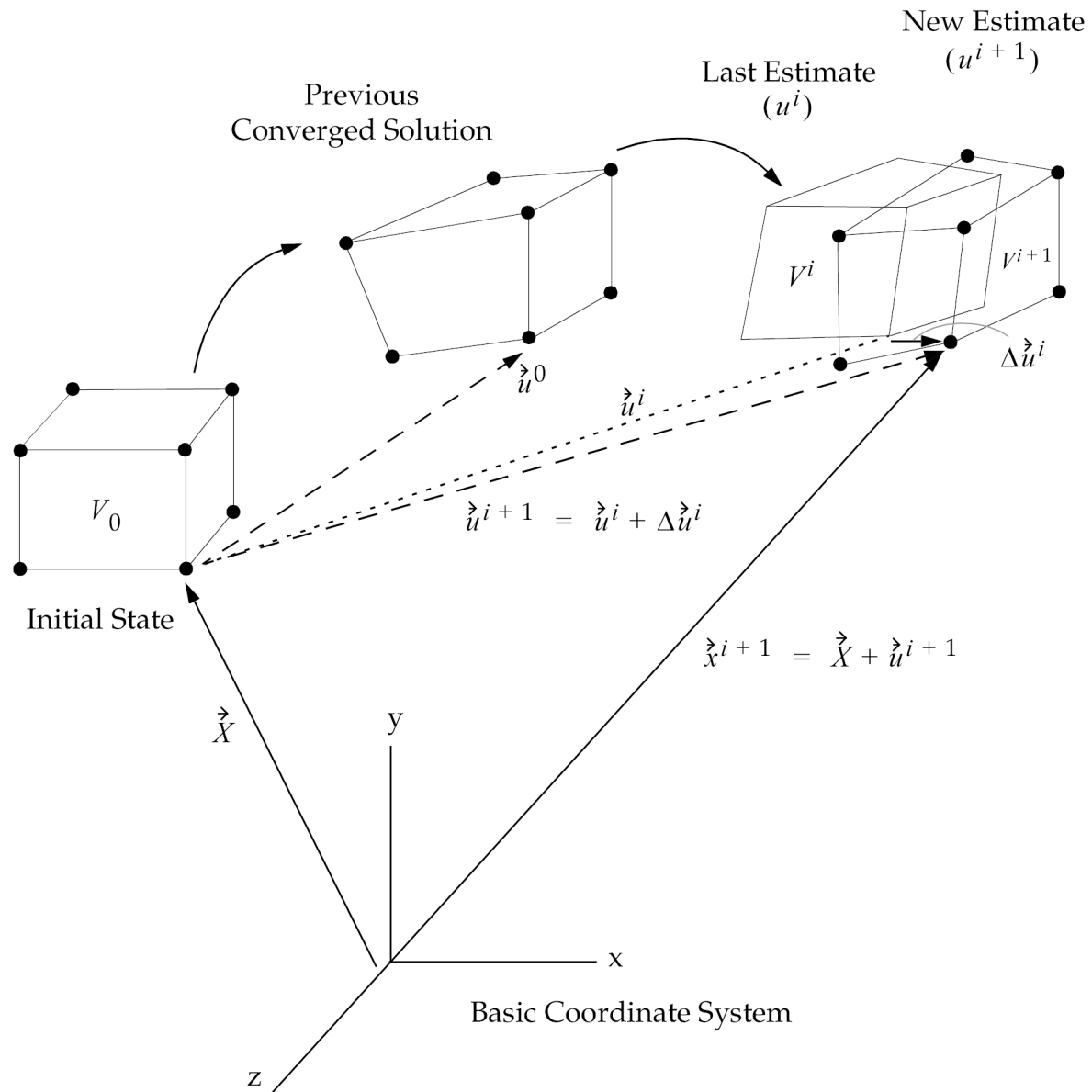


Figure 2-2. Total Lagrangian Concept: Deformations are Measured from an Initial Undeformed State

The elements are especially designed to address volumetric locking and other related problems such as ill-conditioning of the stiffness matrix, incorrect stresses etc., due to near incompressibility. To this end, a mixed formulation based on a three field variational principle in which the pressure and the volume ratio are interpolated independently from the displacements is employed. At the incompressible limit the bulk modulus becomes infinite and the volumetric strain $J - 1$ becomes zero so that the pressure may no longer be obtained from the displacements through the constitutive equation. The pressure and volume ratio interpolations are discontinuous between the elements.

The hyperelastic elements are supported in the nonlinear solutions, SOLs 106 and 129. They can be run through SOLs 101 and 105 for model checkout in linear analysis, the results, however, will generally not be correct. For buckling analysis, PARAM,BUCKLE in SOL 106 should be used instead.

User Interface

Because there is no difference in geometry, you define the hyperelastic elements using the current CQUAD4, CQUAD8, CTRIA3, CTRIA6, CHEXA, CPENTA, and CTETRA connectivity entries. Additional CQUADX4, CQUADX8, CTRAX3, and CTRAX6 connectivity entries are available for the hyperelastic elements. The PID field for these connectivity entries reference either PLPLANE or PLSOLID property entries. The MID field references a MATHP material entry that defines the hyperelastic material. This is the only material model available for fully nonlinear, finite deformation analysis.

See the *Simcenter Nastran Element Library* for more information.

Output Description

The nonlinear stress output for the hyperelastic elements differs from other material nonlinear stress output. The hyperelastic stress output contains the following quantities, at each Gauss point, in the basic coordinate system for solid and axisymmetric elements and in the CID coordinate system, defined on a PLPLANE entry, for plane strain elements:

Cauchy stresses, i.e., components of the stress vector $\mathbf{t} = \frac{d\mathbf{P}}{dA}$ where dA is an infinitesimal area in the deformed configuration, and \mathbf{P} is the force.

- Pressure

$$p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

- Logarithmic strains, i.e., the components of

$$\sum_{l=1}^3 \ln \lambda_l N_l N_l^T$$

where λ_l are principal stretches l/l_0 in the principal directions N_l . See [Figure 2-3](#). For compatibility with other Simcenter Nastran elements, twice the shear components of the logarithmic strain tensor are printed in the output.

N_l = principal directions in undeformed configuration; $n_l = RN_l$ = principal directions in deformed configuration, obtained from N_l by a rigid body rotation. The deformation along N_l is a pure stretch, after the rigid body motion has been factored out.

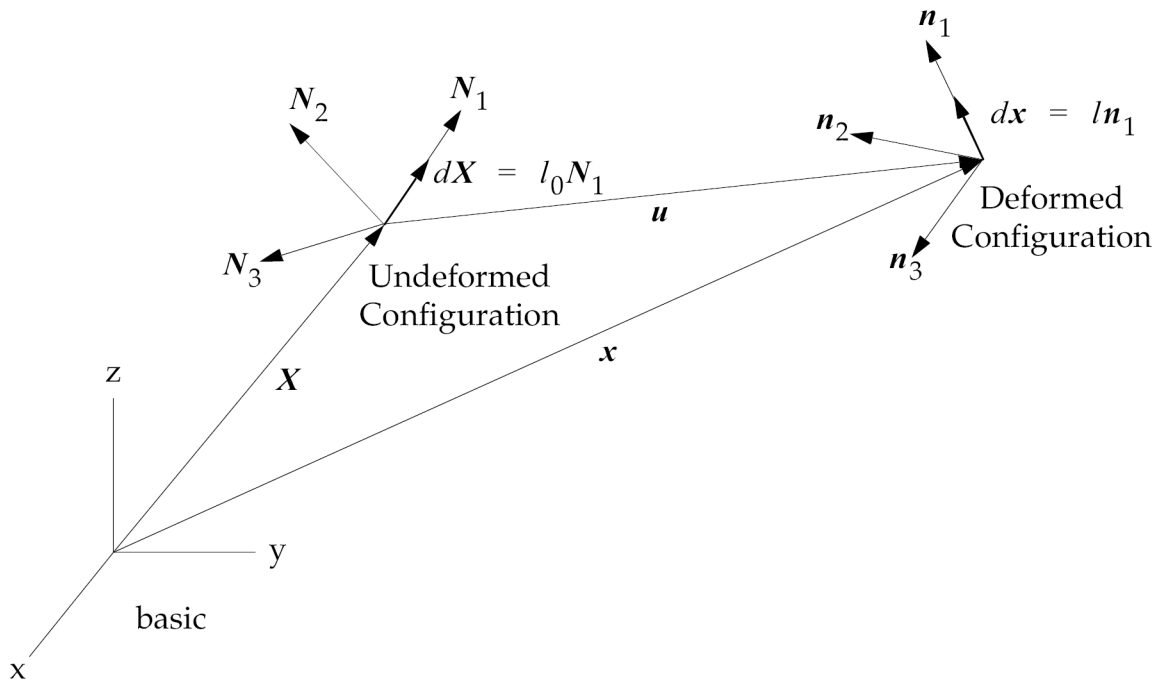


Figure 2-3.

- Volumetric strain

$$J - 1 = \frac{dV - dV_0}{dV_0}$$

where J is an approximation to the determinant of the deformation gradient dx/dX .

This output is obtained with the STRESS (or ELSTRESS) Case Control command. Examples of the new output for the plane, axisymmetric and solid elements are shown in [Listing 2-1](#).

The linear output contains principal Cauchy stresses and directions with respect to the basic (or CID) coordinate system. See [Listing 2-2](#).

```

NONLINEAR STRESSES IN HYPERELASTIC QUADRILATERAL ELEMENTS (QUAD4D)

ELEMENT GRID/ POINT          CAUCHY STRESSES/ LOG STRAINS          PRESSURE          VOL. STRAIN
  ID GAUSS      ID          X          Y          Z          XY
  1 GAUSS      1  1.678579E+02 -1.188351E-03  5.456201E+00  1.560323E-16  5.777097E+01  1.925699E-02
    1.791759E+00 -1.772686E+00  .0  .0
  2  1.678579E+02 -1.188351E-03  5.456201E+00 -1.191492E-15  5.777097E+01  1.925699E-02
    1.791759E+00 -1.772686E+00  .0  .0
  3  1.678579E+02 -1.188351E-03  5.456201E+00 -3.368891E-16  5.777097E+01  1.925699E-02
    1.791759E+00 -1.772686E+00  .0  .0
  4  1.678579E+02 -1.188351E-03  5.456201E+00 -7.083336E-16  5.777097E+01  1.925699E-02
    1.791759E+00 -1.772686E+00  .0  .0

NONLINEAR STRESSES IN HYPERELASTIC AXISYMM. QUADRILATERAL ELEMENTS (QUAD4D)

ELEMENT GRID/ POINT          CAUCHY STRESSES/ LOG STRAINS          PRESSURE          VOL. STRAIN
  ID GAUSS      ID  RADIAL(X)  AXIAL(Y)  THRTA(Z)  XY
  1 GAUSS      1  1.808360E-09  2.917973E+02  1.808360E-09  4.298727E-15  9.726578E+01  3.242192E-02
    -9.420009E-01  1.930910E+00 -9.570014E-01  1.188717E-01
  2  1.808132E-09  2.917973E+02  1.808132E-09 -2.820565E-17  9.726578E+01  3.242192E-02
    -9.420009E-01  1.930910E+00 -9.570014E-01  1.188717E-01
  3  1.808019E-09  2.917973E+02  1.808019E-09 -2.820565E-17  9.726578E+01  3.242192E-02
    -9.420009E-01  1.930910E+00 -9.570014E-01  1.188717E-01
  4  1.808218E-09  2.917973E+02  1.808218E-09  1.004509E-16  9.726578E+01  3.242192E-02
    -9.420009E-01  1.930910E+00 -9.570014E-01  1.188717E-01

NONLINEAR STRESSES IN HYPERELASTIC HEXAHEDRON ELEMENTS (HEXA4D)

ELEMENT GRID/ POINT          CAUCHY STRESSES/ LOG STRAINS          PRESSURE          VOL. STRAIN
  ID GAUSS      ID          X          Y          Z          XY          YZ          ZX
  1 GAUSS      1  2.9180E+02  1.8047E-09  1.8047E-09 -7.8413E-15 -7.7424E-17  2.3330E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  2  2.9180E+02  1.8047E-09  1.8047E-09 -3.1028E-16 -1.4850E-16  9.6370E-16  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  3  2.9180E+02  1.8047E-09  1.8047E-09  2.8235E-15  6.2553E-17 -6.4668E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  4  2.9180E+02  1.8047E-09  1.8047E-09 -5.9973E-16  9.1812E-17 -2.1878E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  5  2.9180E+02  1.8047E-09  1.8047E-09  2.3338E-15 -1.2646E-16  2.8235E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  6  2.9180E+02  1.8047E-09  1.8047E-09 -2.3359E-16 -1.1296E-16  2.1388E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  7  2.9180E+02  1.8047E-09  1.8047E-09  2.1626E-15  2.1405E-17 -6.2480E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0
  8  2.9180E+02  1.8047E-09  1.8047E-09 -4.0477E-16  2.4964E-17 -1.9690E-15  9.7266E+01  3.2422E-02
    1.9459E+00 -9.5700E-01 -9.5700E-01  .0  .0  .0

SIMPLE TENSION
JULY 23, 2003 NX NASTRAN 07/23/03 PAGE 108
    
```

Listing 2-1.

```

NONLINEAR
LOAD STEP = 1.00000E+00
STRESSES IN HYPERELASTIC QUADRILATERAL ELEMENTS (QUAD4)
ELEMENT GRID/ POINT -----CAUCHY STRESSES----- PRINCIPAL STRESSES (ZKRO SHEAR)
ID GAUSS ID NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR
1 GAUS 1 1.678579E+02 -1.188351E-03 1.560323E-16 .0000 1.678579E+02 -1.188351E-03
2 1.678579E+02 -1.188351E-03 -1.191492E-15 .0000 1.678579E+02 -1.188351E-03
3 1.678579E+02 -1.188351E-03 -3.368891E-16 .0000 1.678579E+02 -1.188351E-03
4 1.678579E+02 -1.188351E-03 -7.083336E-16 .0000 1.678579E+02 -1.188351E-03
STRESSES IN HYPERELASTIC AXISYMM. QUADRILATERAL ELEMENTS (QUAD4)
ELEMENT GRID/ POINT -----CAUCHY STRESSES----- PRINCIPAL STRESSES (ZKRO SHEAR)
ID GAUSS ID RADIAL(X) AXIAL(Y) SHEAR-XY ANGLE MAJOR MINOR
1 GAUS 1 1.808360E-09 2.917973E+02 4.298727E-15 90.0000 2.917973E+02 1.808360E-09
2 1.808132E-09 2.917973E+02 -2.820565E-17 -90.0000 2.917973E+02 1.808132E-09
3 1.808019E-09 2.917973E+02 -2.820565E-17 -90.0000 2.917973E+02 1.808019E-09
4 1.808218E-09 2.917973E+02 1.004509E-16 90.0000 2.917973E+02 1.808189E-09
NONLINEAR SUBCASE 100 $ UNIAXI

LOAD STEP = 1.00000E+00
STRESSES IN HYPERELASTIC HEXAHEDRON ELEMENTS (HEXA4)
ELEMENT-ID GRID/ POINT -----CAUCHY STRESSES----- DIR. COSINES MEAN
1 GAUSS ID NORMAL SHRRAR PRINCIPAL -A- -B- -C- PRESSURE
1 X 2.917973E+02 XY -7.841313E-15 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804707E-09 YZ -7.742389E-17 B 1.804693E-09 LY .00 .0 .0
Z 1.804707E-09 ZX 2.332978E-15 C 1.804693E-09 LZ .00 .0 .0
2 X 2.917973E+02 XY -3.102808E-16 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804707E-09 YZ -1.485038E-16 B 1.804693E-09 LY .00 .0 .0
Z 1.804707E-09 ZX 9.636988E-16 C 1.804693E-09 LZ .00 .0 .0
3 X 2.917973E+02 XY 2.823466E-15 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804707E-09 YZ 6.255328E-17 B 1.804707E-09 LY .00 .0 .0
Z 1.804707E-09 ZX -6.466815E-15 C 1.804707E-09 LZ .00 .0 .0
4 X 2.917973E+02 XY -5.997311E-16 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804707E-09 YZ 9.181191E-17 B 1.804707E-09 LY .00 .0 .0
Z 1.804707E-09 ZX -2.187819E-15 C 1.804707E-09 LZ .00 .0 .0
5 X 2.917973E+02 XY 2.333811E-15 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804707E-09 YZ -1.264558E-16 B 1.804707E-09 LY .00 .0 .0
Z 1.804707E-09 ZX 2.823466E-15 C 1.804707E-09 LZ .00 .0 .0
6 X 2.917973E+02 XY -2.335867E-16 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804693E-09 YZ -1.129638E-16 B 1.804694E-09 LY .00 .0 .0
Z 1.804693E-09 ZX 2.138826E-15 C 1.804694E-09 LZ .00 .0 .0
7 X 2.917973E+02 XY 2.162630E-15 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804665E-09 YZ 2.140466E-17 B 1.804636E-09 LY .00 .0 .0
Z 1.804665E-09 ZX -6.248027E-15 C 1.804636E-09 LZ .00 .0 .0
8 X 2.917973E+02 XY -4.047673E-16 A 2.917973E+02 LX 1.00 .0 .0 9.726578E+01
Y 1.804651E-09 YZ 2.496398E-17 B 1.804622E-09 LY .00 .0 .0
Z 1.804651E-09 ZX -1.969031E-15 C 1.804622E-09 LZ .00 .0 .0
LOAD STEP = 1.00000E+00

```

Listing 2-2.

Guidelines

Simcenter Nastran performs a fully nonlinear analysis, including the effect of large strain and large rotation, if a PLPLANE and/or PLSOLID Bulk Data entry are encountered. The hyperelastic material is defined by using the MATHP Bulk Data entry. A nonlinear analysis is always performed except in linear solutions when a hyperelastic material is defined.

It is best that the differential stiffness PARAM,LGDISP,1 be used in all hyperelastic runs, but it is not the default. If the strains and the rotations are small, then the hyperelastic analysis should compare to the linear elastic analysis. However, for hyperelastic analysis, it is always best to use PARAM,LGDISP,1.

Due to the independent pressure interpolations, particularly at very high compressive values of the pressure (of the order of the bulk modulus), indefiniteness of the stiffness matrix may occur, caused by an apparent material instability. By default in such instances Simcenter Nastran will ignore the so-called differential stiffness. Although indefiniteness of the stiffness matrix is probably not associated with a geometric instability, this may indeed help, as many of the terms, which heavily depend on the pressure, are part of the differential stiffness.

Limitations

The fully nonlinear hyperelastic CQUAD, CQUAD4, CQUAD8, CTRIA3, and CTRIA6 are plane strain elements only; they do not contain bending properties, nor can they be used in plane stress analysis.

Therefore, they cannot be subject to out-of-plane loading conditions and do not sustain a thickness. The elements assume a thickness of 1.0 for mass calculation in dynamic analyses.

The lower order fully nonlinear hyperelastic elements CQUAD4, CTRIA3, 8-noded CHEXA, 6-noded CPENTA and 4-noded CTETRA are not immune to shear locking and therefore are not suitable for bending type loading. The higher order elements CQUAD8, 9-noded CQUAD, CTRIA6, 20-noded CHEXA, 15-noded CPENTA, and 10-noded CTETRA should be used in such situations. See **Example 1: Rubber Bushing Problem**.

Even though the fully nonlinear elements are defined using the current CQUAD4, CQUAD8, CTRIA3, CTRIA6, CHEXA, CPENTA, and CTETRA Bulk Data entries, internally, Simcenter Nastran designates them as the separate elements, CQUAD4FD, CHEXAFD (for Finite Deformation) etc., because of the different element formulations. This designation is printed in all the output in which the elements are referenced by name, such as PARAM,EST, PARAM,GPECT, the sequence processor, and the element stress output.

GPSTRESS and FORCE output is not available for the fully nonlinear hyperelastic elements.

The element types for the fully nonlinear hyperelastic elements are as follows:

Element	Element Number	Element Number in OESNL Data Block
CQUAD4	139	201
CQUAD8 (Reverts to CQUAD4 when all midside grids are omitted)	164	208
CQUAD (Reverts to CQUAD4 when center and all midside grids are omitted)	164	208
CTRIA3	162	206
CTRIA6 (Reverts to CTRIA3 when all midside grids are omitted)	167	211
CHEXA with 8 nodes	140	202
CHEXA (Reverts to 8-noded CHEXA when all midside grids are omitted)	163	207
CPENTA with 6 grids	160	204
CPENTA (Reverts to 6-noded CPENTA when all midside grids are omitted)	165	209
CTETRA	161	205
CTETRA (Reverts to 4-noded CTETRA when all midside grids are omitted)	166	210
CQUADX4	251	262
CQUADX8 (Grid option only)	253	264
CQUADX8 (Gauss option only)	253	266
CTRAX3	250	261
CTRAX6	252	263

Example 1: Rubber Bushing Problem

The cross section of a rubber bushing is shown in [Figure 2-4](#). It is assumed that the frame and internal shaft are rigid and the rubber bushing is perfectly bonded to these components. The rubber material is assumed to be of the Mooney-Rivlin type with $A_{10} = 0.177 \text{ N/mm}^2$, $A_{01} = 0.045 \text{ N/mm}^2$, and $D_1 = 333 \text{ N/mm}^2$. Considering symmetric conditions, only one-half of the rubber bushing is modeled with 72 finite deformation 4-noded CQUAD elements. The same problem is also modeled with 72 hyperelastic 9-noded CQUAD elements. The grid points on the outer boundary are fully constrained to simulate the rubber/frame interface. For the grid points on the inner boundary, only the horizontal degrees-of-freedom were constrained, while the vertical degrees-of-freedom were tied together with MPCs. Force is applied to the top grid point on the inner wall in the vertical direction.

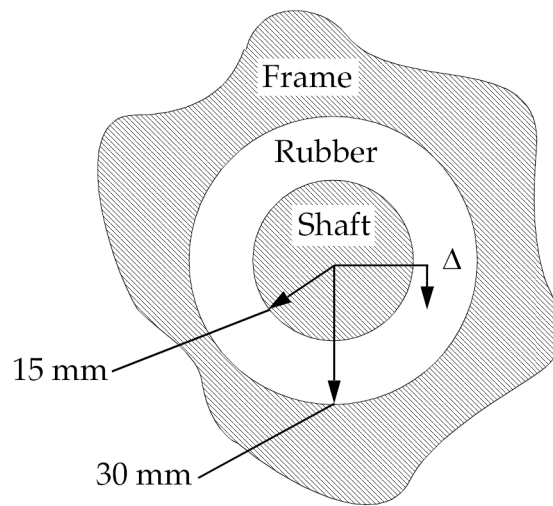


Figure 2-4. Rubber Bushing

The goal of the analysis is to determine the force-displacement curve of the unit and compare it with the solution in the reference. (Reference: Sussman, T., and Bathe, K. J., "A Finite Element Formulation for Nonlinear Incompressible Elastic and Inelastic Analysis," *Computers and Structures* (26), pages 357-409, 1987.)

Annotated Bulk Data input of interest is shown in [Listing 2-3](#).

```

ID RDS,BUSHING $
SOL 106
TIME 180
CHMD
TITLE = RUBBER BUSHING                      BUSHING
SUBTITLE = HYPERELASTIC MODEL TEST
ECHO=UNSORT
DISP(PLOT)=ALL
STRESS(PLOT)=ALL
SUBCASE 100
MFC = 100
MLPARM = 100
LOAD = 100 $
.
.
SUBCASE 800
MFC = 100
MLPARM = 800
LOAD = 800 $
BEGIN BULK
$ PARAMETERS
PARAM,DEDROPT,0
PARAM,POST,0
PARAM,LGDISP,1
$
NLPARM 100 5          ITER 1          NO          +ML101
+ML101          -3          +ML102
.
.
NLPARM 800 5          ITER 1          NO          +ML801
+ML801          -3          +ML802
$ DEFINE GEOMETRY
CORD2C 1          0. 0. 0. 0. 0. 1.  +COR1
+CORD 1. 0. 0.
GRDSET
GRID 101 1 15.0 -90. 0. 13456
GRID 102 1 17.5 -90. 0. 13456
GRID 103 1 20.0 -90. 0. 13456
.
.
$ DEFINE ELEMENTS
CQUAD4 101 1 101 102 202 201
CQUAD4 102 1 102 103 203 202
CQUAD4 103 1 103 104 204 203
.
.
$ PROPERTY ENTRY FOR 4-NODE PLANE STRAIN ELEMENT WITH MIXED ELEMENT FORMULATION
PLPLANE 1 1
$ HYPERELASTIC MATERIAL - MOONEY RIVLIN TYPE
MATHP 1 0.177 0.045 333.3
$ DEFINE LOADING CONDITIONS
FORCE 100 1301          -50. 0. 1. 0.
.
.
FORCE 800 1301          -400. 0. 1. 0.
$ DEFINE CONSTRAINTS
MPC 100 101 2 1. 1301 2 -1.
.
.
ENDDATA

```

Listing 2-3.

The resulting force-displacement curve at grid point 1301 is shown in [Figure 2-5](#) and the original and deformed models at a force of 800N are shown in [Figure 2-6](#). The solution obtained with 4-noded CQUAD elements is overly stiff for a force greater than 200N (shear locking). The shear locking problem is addressed with the higher order 9-noded CQUAD elements.

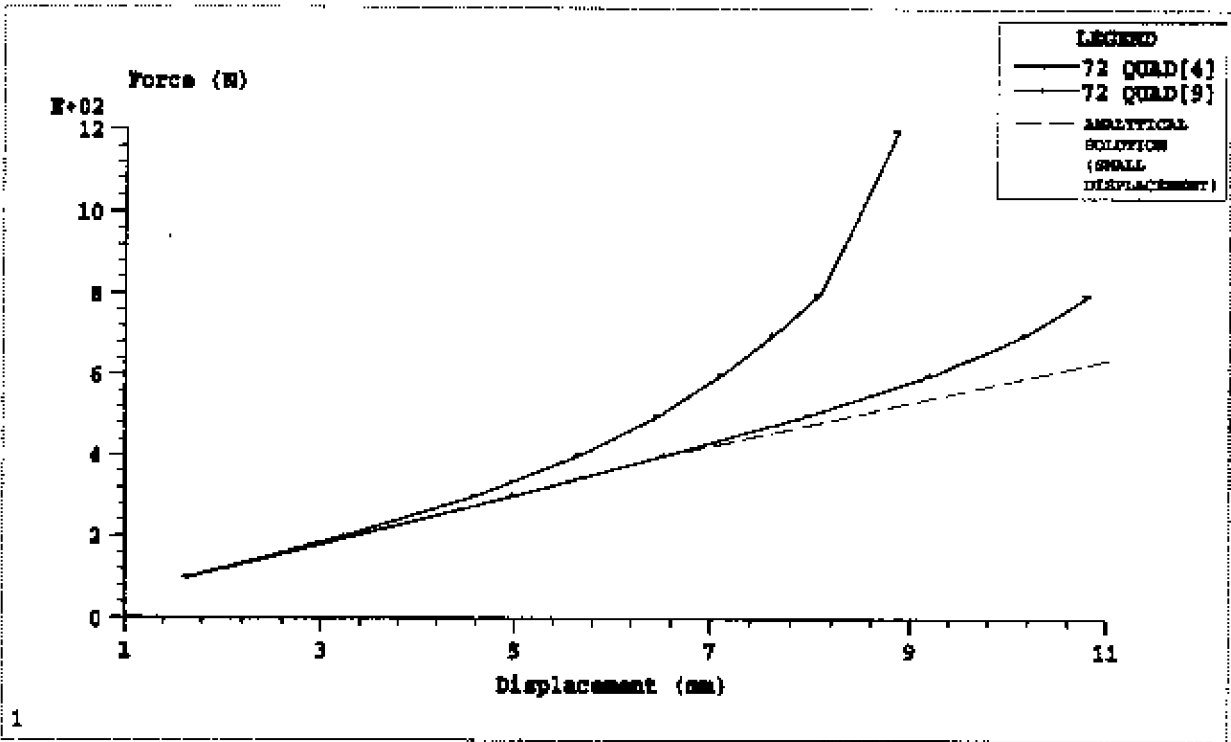


Figure 2-5. Force-Displacement Curve of a Rubber Bushing

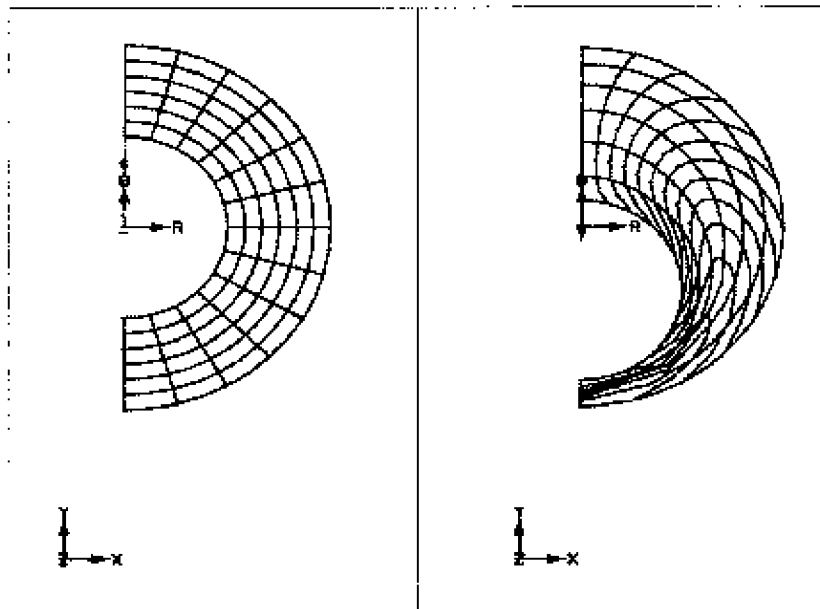


Figure 2-6. Initial and Deformed Shapes of a Rubber Bushing

The model file for this example is *bushing.dat*, which can be found in the examples directory (see *Simcenter Nastran Installation and Operations Guide*).

Example 2: Pressure of a Circular Plate

A circular plate is 15 in. in diameter and 0.5 in. thick. It is simply supported along the edge and is subjected to a uniform follower pressure. The plate is modeled with solid (CHEXA and CPENTA) and with axisymmetric elements. The solid model represents a 10° segment. Material constants are $A_{10} = 80$ psi, $A_{01} = 20$ psi, $D_1 = 50,000$ psi.

Reference: J. T. Oden, *Finite Elements of Nonlinear Continua*, McGraw-Hill, 1972.

Annotated Bulk Data input of interest for the wedge model is shown in [Listing 2-4](#) and for the axisymmetric model is shown in [Listing 2-5](#).

```

BEGIN BULK
$ DEFINE PARAMETERS
PARAM, LGDISP, 1
PARAM, DBDRNL, -1
PARAM, POST, 0
NLPARM 10      10          ITER 1      100          +NL10
+NL10      -3
.
.
.
$ DEFINE GEOMETRY
CORD2C 1      0.      0.      0.      0.      0.      1.      +COR1
+COR1 1.      0.      0.
GRDSET
GRID 101      1      0.      -5.      -0.25      1      12456
GRID 102      1      0.75      -5.      -0.25      1
GRID 103      1      1.50      -5.      -0.25      1
.
.
.
$ DEFINE ELEMENTS
CHEXA 102      1      102      103      1103      1102      202      203      +HX102
+HX102 1203      1202
CHEXA 103      1      103      104      1104      1103      203      204      +HX103
+HX103 1204      1203
CHEXA 104      1      104      105      1105      1104      204      205      +HX104
+HX104 1205      1204
.
.
.
CPENTA 101      1      101      102      1102      201      202      1202
CPENTA 201      1      201      202      1202      301      302      1302
CPENTA 301      1      301      302      1302      401      402      1402
CPENTA 401      1      401      402      1402      501      502      1502
.
.
.
$ DEFINE ELEMENT AND MATERIAL PROPERTIES
PLSOLID 1      1
MATHP 1      80.      20.      5.+4
$ DEFINE LOADING CONDITIONS
LOAD 3      1.      3.      1
LOAD 15      1.      15.      1
.
.
.
LOAD 45      1.      45.      1
PLOAD4 1      101      1.
PLOAD4 1      102      1.      101      1103
.
.
.
$ DEFINE CONSTRAINTS
MPC 10      1102      1      1.      102      1      -1.
MPC 10      1102      3      1.      102      3      -1.
.
.
.
ENDDATA

```

Listing 2-4.


```

$ INFLATION OF A CIRCULAR PLATE
$ AXISYMMETRIC QUAD4 ELEMENTS
BEGIN BULK
$ DEFINE PARAMETERS
PARAM,LGDISP,1
PARAM,POST,0
NLPARM 10      10          AUTO 1          +NL10
+NL10          -3
.
.
$ DEFINE GEOMETRY
GRDSET
GRID 101      0.      -0.25  0.      3456
GRID 102      0.75   -0.25  0.      13456
GRID 103      1.50   -0.25  0.
.
.
$ DEFINE ELEMENTS
CQUADX 101    1      101    102    202    201
CQUADX 102    1      102    103    203    202
CQUADX 103    1      103    104    204    203
.
.
$ DEFINE ELEMENT AND MATERIAL PROPERTIES
PLPLANE 1      1
MATHP 1      80.    20.    5.+4
$ DEFINE LOADING CONDITIONS
PLOADX1 3      101    3.      101    102
PLOADX1 3      102    3.      102    103
.
.
PLOADX1 45     110    45.      110    111
ENDDATA

```

Listing 2-5.

Original and deformed models are shown in [Figure 2-7](#) and [Figure 2-8](#).

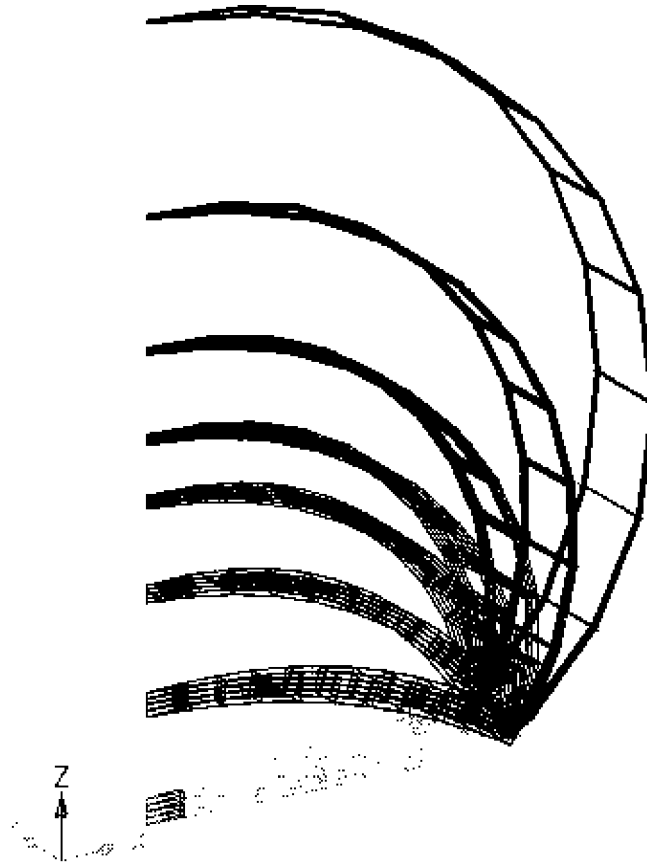


Figure 2-7. Deformed Shape of Circular Plate Under Pressure – CHEXA and CPENTA Hyperelastic Elements

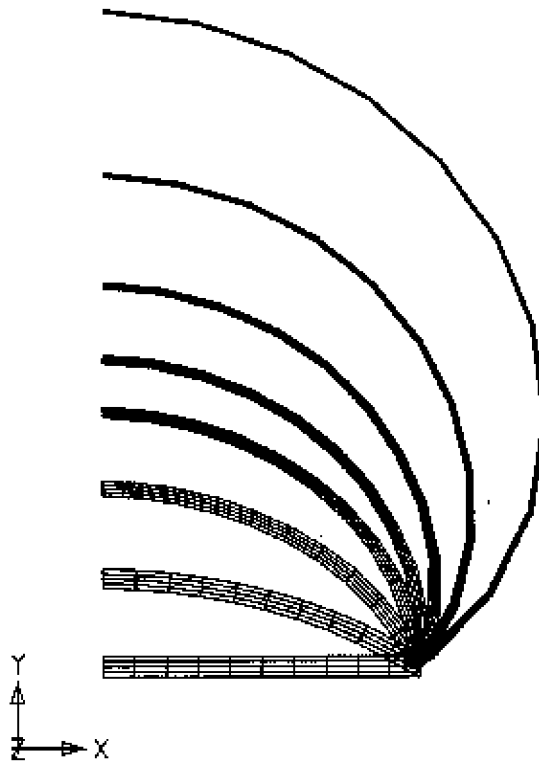


Figure 2-8. Deformed Shape of Circular Plate Under Pressure – Axisymmetric Hyperelastic Elements

The model files for this example are `prplth.dat` and `prpltg.dat`, which can be found in the examples directory (see *Simcenter Nastran Installation and Operations Guide*).

2.4 Bushing Elements

The CBUSH element is a generalized spring and damper. Stiffness and damping can be specified for 6 degrees-of-freedom, 3 translations and 3 rotations. In SOL 106 and 129, you can specify 6 nonlinear force-deflection curves instead of combined stiffness values. The properties of the CBUSH element are defined on the PBUSH and PBUSHi entries. The PBUSHT entry refers to TABLEDi entries for force-deflection curves. The CBUSH element is geometric linear only. The circulation is not updated.

The CBUSH1D element is a rod-type spring and damper, it is a 1D version of the CBUSH element. The CBUSH1D element can be geometric and material nonlinear. General nonlinear functions of the axial forces versus axial displacement and for velocity are defined with the PBUSH1D entry. The CBUSH1D element is the only element which can model rotation damping. The effect is demonstrated in the following example.

The CBUSH and CBUSH1D elements are recommended over the CELASi, CVISC, and CDAMP elements. The bushing elements always insure rigid body invariance and they offer nonlinear capabilities.

The STRESS case control command should be used to request the element force and stress output for the CBUSH and CBUSH1D elements.

Swinging Pendulum

The solution to a swinging pendulum is calculated using SOL 129 of Simcenter Nastran Version 70.5+. The pendulum has a length of 1.0 [m], and has a concentrated mass of $M = 1,000.0$ [kg] at the free end, see [Figure 2-9](#). We start the analysis with the pendulum at rest in horizontal position. The free end is loaded with a gravity induced load of $G = 10,000.0$ [N]. The leg of the pendulum is very stiff, $K = 1.e+7$ [N/m], so that the relative axial deformation is small compared to the overall motion of the pendulum. Large deformation effects are turned on with `PARAM,LGDISP,1`.

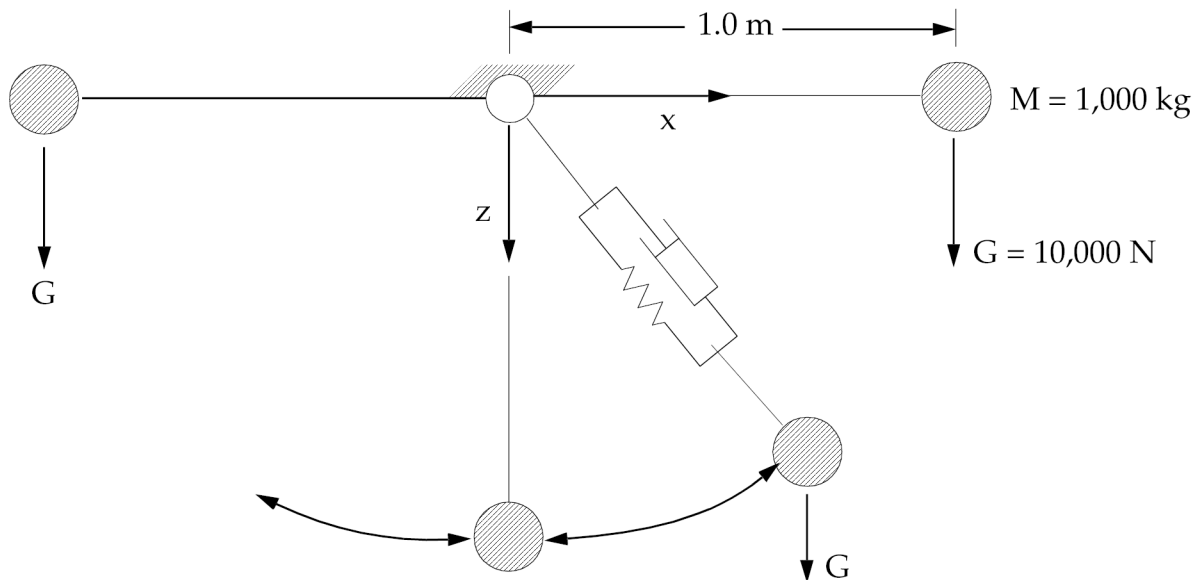


Figure 2-9. Swinging Pendulum

```

$
$ Name: ar29src1
$
$ A swinging pendulum is modeled. The pendulum is loaded with
$ gravity load. The analysis goes through one and a half cycles,
$ starting in horizontal position. Large deformation is turned on.
$
$ The pendulum is modeled with two separate systems.
$ 1. CROD and CVISC, CROD rotates, CVISC stays in horizontal
$ orientation.
$ 2. CBUSH1D with SPRING and DAMPER, spring and damper rotate.
$
TIME 10 $
SOL 129 $
CEND
$
TITLE= ar29src1 swinging elastic pendulum
SUBTITLE= demonstrate rotating damping
.
.
.

```

```

$
BEGIN BULK
$
param, lgdisp, 1
$
$ GEOMETRY
$
.
.
.
$ CBUSH1D spring and rotating viscous damper
$
$CBUSH1D,EID,GA,GB,CID
cbush1d, 108, 208, 41, 42
$PBUSH1D,PID,K,C
pbush1d, 208, 1.e+7, 1000.
$
$          $          $          $          $          $          $          $          $
$ SOLUTION STRATEGY
$
.
.
.
$
ENDDATA

```

Listing 2-6. Simcenter Nastran Input File for the Swinging Pendulum

We investigate two different models of the pendulum in one run. In the first model, we use a Rod element for the stiffness of the leg, $K = 1.e+7$ [N/m], and a Visc element for 5% equivalent viscous damping, $C = 1,000.0$ [Ns/m]. The x- and z-displacements of the free end are shown in [Figure 2-10](#). The Rod rotates while the viscous damper Visc stays fixed in its initial horizontal position. The Visc element is linear and does not follow large deformations. The displacements are damped out because the viscous damper is acting in the x-direction during the whole motion.

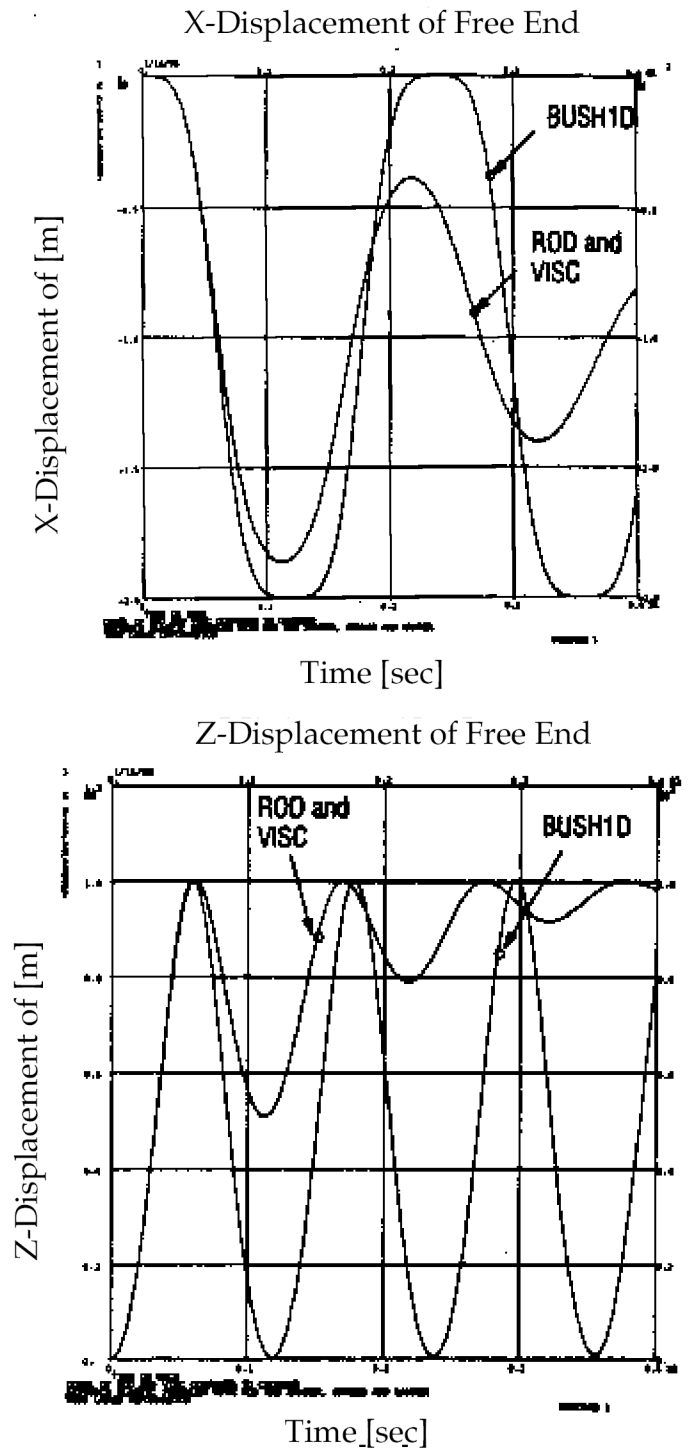


Figure 2-10. Displacement of Swinging Pendulum, Model with Rod/Visc Element Versus Model with BUSH1D Element

In the second model, we use a BUSH1D element. The element has a linear stiffness (K) and a viscous damper (C) with the same values as in the first model. In the BUSH1D element, the spring and damper rotate. The relative axial displacements and velocities are small because of the high axial stiffness of the element. The rotating damper has no noticeable effect on the overall motion of

the pendulum because it damps only the small relative deformations. The x- and z-displacements of the free end are not damped, see [Figure 2-10](#).

The example demonstrates how the answers can change if dampers rotate with the deformation compared to dampers which stay fixed in space.

2.5 Adaptive Gap Element

The gap element is used in Simcenter Nastran to model surface contact problems. The adaptivity is based on the subincremental scheme with the capability to update the stiffness at the proper timing, bisect the increment when it is excessive, and adjust the penalty stiffness to the proper values. Generally, in a non-adaptive gap element, the axial stiffness has to be properly chosen in order to reach convergence. However, the main goal of the self-adaptive method is to relieve users from this trial and error method and to achieve convergence on the first try. Due to the adaptive nature of the algorithm, the adaptive gap method costs more in CPU time when compared to the non-adaptive gap element. However, the adaptive gap method allows a user to choose a wider latitude of axial stiffness and yet still achieve convergence. Therefore, with more experience, you can decrease CPU time by fine-tuning this axial stiffness.

Usage

The gap element is used in nonlinear static analysis (SOL 106) and nonlinear transient analysis (SOL 129). The gap element provides point-to-point contact. When the gap element is open, there is no contact and no friction. When the gap element is closed, there are three different conditions. The first case is when the gap is sliding (no friction). The second case occurs when the gap element is sticking (static friction). The third case occurs when the gap element is slipping (kinetic friction). Sometimes, it may be difficult to converge because of the switching among these conditions.

Limitations

One of the contact surfaces should not rotate by a large angle because the gap element orientation is not updated for large rotations.

How to Use the CGAP and PGAP Bulk Data Entries

Often a mistake is made in the orientation of the gap element, which is defined through G0 or X1, X2, and X3. If the grid points GA and GB are not coincident, then line AB is the element x-axis and the orientation vector lies in the X-Y plane. The use of coincident grid points is not recommended.

On the PGAP Bulk Data entry, U0 and F0 are the initial gap opening and the preload, respectively. U0 is the separation of the gap element. If you are unsure about what preload to use, use the default value of zero. KA, KB, and KT are called the penalty values. KA is the axial stiffness when the gap is closed, KB is the axial stiffness when the gap is open, and the KT is the transverse gap stiffness. All you need to input is the KA value. KB is defaulted to be zero, and KT is defaulted as a function of KA. The selection of KA or a penalty value is critical to the convergence of the problem. The non-adaptive gap does not update this KA value. The new adaptive gap updates this KA value if you specify a positive value for TMAX. The default option for the new adaptive gap is NO adaptive stiffness update. This is when the TMAX value is equal to 0.0. If TMAX is a negative number, then Simcenter Nastran uses the non-adaptive gap formulation. A good initial KA selection is roughly 1000 times the Young's modulus of the material that is in the neighborhood of the gap element.

Fields 8 and 9 also require some explanation. For the new adaptive value this is MU1, which is the static coefficient of friction, and MU2, which is the dynamic coefficient of friction. The non-adaptive gap elements use MUY and MUZ in the same fields instead of MU1 and MU2 values. Therefore, to compare the results using adaptive and non-adaptive gap elements, set both of these entries to the same values. On most problems, you will not know how to estimate the penetration depth or field 2 on the continuation entry. Therefore, if you skip the continuation entry, the default new adaptive gap is obtained, which is everything adaptive except the KA update.

The recommended allowable penetration of TMAX is 10 percent of the thickness for the shell element, and for the solid element it is roughly 1/10,000 of the characteristic length for the problem. A small positive value will turn on the adaptive stiffness update, which will then converge at a faster rate.

The maximum adjustment ratio or MAR in field 3 of the continuation entry is used only for the penalty value adjustment of the adaptive gap element. The default value of 100 is sufficient for most problems.

Output

The gap element output is obtained by the STRESS output request in the Case Control. An example of the gap output format is shown in [Listing 2-7](#). The load step is 1.95 on the top left-hand corner. The digit on the left-hand side of the decimal indicates that subcase 1 is completed. The digits on the right-hand side of the decimal point indicate that 95 percent of subcase 2 is completed. The far left column lists the gap element numbers. The second column provides the forces in the axial direction of the gap element. The third and the fourth columns represent forces in the Y and Z direction. These forces should be zero if the gap is in the open position. Columns 5 to 9 represent the displacements of the gap in its element coordinate system. The last column indicates the status of the gap. This status can be “open”, “stick”, or “slip”.


```

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID          N36603          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 177
  USE TMAX=0.0016,FULLY ADAPTIVE FEATURE
  KA=1.0E6
  LOAD STPF = 1.92500E+00
  SUBCASE 2
  STRESSES (FORCES) IN GAP ELEMENTS (CGAP)

ELEMENT - FORCES IN ELEM SYST - - DISPLACEMENTS IN ELEM SYST -
  ID   COMP-X   SHEAR-Y   SHKR-Z   AXIAL-U   TOTAL-V   TOTAL-W   SLIP-V   SLIP-W   STATUS
2001  3.60382E+00 -4.47466E-01  0.0      2.23604E-02 -8.30369E-04  0.0      -6.81213E-04  0.0      STICK
2003  5.04829E+00 -1.10082E+00  0.0      8.85048E-02 -7.80763E-03  0.0      -7.44069E-03  0.0      STICK
2005  0.0           0.0           0.0      1.95640E-01 -1.70540E-02  0.0      -1.70540E-02  0.0      OPEN
2007  0.0           0.0           0.0      2.23109E-01 -9.91112E-03  0.0      -9.91112E-03  0.0      OPEN
2009  0.0           0.0           0.0      2.40853E-01 -4.95583E-03  0.0      -4.95583E-03  0.0      OPEN

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID          N36603          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 178
  USE TMAX=0.0016,FULLY ADAPTIVE FEATURE
  KA=1.0E6
  LOAD STPF = 1.95000E+00
  SUBCASE 2
  STRESSES (FORCES) IN GAP ELEMENTS (CGAP)

ELEMENT - FORCES IN ELEM SYST - - DISPLACEMENTS IN ELEM SYST -
  ID   COMP-X   SHEAR-Y   SHKR-Z   AXIAL-U   TOTAL-V   TOTAL-W   SLIP-V   SLIP-W   STATUS
2001  3.64258E+00 -4.47838E-01  0.0      2.23643E-02 -8.30493E-04  0.0      -6.81213E-04  0.0      STICK
2003  5.15982E+00 -1.12016E+00  0.0      8.85160E-02 -7.81408E-03  0.0      -7.44069E-03  0.0      STICK
2005  8.83100E-02 -5.82848E-03  0.0      1.97000E-01 -1.73532E-02  0.0      -1.73532E-02  0.0      STICK
2007  0.0           0.0           0.0      2.25831E-01 -1.01834E-02  0.0      -1.01834E-02  0.0      OPEN
2009  0.0           0.0           0.0      2.44078E-01 -5.09924E-03  0.0      -5.09924E-03  0.0      OPEN

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID          N36603          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 179
  USE TMAX=0.0016,FULLY ADAPTIVE FEATURE
  KA=1.0E6
  LOAD STPF = 1.97500E+00
  SUBCASE 2
  STRESSES (FORCES) IN GAP ELEMENTS (CGAP)

ELEMENT - FORCES IN ELEM SYST - - DISPLACEMENTS IN ELEM SYST -
  ID   COMP-X   SHEAR-Y   SHKR-Z   AXIAL-U   TOTAL-V   TOTAL-W   SLIP-V   SLIP-W   STATUS
2001  3.66578E+00 -4.47205E-01  0.0      2.23666E-02 -8.30282E-04  0.0      -6.81213E-04  0.0      STICK
2003  5.22790E+00 -1.12097E+00  0.0      8.85228E-02 -7.81435E-03  0.0      -7.44069E-03  0.0      STICK
2005  2.37324E-01 -3.59185E-02  0.0      1.97024E-01 -1.73652E-02  0.0      -1.73532E-02  0.0      STICK
2007  0.0           0.0           0.0      2.27630E-01 -1.03924E-02  0.0      -1.03924E-02  0.0      OPEN
2009  0.0           0.0           0.0      2.46395E-01 -5.21857E-03  0.0      -5.21857E-03  0.0      OPEN

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID          N36603          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 180
  USE TMAX=0.0016,FULLY ADAPTIVE FEATURE
  KA=1.0E6
  LOAD STPF = 2.00000E+00
  SUBCASE 2
  STRESSES (FORCES) IN GAP ELEMENTS (CGAP)

ELEMENT - FORCES IN ELEM SYST - - DISPLACEMENTS IN ELEM SYST -
  ID   COMP-X   SHEAR-Y   SHKR-Z   AXIAL-U   TOTAL-V   TOTAL-W   SLIP-V   SLIP-W   STATUS
2001  3.68888E+00 -4.46565E-01  0.0      2.23689E-02 -8.30068E-04  0.0      -6.81213E-04  0.0      STICK
2003  5.29569E+00 -1.12167E+00  0.0      8.85296E-02 -7.81458E-03  0.0      -7.44069E-03  0.0      STICK
2005  3.86731E-01 -6.51561E-02  0.0      1.97039E-01 -1.73749E-02  0.0      -1.73532E-02  0.0      STICK
2007  0.0           0.0           0.0      2.29423E-01 -1.06010E-02  0.0      -1.06010E-02  0.0      OPEN
2009  0.0           0.0           0.0      2.48707E-01 -5.33770E-03  0.0      -5.33770E-03  0.0      OPEN
  
```

Listing 2-7. Sample of CGAP Output

Example: Hertz Contact Stress

Hertz contact stress is a well known problem arising in the contact of a wheel and a rail. To guard against the possibility of surface failure, it is necessary to calculate the stress that results from loading one body against another. The gap element is used to model the point-to-point contact phenomenon. In this case, the gap element represents the contact surface. The Hertz contact stress problem considered here is a nonlinear static analysis using SOL 106.

Modeling Techniques

For the static analysis, a symmetric half of a 10-degree sector of the sphere is modeled by CHEXA and CPENTA elements with axisymmetric boundary conditions as shown in Figure 2-11. Contact with a rigid plane is simulated by adaptive gap elements.

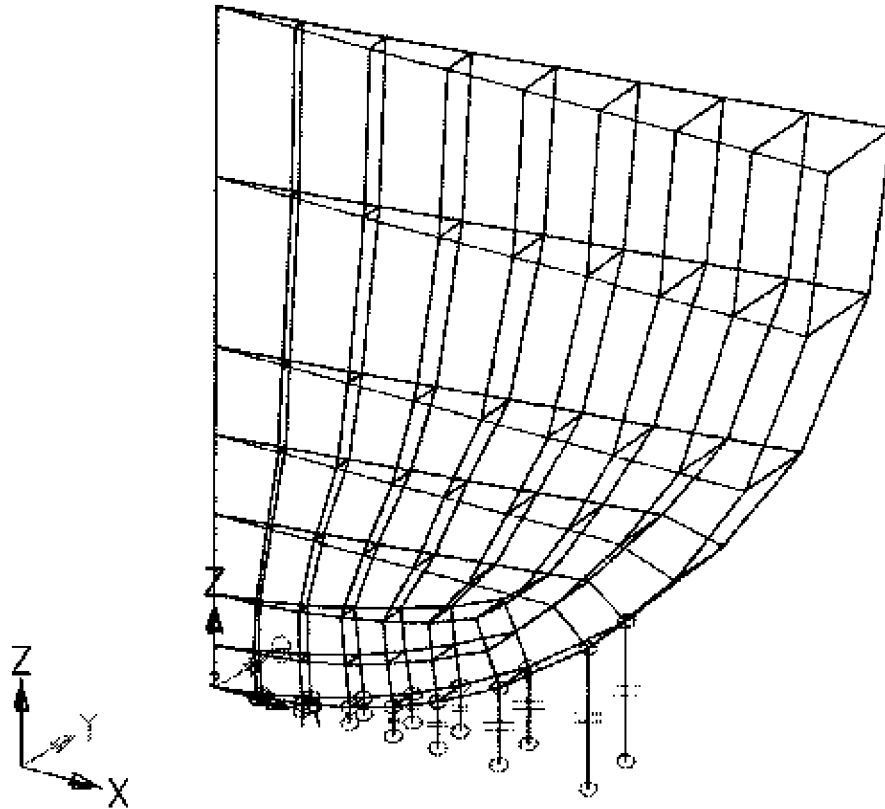


Figure 2-11. Hertz Contact Stress

Uniform vertical displacement of the symmetric plane is maintained by constraining the vertical movement of all the nodes on the top plane by MPC relations at the center line. The radius of the sphere is 8 inches, Young's modulus is 1000 psi, and Poisson's ratio is 0.3. The static analysis is performed in 17 increments up to the maximum loading. The characteristic length of the problem is the diameter of the sphere, i.e., 16 inches. Therefore, the TMAX value is 16/10000, or 0.0016. The initial KA or axial gap stiffness is selected to be 1000 times Young's modulus. These values are inserted in the PGAP entry as shown in [Figure2-12](#).

1	2	3	4	5	6	7	8	9	10
\$	EID	PID	GA	GB	X1	X2	X3		
CGAP	2001	1	105	301	1.	0.	0.		

\$	PID	U0	F0	KA	KB	KT	MU1	MU2	
PGAP	1	.022		1.+6			.3	.3	+
\$	TMAX	MAR	TRMIN						
+	0.0016								

\$	TMAX	MAR	TRMIN						
+	0.0016								

Figure 2-12. Partial Input Data

Discussion of Results

The objective of this example is to demonstrate that the adaptive features of the gap element allows you to use a wider range of KA values, and still get good results. Figure 2-13 shows that if the adaptive features of gap element are used, good results are obtained for KA values ranging from 1.0E1 to 1.0E6. If adaptive features are not used, good results are obtained for KA values between 1.0E3 and 1.0E5. For KA values higher than 1.0E5 non-adaptive gap element failed to converge, and below 1.0E3, the accuracy decreased.

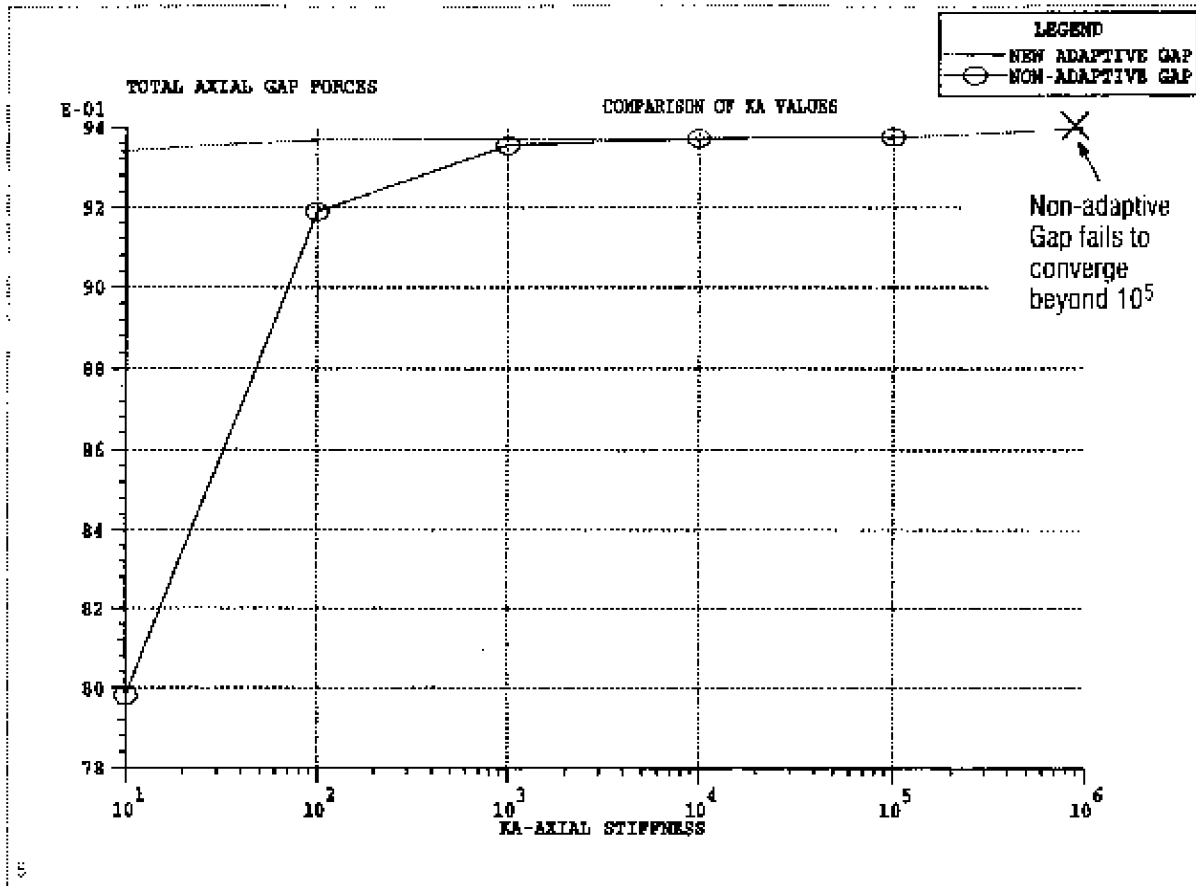


Figure 2-13. Comparison of Axial Stiffness Values

Chapter 3: Material and Geometric Nonlinearity

3.1 Overview of Material Nonlinearity

Material nonlinearity becomes an issue in your analysis if the stress-strain relationship is nonlinear. Constitutive equations characterize the macroscopic behavior of the material, which stems from its microscopic constitution. All engineering materials are inherently nonlinear. However, it isn't feasible to characterize a nonlinear material by a single set of equations for the entire range of environmental conditions, such as loading, temperature, and rate of deformation. Depending on the application or the situation, the material behavior may be idealized or simplified to account for certain effects, which are important in the analysis.

The linear elastic (Hookean) material is the simplest case of all. As the strain exceeds the infinitesimal range, the material exhibits nonlinear behavior:

- nonlinear elastic behavior if it is recoverable
- plastic behavior if it is irrecoverable

In Simcenter Nastran, you can use the following types of nonlinear materials:

- Nonlinear elastic
- Hyperelastic
- Elastic-plastic
- Visco-elastic
- Temperature dependent

3.2 Nonlinear Elastic Material

Typical stress-strain relationship for a nonlinear elastic material is shown in [Figure 3-1](#). Notice that both the loading and unloading occur along the same path.

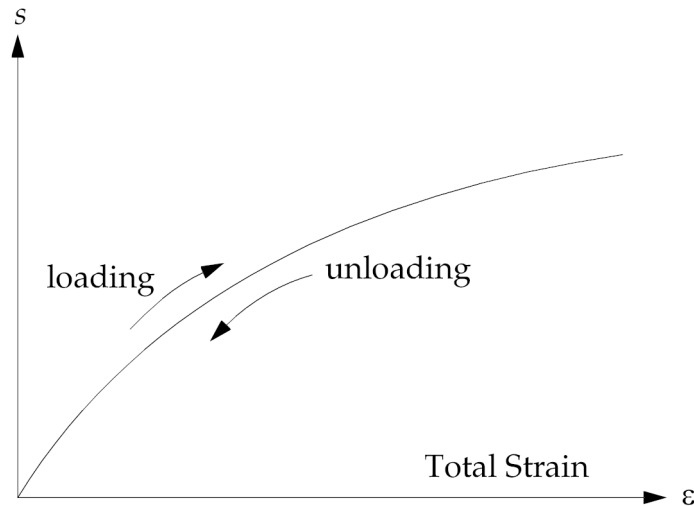


Figure 3-1. Nonlinear Elastic Material

The nonlinear elastic theory implemented is useful to predict multiaxial stress state for materials for which only the tension and compression stress-strain data is available. The stress-strain data in tension may be different from that in compression.

The theory implemented in Simcenter Nastran is adequate only for small strains (less than 20%) in multiaxial stress state as it is not based on the classical theory of finite elasticity. For large strains in multiaxial state, you should use a hyperelastic material instead.

Defining Nonlinear Elastic Materials in Simcenter Nastran

In Simcenter Nastran, you can perform nonlinear elastic material analyses using isotropic materials. To define the material properties for a nonlinear elastic analysis, use the MAT1 bulk data entry in conjunction with the MATS1 and TABLES1 entries to define the stress-dependent material properties. On the MATS1 entry, when you're creating a nonlinear elastic material, use field 4 to set the TYPE option to NLELAST.

See Also

- “MAT1” in the *Simcenter Nastran Quick Reference Guide*
- “MATS1” in the *Simcenter Nastran Quick Reference Guide*
- “TABLES1” in the *Simcenter Nastran Quick Reference Guide*

3.3 Hyperelastic Material

You use the hyperelastic material model to analyze elastomers up to a large strain. Elastomers are natural or synthetic rubberlike materials that have the ability to undergo deformation under the influence of a force and regain their original shape once the force is removed (unless damage occurs). They are used extensively in many industries, because of their wide availability and low cost. For example, in the automotive industry, hyperelastic materials would be used to model parts such as o-rings, bushings, gaskets, seals, rubber boots, and tires.

The hyperelastic constitutive model available in Simcenter Nastran is based on a generalized Rivlin strain energy function expressed as follows:

$$U = \sum_{i+j=1}^{N_a} A_{ij}(\bar{I}_1 - 3)(\bar{I}_2 - 3)^j + \sum_{i=1}^{N_d} D_i(J - 1 - \alpha_v(T - T_0))^{2i}$$

Equation 3-1.

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

The strain energy function is a potential for the stresses. For example, the Cauchy stresses; i.e., components of the stress vector

$$\mathbf{t} = \frac{d\mathbf{P}}{dA}$$

where dA is an infinitesimal area in the deformed configuration and P is the force, are obtained as follows:

$$\boldsymbol{\sigma} = \frac{2}{J} \mathbf{F} \frac{\partial U}{\partial \mathbf{C}} \mathbf{F}^T$$

Equation 3-2.

where

$\mathbf{F} = dx/dX$	deformation gradient
x	position of a material point in the deformed configuration
X	position of a material point in the undeformed configuration

$\mathbf{C} = \mathbf{F}^T \mathbf{F}$ is the right Cauchy-Green deformation tensor; $J = \det \mathbf{F}$ is the volume ratio dV/dV_0 ; and \bar{I}_1 and \bar{I}_2 are the first and second distortional strain invariants,

$$\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2 \quad \bar{I}_2 = \bar{\lambda}_1^{-2} + \bar{\lambda}_2^{-2} + \bar{\lambda}_3^{-2}$$

Equation 3-3.

where

$$\bar{\lambda}_I$$

are principal stretches l/l_0 of the distortional deformation; l is the current and l_0 is the original length along a principal direction;

$$\bar{\lambda}_I^2$$

are principal values of

$$\bar{C} = J^{-2/3} C$$

α_v is the volumetric coefficient of thermal expansion; $\alpha_v = 3\alpha$, where α is the coefficient of linear thermal expansion of an isotropic material; T is the current and T_0 is the initial temperature.

The use of a strain energy function implies isotropy in the undeformed configuration. The model can have up to 25 material constants.

Special Cases of the Material Law

Special cases of the material law are:

- Mooney-Rivlin material
- Neo-Hookean material

Mooney-Rivlin Material

With Mooney-Rivlin materials:

$$N_a = 1$$

It is a two parameter model that results in a constant shear modulus G .

Neo-Hookean Material

With Neo-Hookean materials:

$$N_a = 1, A_{01} = 0.0$$

It is a one parameter model that results in a constant shear modulus G .

Defining Hyperelastic Materials in Simcenter Nastran

You use the MATHP material bulk data entry to define a hyperelastic material. The MATHP entry consists of five continuation entries that define the A_{ij} and D_i material constants of the strain energy function. The continuation entries are not required if you're modeling a Neo-Hookean or a Mooney-Rivlin material.

You specify the order of the strain energy polynomial function using the NA and ND fields of the MATHP entry for the distortional and volumetric parts, respectively.

If you don't know the values for the material constants, you can use the curve-fitting algorithm to determine them from experimental data.

Using the Curve Fitting Algorithm to Determine Material Constants

Simcenter Nastran has a curve-fitting algorithm based on least squares fitting that lets you determine the material constants A_{ij} and D_i based on experimental data up to the fifth-order polynomial form (N_a and/or N_d up to 5). To initiate the curve-fitting algorithm, enter either TAB1, TAB2, TAB3, TAB4, or TABD in the sixth continuation entry of the MATP bulk data entry. These TABi fields reference experimental data that is contained in TABLES1 entries.

Simcenter Nastran uses TAB1, TAB2, TAB3, and TAB4 to obtain the material constants A_{ij} . Simcenter Nastran uses the TABD option (volumetric compression) to obtain the material constants D_i . In obtaining the material constants, the software takes compressibility into account, in the case of simple tension/compression and equibiaxial tension, when the volumetric part of the strain energy function is of the order of one.

Once the curve-fitting algorithm computes the material constant, it prints the results in an updated MATHP entry image in the output file. The image is printed using a 16-character field. This allows you to directly input the produced MATHP entry into a data file in large-field format. An example of the produced output is shown in the listing .

Simple Tension/Compression (TAB1 Option)

The simple tension/compression test is characterized by a prescribed axial force or displacement. X values in the TABLES1 entry which contains experimental data must be stretch ratios l/l_0 . Y values must be values of the engineering stress F/A_0 .

Equibiaxial Tension (TAB2 Option)

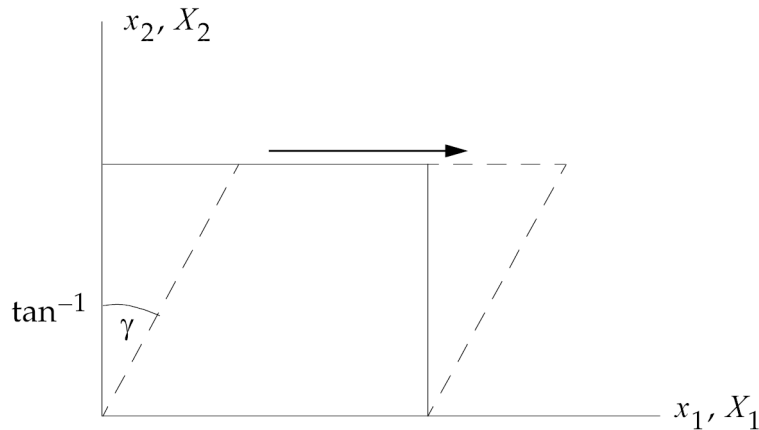
The equibiaxial tension test is characterized by prescribed equal tensile stresses or elongations in two directions and is typically produced by the pressure of a spherical membrane. X values in the TABLES1 entry which contains experimental data must be stretch ratios l/l_0 . Y values must be values of the engineering stress F/A_0 , l = current length, F = current value of the force, l_0 and A_0 = initial length and cross-sectional area, respectively. In the case of pressure of a spherical membrane, the engineering stress is given by $Pr_0\lambda^2/2t_0$ where P = current value of the pressure and r_0 , t_0 = initial radius and thickness.

Simple Shear (TAB3 Option)

The simple shear test is obtained by a prescribed shearing force or displacement of a cube. This is not a homogeneous deformation. It is approximated by one that is homogeneous isochoric as follows:

$$x_1 = X_1 + \gamma X_2 \quad x_2 = X_2 \quad x_3 = X_3$$

Equation 3-4.



where x_1, x_2, x_3 are the coordinates of the deformed body and X_1, X_2, X_3 are the coordinates of the body in its undeformed configuration. You should be aware of this approximation. X values in the TABLES1 bulk data entry which contains experimental data must be values of the shear tangent γ . Y values must be values of the engineering shear stress F/A_0 . It is recommended that simple shear be used along with another test case; otherwise, the coefficients cannot be determined uniquely.

Pure Shear (TAB4 Option)

The pure shear test is a more direct way of imposing a state of shear deformation and is obtained by a prescribed uniform stretch λ_1 of a specimen, which is long in the 2-direction leaving the 3-direction stress free, or by holding the 2-direction fixed while applying strains in the 1-direction of a thin rectangular sheet, as in the classical experiments of Rivlin and Saunders.¹ If the material is incompressible, this results in $\lambda_3 = 1/\lambda_1$, which is characteristic of a state of shear deformation. It is recommended that pure shear be used along with another test case; otherwise, the coefficients cannot be determined uniquely. X and Y values, respectively, in the TABLES1 entry which contains experimental data must be stretch ratios $\lambda_1 = l/l_0$ and values of the nominal stress F/A_0 = current length, F = current value of the force, l_0 and A_0 = initial length and cross-sectional area, respectively in the 1-direction.

Volumetric Compression (TABD Option)

Pure volumetric compression data may be used to obtain fitted values of the material constants related to volumetric deformation. X values in the TABLES1 entry which contains experimental data must be values of the volume ratio $J = \lambda^3$, where $\lambda = l/l_0$ is the stretch ratio in all three directions; Y values must be values of the pressure, assumed positive in compression.

Guidelines for Using Hyperelastic Materials

- If the strains are small, the results of the hyperelastic elements should compare with those of the linear elastic elements where the shear modulus and the bulk modulus $G \approx 2(A_{10} + A_{01})$ and the bulk modulus $K \approx 2D_1$.

1. Rivlin, R. S. and Saunders, D. W., "Large Elastic Deformations of Isotropic Material VII: Experiments on the Deformation of Rubber," *Phil. Trans. R. Soc. London*, A243, 251-288, 1951.

- Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not available but may be simulated with a large enough value of D_1 . However, a value of D_1 higher than $10^3 \cdot (A_{10} + A_{01})$ is not recommended.
- When determining the material constants from experimental data, it is best to use data from more than one test involving different kinds of deformation. A rank deficiency warning may occur in the least squares fitting algorithm if there is insufficient experimental data. It is best to try to avoid rank deficiency by providing more experimental data or by lowering the order of the strain energy polynomial.

Limitations of Using Hyperelastic Materials

- In general, Poisson's ratio for a hyperelastic material should be no greater than .4995, or, equivalently, that $D_1/(A_{10} + A_{01})$ does not exceed 1000. As incompressibility increases, convergence difficulties and/or ill conditioning of the stiffness matrix occurs.
- You can't define a hyperelastic material for a small strain element. For example, a hyperelastic material can't be referenced by a PSHELL or PSOLID Bulk Data entry.
- Simcenter Nastran doesn't support temperature dependence of the hyperelastic material constants.

Example: Curve-Fitting Algorithm

An example created to solve a fully nonlinear (finite deformation) CQUAD4 element under pure shear with material constants obtained from the simultaneous fitting of experimental data in simple tension, equibiaxial tension, and pure shear. Annotated Bulk Data input is shown in the following listing that illustrates the MATHP and TABLES1 entries, which contain the experimental data for curve fitting.

```

BEGIN BULK
$ PARAMETERS
PARAM DEDROPT 0
PARAM LGDISP 1
$
NLPAEM 10 40 AUTO 1
$
$ DEFINE GEOMETRY
GRID 1 0. 0. 0. 123456
GRID 2 1. 0. 0. 23456
GRID 3 1. 0. 1. 2456
GRID 4 0. 0. 1. 12456
$ DEFINE ELEMENTS
CQUAD4 1 1 1 2 3 4
$ CONSTRAINTS AND LOADING
MPC 10 2 1 1. 3 1 -1.
MPC 10 4 3 1. 3 3 -1.
SPC 10 3 1
SPCD 100 3 1 5.
$ DEFINE PROPERTIES
$ LARGE STRAIN QUAD4 ELEMENT
PLPLANE 1 1 ISOP 2
$ HYPERELASTIC MATERIAL
$ D1 SPECIFIRD
$ ALJ MATERIAL CONSTANTS TO BE DETERMINED FROM EXPERIMENTAL DATA
MATHP 1 1500.
+MA101 +MA101
+MA102 +MA102
+MA103 +MA103
+MA104 +MA104
+MA105 +MA105
+MA106 +MA106
$ TAB1 TAB2 TAB3 TAB4 TABD
+MA106 100 200 400
$
$ TAB1 DATA
$
$ Treloar [1944] data in simple tension. Nominal stresses in kg-cm-2
$
TABLES1 100
+TA101 1. 0. 1.125 1. 1.25 2. 1.5 3. +TA101
+TA102 1.525 4. 1.875 5. 2. 6. 2.25 7. +TA102
+TA103 2.5 8. 3. 9. 3.625 10. 4. 12.5 +TA103
+TA104 4.75 16. 5.25 20. 5.75 23. 6. 27. +TA104
+TA105 6.25 31. 6.5 34. 6.75 38.75 7. 42.5 +TA105
+TA106 RNDT
$
$ TAB2 DATA
$
$ Treloar [1944] data in equibiaxial tension. Nominal stresses in kg-cm-2
$
TABLES1 200
+TA201 1.016 0.83 1.07 1.56 1.15 2.55 1.21 3.25 +TA201
+TA202 1.335 4.31 1.44 5.28 1.66 6.65 1.93 7.88 +TA202
+TA203 2.46 9.74 3. 12.64 3.4 14.61 3.77 17.33 +TA203
+TA204 4.1 20.11 4.32 22.40 4.54 24.41 RNDT
$
$ TAB4 DATA
$
$ Treloar* [1944] data in pure shear. Units of the nominal stress are kg-cm-2
$
TABLES1 400
+TA401 1.01 0.24 1.1 1.12 1.2 1.92 1.31 2.87 +TA401
+TA402 1.49 3.62 1.86 5.61 2.36 7.2 2.97 9.045 +TA402
+TA403 3.45 10.8 3.93 12.45 4.4 14.23 4.714 15.826 +TA403
+TA404 4.96 17.46 RNDT
ENDDATA

```

Listing 3-1.

Note

See Reference [1]. The data was discretized from plots in Reference [2].

[1] Treloar, L. R. G. *The Physics or Rubber Elasticity*, 3rd ed., Oxford: Clarendon Press, 1975.

[2] Ogden, R. W. "Elastic Deformations of Rubberlike Solids: in "Mechanics of Solids," The Rodney Hill 60th Anniversary Volume, H.G. Hopkins and M. J. Sewell, eds., Oxford: Pergamon Press, pp. 499-537.

The following listing illustrates the produced MATHP entry image that is printed in the output just after the ENDDATA entry image.

```

*** SYSTEM INFORMATION MESSAGE 6410 (IFP8).  BEGIN PROCESSING OF MATHP ENTRY ID = 1
DISTORTIONAL PARAMETER FITTING.
SQUARE ROOT OF SUM OF THE ERRORS SQUARED = 0.25629E+01
UPDATED MATHP ENTRY
1      1.54082      0.24046      1500.00000      0.00000      0.00000      0.00000      0.00000
-0.00375  -0.01743  0.00104  0.00000  0.00000  0.00000  0.00000  0.00000
0.00029  0.00033  -0.00004  0.00000  0.00000  0.00000  0.00000  0.00000
0.00000  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
0.00000  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
    
```

Figure 3-2 through Figure 3-4 illustrate the curves obtained from this and similar decks containing a large strain CHEXA in simple and equibiaxial tension with the material constants generated for the first-, second-, and third-order strain energy polynomial functions from simultaneous fitting of experimental data in simple tension, equibiaxial tension, and pure shear.

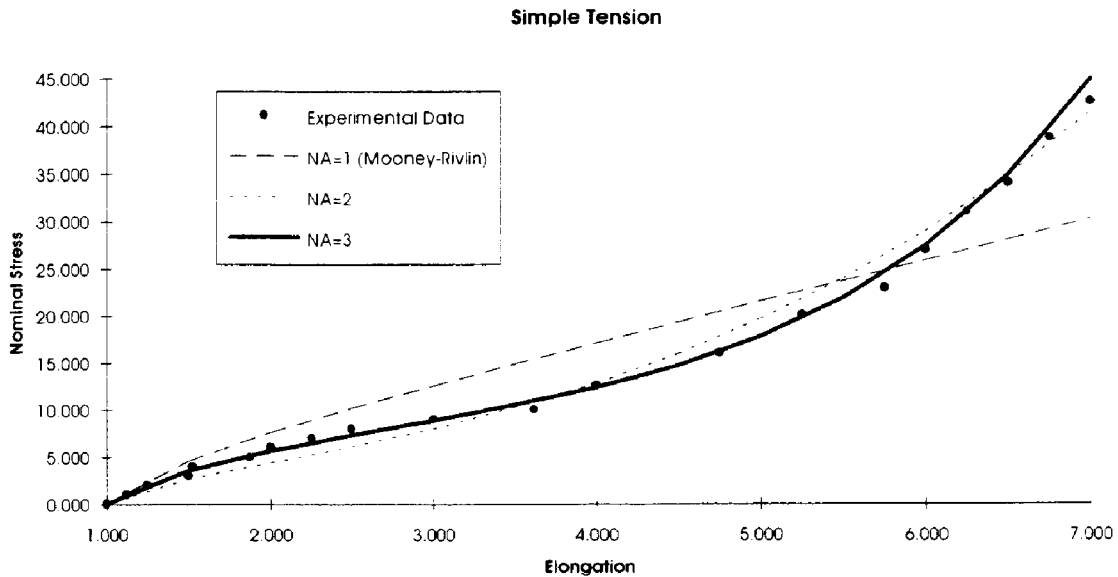


Figure 3-2. Simcenter Nastran Curve-Fitting Algorithm for Hyperelastic Materials (a)

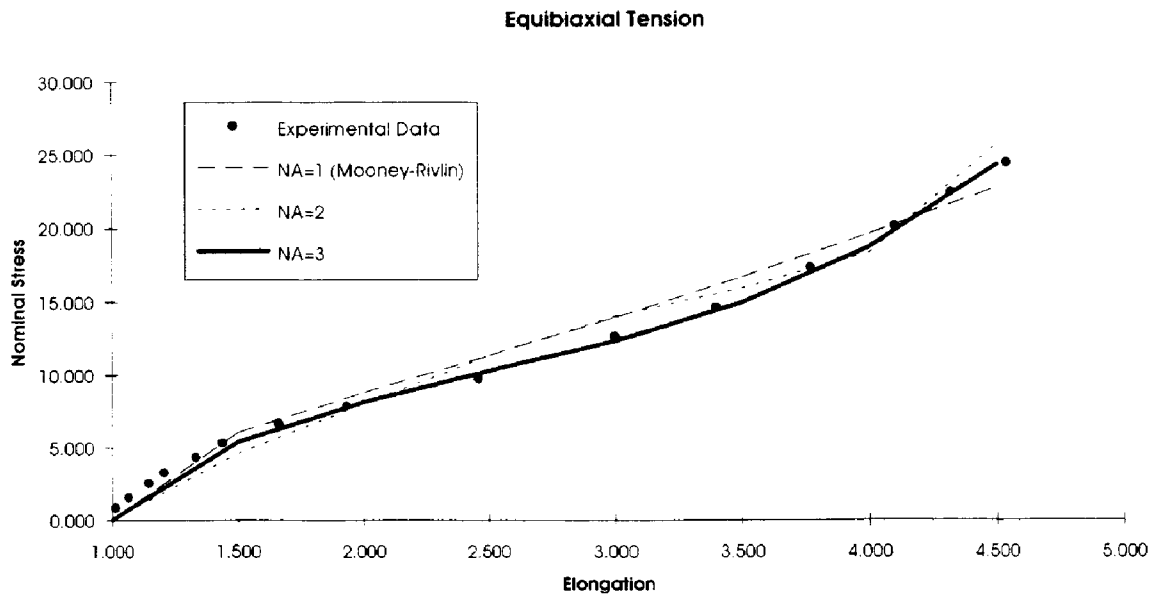


Figure 3-3. Simcenter Nastran Curve-Fitting Algorithm for Hyperelastic Materials (b)

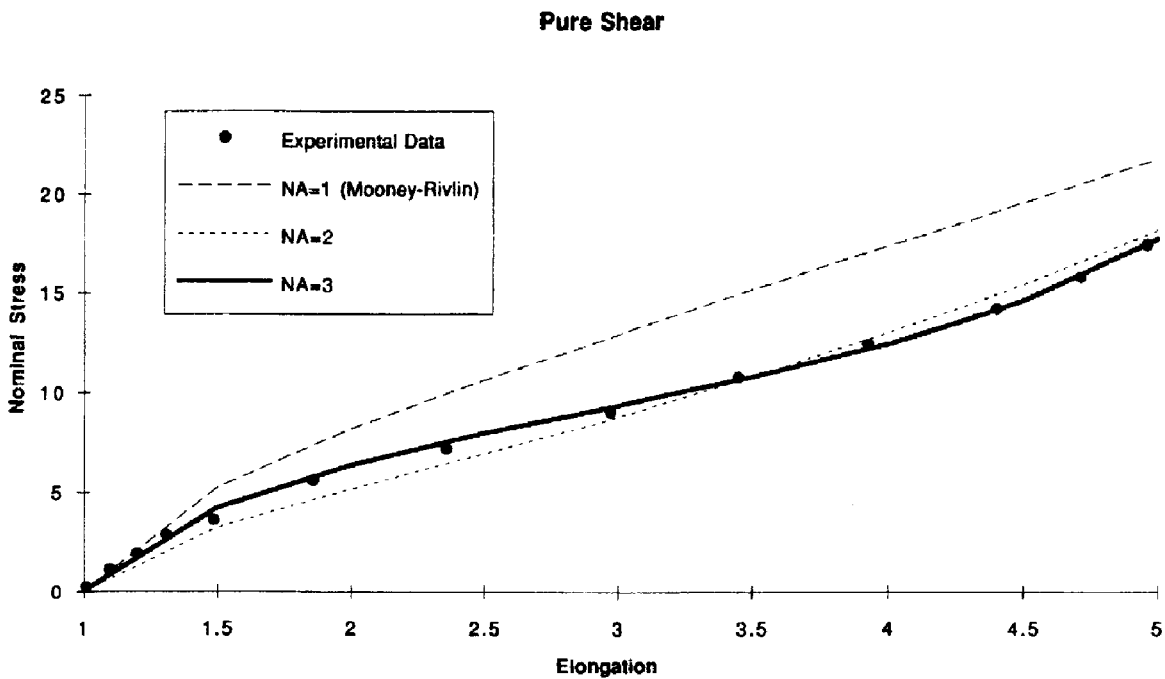


Figure 3-4. Simcenter Nastran Curve-Fitting Algorithm for Hyperelastic Materials (c)

3.4 Elastic-Plastic Material

Before discussing the computational aspects associated with the various models, the definitions of some of the commonly used terms in plasticity are given below.

Yield Stress

Yield Stress is usually measured as the value of stress which produces the smallest measurable permanent strain.

Bauschinger Effect

When a plastically deformed specimen is unloaded, residual stresses on a microscopic scale remain and influence the plastic yielding for the different loading. If the previous strain was a uniform extension and the specimen is then reloaded in compression in the opposite direction, it is observed that yielding occurs at a much reduced stress. This is known as the Bauschinger Effect.

The phenomenon of Bauschinger effect can best be described with reference to **Figure 3-5** which shows an idealized stress-strain curve of a metal first deformed by uniform tension, the load removed and then reloaded in compression. According to one model (which is one extreme viewpoint), it is assumed that the elastic unloading range will be double the initial yield stress. If the initial yield stress in tension is σ_y and the specimen is loaded up to a stress σ_1 and unloaded, the plastic yielding will begin in compression at a stress σ_2 given by

$$\sigma_2 = \sigma_1 - 2\sigma_y$$

Equation 3-5.

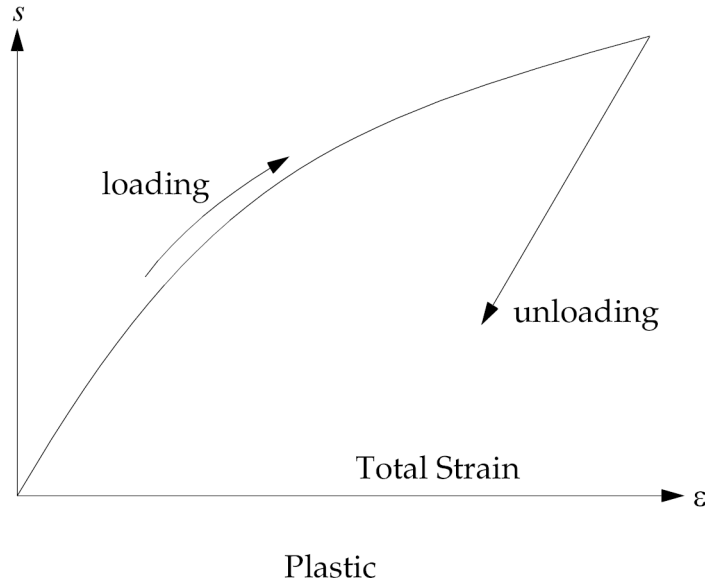


Figure 3-5. Material Nonlinear Models

Yield Function

Simcenter Nastran supports the following types of yield criteria:

- von Mises (ductile materials)
- Tresca (brittle and some ductile materials)

- Drucker-Prager (sand and concrete)
- Mohr-Coulomb (rock material)

Associated with the yield criteria, there exist yield functions in the form of

$$F(\sigma, \bar{\epsilon}^P) = f(\sigma) - Y(\bar{\epsilon}^P) = 0$$

Equation 3-6.

where:

- $f(\sigma)$ = an effective stress, a function of the stress state
- $Y(\bar{\epsilon}^P)$ = a tensile yield stress, a function of the stress hardening.

Geometric representation of the yield function is a surface in the stress space ($\sigma_1 - \sigma_2 - \sigma_3$ coordinates) which is called a yield surface. The von Mises yield surface is a circular cylinder and Tresca's is a hexagonal cylinder. The yield surface is reduced to the yield locus in the case of plane stress, as shown in [Figure 3-6](#) and [Figure 3-7](#) for the von Mises and Tresca criteria, respectively. The Mohr-Coulomb and Drucker-Prager yield functions represent conical surfaces as shown in [Figure 3-8](#) and [Figure 3-9](#).

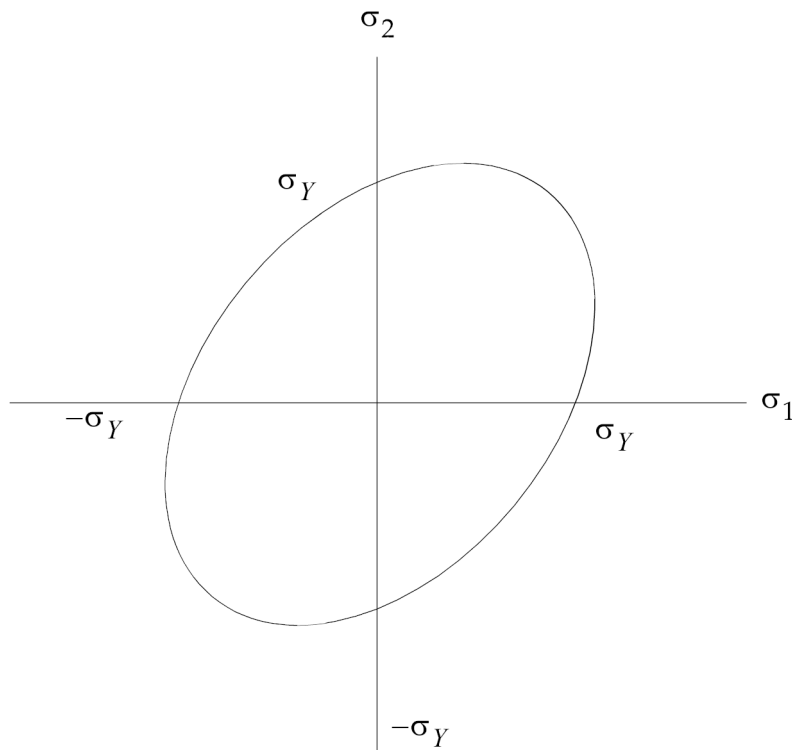


Figure 3-6. von Mises Yield Locus for Plane Stress

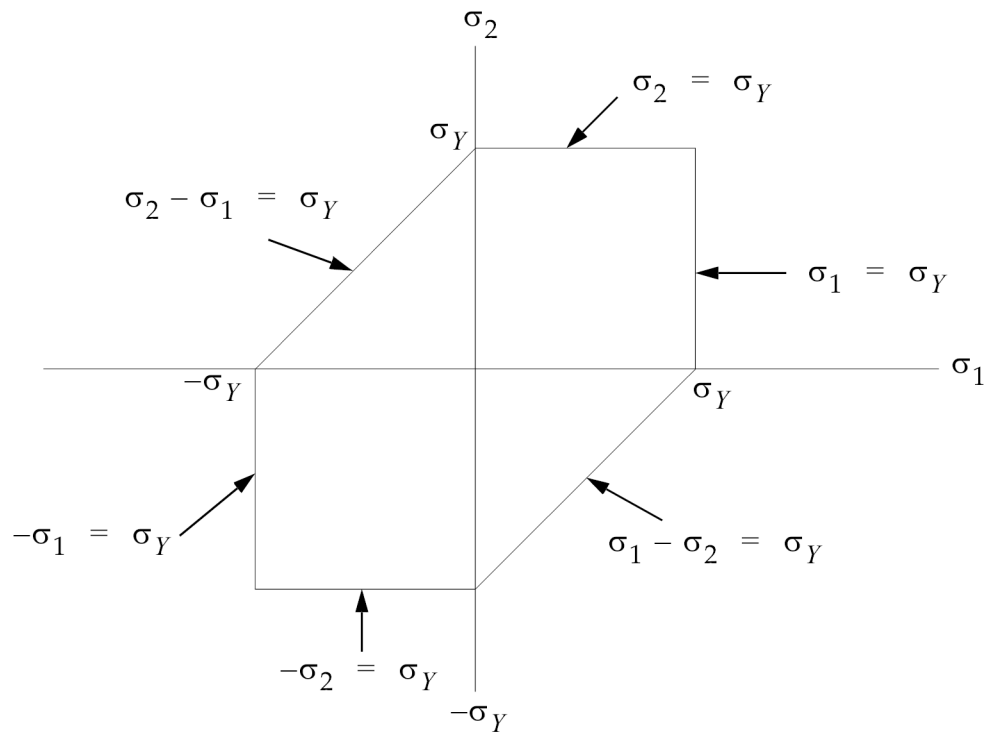


Figure 3-7. Tresca's Yield Locus for Plane Stress

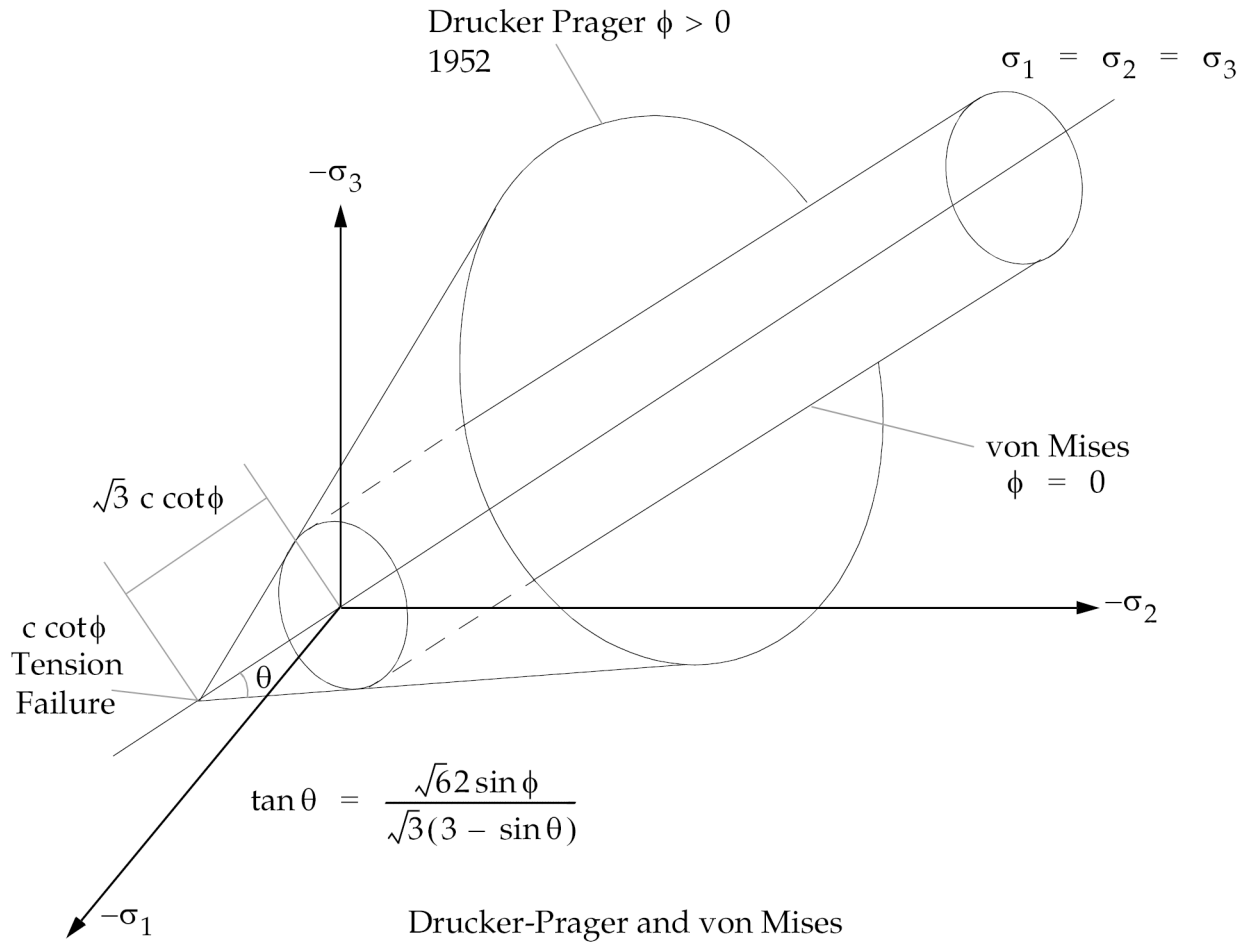
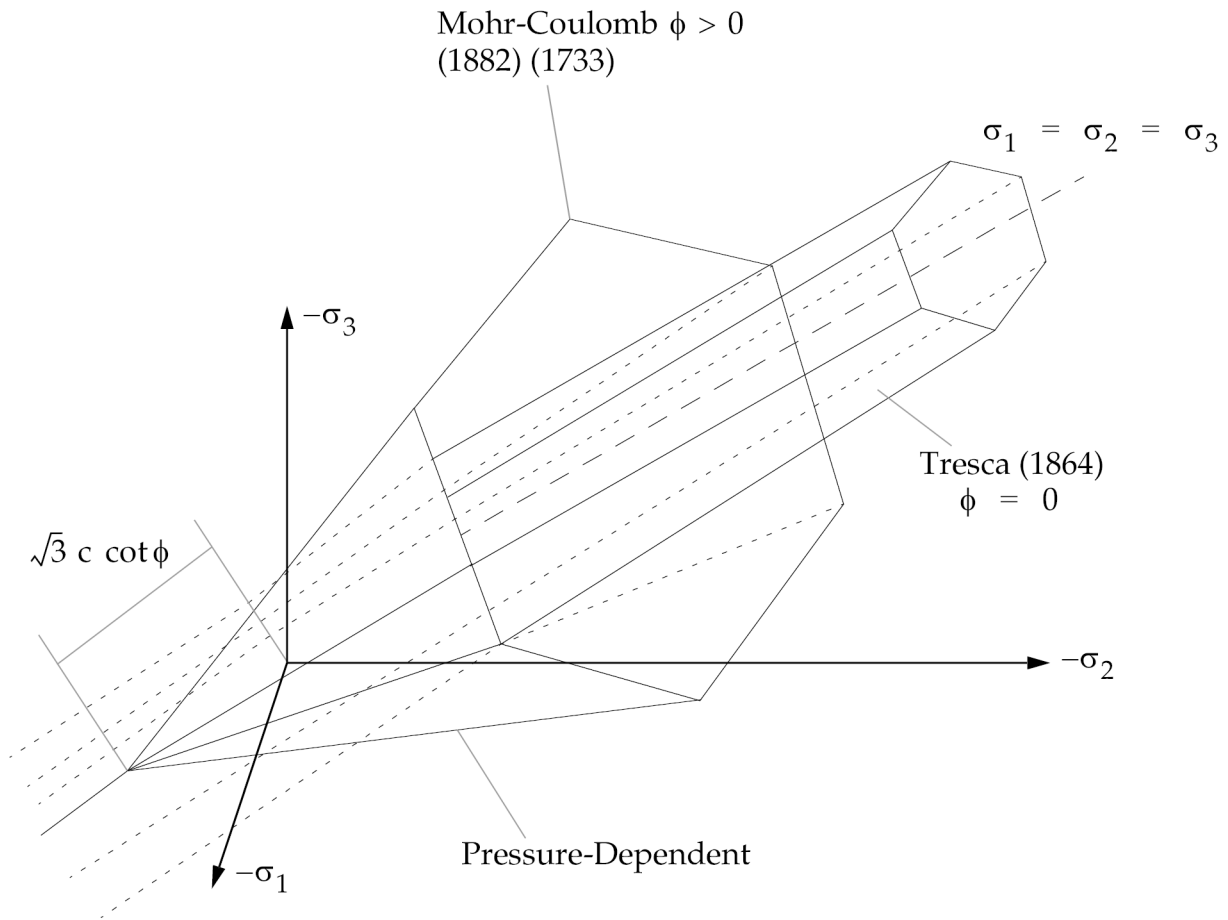


Figure 3-8. Some Isotropic Yield Surfaces in Principal Stress Space



Mohr-Coulomb and Tresca

Figure 3-9. Some Isotropic Yield Surfaces in Principal Stress Space

Defining Elastic-Plastic Materials in Simcenter Nastran

In Simcenter Nastran, you can perform elastic-plastic material analyses using isotropic materials. To define the properties for an elastic-plastic material, use the MAT1 bulk data entry in conjunction with the MATS1 and TABLES1 entries to define the stress-dependent material properties.

On the MATS1 entry, when you're creating an elastic-plastic material, use field 4 to set the TYPE option to PLASTIC. You can then specify additional options for the work hardening slope, yield function, and hardening rule.

When TYPE = "PLASTIC", the software interprets the stress-strain table as engineering stress and strain.

See Also

- "MAT1" in the *Simcenter Nastran Quick Reference Guide*
- "MATS1" in the *Simcenter Nastran Quick Reference Guide*

- “TABLES1” in the *Simcenter Nastran Quick Reference Guide*

Grid Point Stresses in Solid Elements

When material plasticity is present in a SOL 106 or SOL 129 run, the stress and strain at the grid points of CHEXA, CPENTA, CPYRAM, and CTETRA elements is computed as follows:

1. The strain field is evaluated at specific integration points to obtain the total strain at those points.
2. The total strain at the integration points is extrapolated to the grid points.
3. The elastic strain, plastic strain, and stress are computed at the grids from extrapolated total strain.

This method assures that the values for elastic strain, plastic strain, and stress that are recovered at the grid points lie on the stress-strain curve of the material.

3.5 Viscoelastic Material in Nonlinear Analysis

A viscoelastic analysis lets you analyze situations in which the strain rate has a significant effect on the material response. You can analyze the quasi-static behavior of a viscoelastic material using the creep capability of Simcenter Nastran. In Simcenter Nastran, you can perform a creep analysis on linear elastic isotropic and elastoplastic isotropic materials only. You can't perform a creep analysis on anisotropic, nonlinear elastic, and hyperelastic materials.

In Simcenter Nastran, you can model primary and secondary creep, but not tertiary creep. Simcenter Nastran uses the Kelvin model for primary creep and the Maxwell model for secondary creep.

Note: Before you can perform a creep analysis, you must first perform a static analysis.

Defining Viscoelastic Materials

You can define elastoplastic properties to consider plastic deformation using the MATS1 bulk data entry in conjunction with the MAT1 bulk data entry.

You can define viscoelastic properties (linear or nonlinear) in terms of rheological model parameters (K_p , C_p , and C_s) of the Maxwell-Kelvin model in the CREEP and TABLES1 bulk data entries. Then, you can perform a quasi-static analysis of the viscoelastic materials by specifying time increment with the NLPARM bulk data entry (selected by the subcase) in a SOL 106 analysis.

Two classes of empirical creep law are available. **Creep Law Class 1** is expressed as:

$$\varepsilon^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t$$

Equation 3-7.

Parameters $A(\sigma)$, $R(\sigma)$, and $K(\sigma)$ are specified in the following form, as recommended by the Oak Ridge National Laboratory:

Parameter	Function 1	Digit	Function 2	Digit
$A(\sigma)$	$a\sigma^b$	i=1	$ae^{b\sigma}$	i=2

Parameter	Function 1	Digit	Function 2	Digit
R(σ)	$ce^{d\sigma}$	j=1	$c\sigma^d$	j=2
K(σ)	$e \cdot [\sinh(f\sigma)]^g$	k=1	$ee^{f\sigma}$	k=2

TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines $A(\sigma) = a\sigma^b$, $R(\sigma) = c\sigma^d$, and $K(\sigma) = ee^{f\sigma}$.

Creep Law Class 2 (TYPE=300) is expressed as:

$$\varepsilon^c(\sigma, t) = a\sigma^b t^d$$

Equation 3-8.

where the values of b and d must be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

The constants for the creep laws are obtained by curve fitting your experimental data. Simcenter Nastran does not provide any methods or tools for curve fitting your creep data. You should seek other resources for this process. You may need to fit all of the creep laws to determine which is best suited for your data. To determine if a particular creep law is a good fit, you will need to graphically plot the curve-fitting result and compare it with your experimental data.

See “Creep and Viscoelasticity” in the *Handbook for Nonlinear Analysis*.

See Also

- “MAT1” in the *Simcenter Nastran Quick Reference Guide*
- “MATS1” in the *Simcenter Nastran Quick Reference Guide*
- “TABLES1” in the *Simcenter Nastran Quick Reference Guide*
- “CREEP” in the *Simcenter Nastran Quick Reference Guide*

3.6 Temperature-Dependent Material

In Simcenter Nastran, only linear and nonlinear elastic materials can be temperature dependent. Elastic-plastic and hyperelastic materials can't be temperature dependent.

You can specify temperature-dependent properties for elastic and nonlinear elastic materials by using the MATTi, TABLEST, and TABLES1 bulk data entries.

Note: A limited temperature dependence for creep materials is available based on the Arrhenius equation.

See Also

- “TABLES1” in the *Simcenter Nastran Quick Reference Guide*
- “TABLEST” in the *Simcenter Nastran Quick Reference Guide*

3.7 Understanding Geometric Nonlinearity

Geometric nonlinearities are manifested in problems involving large rotations and large deformation. The characteristics of a geometrically nonlinear problem are:

- follower forces due to large rotations
- geometric stiffening due to initial stress effect (as a result of large rotations)
- large strains due to large deformation

Geometric nonlinear effects should be significant if the deformed shape of the structure appears distinctive from the original geometry when you inspect it visually.

A more rigorous and quantitative definition for the large displacements can be derived from the plate theory of Kirchhoff and Love, which states that the small deflection theory is valid for a maximum deflection of less than 20% of the plate thickness or 2% of the small span length. However, this definition seems to be a little conservative for numerical analysis. Additionally, there is no distinct limit for large displacements because geometric nonlinear effects are related to the boundary conditions as well as the dimensions of the structure. If the load-deflection curve of the critical point can be estimated, the loading point should be in the nonlinear portion of the curve.

Geometric nonlinear effects in the structure involving large rotations, whether rigid body rotations or deformation induced rotations, are self-evident. Stiffening of a membrane, stiffness in a pendulum or snap-through of an arch belong to this category.

The motion of a pendulum under gravity is caused by geometric (differential) stiffness. Follower forces are manifested when the applied loads are displacement dependent, such as pressure load and thermal load applied on the surface that rotates. Centrifugal force is another example of follower forces.

Large strain effects are pronounced in metal forming, rubber and elastomer applications. In such applications, the strains exceed 100%. Finite strain formulation is required to treat the problems in this category.

Simcenter Nastran supports the large strain capability for rubbers and elastomers. This formulation contains the effect of large strain as well as geometric nonlinearity. However, Simcenter Nastran doesn't support large strain formulation for metal forming applications. In most structural applications, however, moderately large strains (20 to 30%) appear in local areas if there is any large deformation

Other geometric nonlinear effects are treated by updated element coordinates, gimbal angles (or rotation vector), and the differential stiffness $[K^q]$.

Chapter 4: Loads for Nonlinear Analysis

4.1 Overview of Nonlinear Static Loads

The static loads in nonlinear analysis consist of a subset of the total Simcenter Nastran static load set. Most of the relevant loads data applicable to the linear static analysis are also applicable to nonlinear static analysis, except for the Bulk Data entry DEFORM. The enforced displacements can be applied as loads using the Bulk Data entry SPCD, which defines an enforced motion of a grid in a specific fixed direction. Each grid point with an enforced displacement must also appear on an SPC or SPC1 entry.

The loads are selected in each subcase by the LOAD Case Control command. The load for a subcase is subdivided into the number of increments specified for the subcase. The solution strategy in nonlinear is to apply the loads in an incremental fashion until the desired load level is reached. The algorithms “remember” the loads from one subcase to the next. If the load reaches the desired level in a subcase and if the load description is left out in the subsequent subcase inadvertently, the nonlinear solution algorithm will begin in an incremental fashion to remove the load. The loads described below make up the valid subset applicable for nonlinear static analysis.

Concentrated Loads

The concentrated loads consist of two categories: those which are stationary in direction throughout the analysis and those which follow the grid motion. The first category consists of the following:

FORCE

MOMENT

SLOAD

The second category consists of the following:

FORCE1

FORCE2

MOMENT1

MOMENT2

The three FORCE entries differ only in the manner in which the direction of the force is specified. FORCE uses the components of a vector. FORCE1 uses two grid points, not necessarily the same as the loaded grid points. FORCE2 defines the direction of the force as a vector which is the vector product of two other vectors. The distinctions between the three MOMENT entries are similar. The SLOAD entry is used to apply loads to scalar points.

Distributed Loads

The distributed loads consist of two categories:

- those which are stationary in direction throughout the analysis (PLOAD1)

- those which follow the element motion (PLOAD, PLOAD2, PLOAD4)

Loads may be applied to the interiors of finite elements. The PLOAD1 entry defines concentrated and linearly distributed forces and moments to the CBEAM element. However, the stress calculation for nonlinear CBEAM element ignores the distributed loads, but lump them onto the nodal points. The PLOAD entry is exceptional in that it references three or four arbitrarily located grids rather than an element. The PLOAD2 entry defines a uniform static pressure load applied to the CQUAD4 or CTRIA3 elements. The PLOAD4 defines a pressure load on surfaces of CHEXA, CPENTA, CTETRA, CTRIA3, and CQUAD4 elements which need not be normal to the surface. Since the surface hyperelastic elements CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6 are plane strain elements, no pressure load may be applied on these elements.

Mass Related Loads (GRAV and RFORCE Entries)

The mass related loads are gravity and centrifugal loads, which require specification of mass data. The gravity loads which can be specified on the GRAV Bulk Data entry are stationary in direction throughout the analysis. The GRAV entry is used to define the direction and magnitude of the gravity vector in some identified coordinate system. The components of the gravity vector are multiplied by the mass matrix to obtain the components of the gravity force at each grid point.

Centrifugal loads are specified on the RFORCE Bulk Data entry. RFORCE defines the components of a spin vector which is used internally to compute centrifugal forces. Each component of the spin vector is multiplied by a scale factor.

Thermal Loads

Thermal loads are selected in each subcase by the Case Control command TEMP(LOAD). Thermal loading which is a follower loading is accomplished through the following entries:

TEMP

TEMPO

TEMPP1

TEMPP3

TEMPRB

The TEMP and TEMPO provide for grid point temperature specification which are then interpolated to points within elements. For the nonlinear CQUAD4 and CTRIA3 elements the TEMPP1 provides for an average temperature and thermal gradient in the thickness direction and the TEMPP3 provides for temperature at points through the thickness. TEMPP1 and TEMPP3 are not available for the plane strain hyperelastic elements CQUAD4, CQUAD8, CQUAD, CTRIA3, and CTRIA6. The TEMPRB provides for average temperature at ends A and B for the CROD and CBEAM element. Average temperature specified directly for an element will take precedence over the temperature interpolated from the elements connected grid points.

The Case Control command TEMP(INIT) is required above subcase level to specify an initial temperature distribution. The use of TEMP(MAT) or TEMP(BOTH) will cause a fatal error. Subcases that do not contain a TEMP(LOAD) will default to TEMP(INIT), which implies no thermal load. Thermal loads are often used to provide initial strains.

Load Combinations

Various load specifications can be combined using the LOAD, CLOAD, and LSEQ entries.

The LOAD provides for the linear combination of static load sets. The static loads (LOAD, FORCE, etc.) applied to the upstream superelements cannot be referenced by the LOAD Case Control command. The CLOAD entry is designed to apply static loads to upstream superelements by combining loads defined in LSEQ entries. The (CLOAD, LSEQ) combination allows the nonlinear algorithms to apply in an incremental fashion on the upstream superelement loads to the boundary of the residual structure.

The CLOAD defines a static load combination for superelement loads acting on residual boundary and the LSEQ defines a static load combination for superelement upstream loads. The LID and TID field entries point to the appropriate load and temperature distribution entries. The CLOAD defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ. The IDVi fields correspond directly to the numeric labels defined via the EXCITEID fields of the LSEQ entries.

The CLOAD must be selected in the residual solution subcase via the Case Control command, CLOAD. The LSEQ is selected by a LOADSET Case Control command above any residual subcase. Only one LOADSET may appear in Case Control. The hierarchy of the loads data is shown schematically in [Figure 4-1](#). An example for the Case Control set-up follows:

```

SEALL=ALL
LOADSET = 1000
$ Selects LSEQ 1000 for upstream loads
SUPER = ALL $ Identify SEs to process
.
.
.
. DISP = ALL
.
.
.
.
.
$ NONLINEAR SOLUTIONS FOR RESIDUAL SUPERELEMENTS
SUBCASE 10
    CLOAD = 1001 $ Refers to CLOAD Bulk Data
    NLPARM = 12 $ Iteration control
.
.
.
SUBCASE 20
    CLOAD = 1002
    LOAD = 10 $ Residual SE load
    NLPARM = 22

```

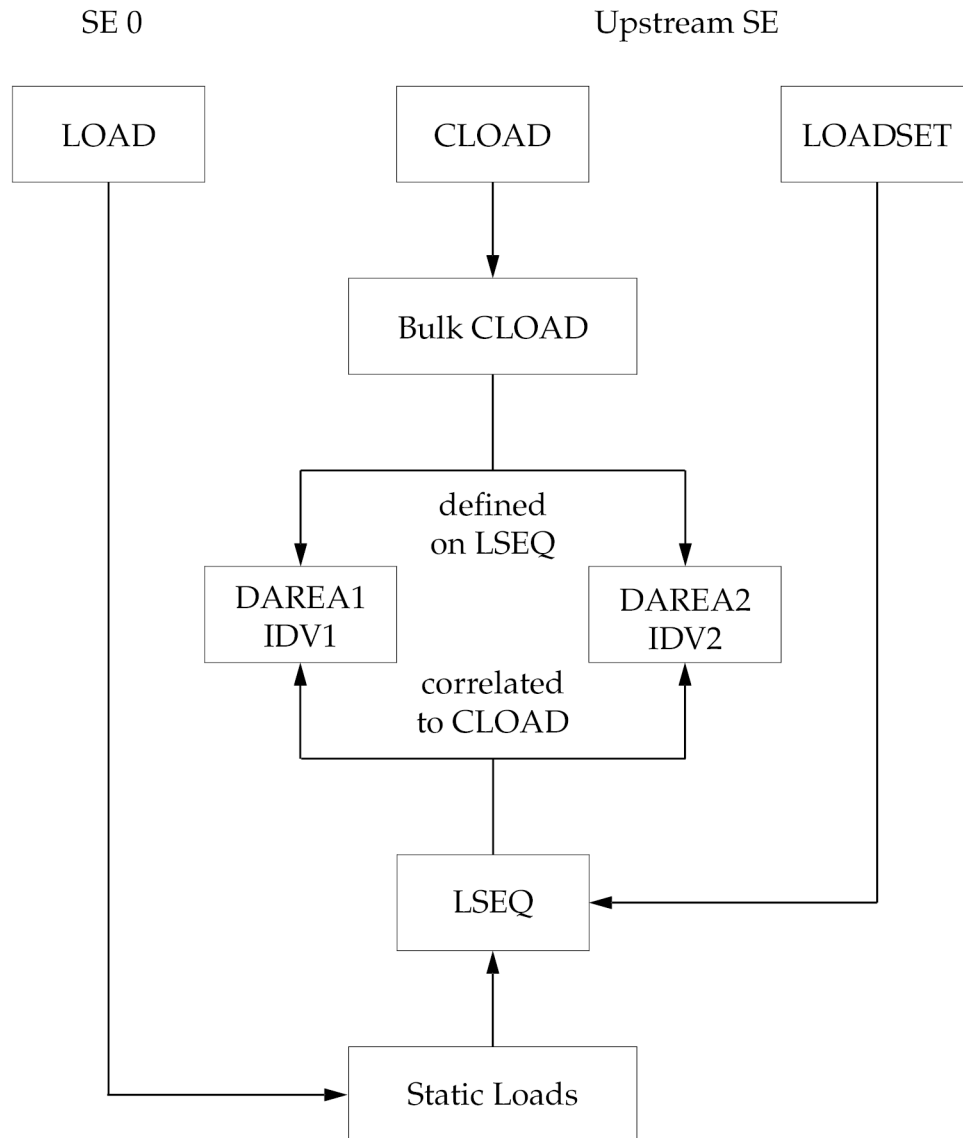


Figure 4-1. Hierarchy of Load Data

4.2 Overview of Nonlinear Transient Loads

The methods employed to define loads in nonlinear transient analysis are similar to those used in the linear solutions. A single degree-of-freedom or a set of grid points may be loaded with force pattern that varies with time. Functions may be tabular such as an earthquake or a booster liftoff, or they may be simple analytic functions such as a sine wave. Simple static load sets generated in both upstream superelements and the default residual may be used to create the dynamic loads. They may be scaled and combined with other loads to simulate complex loading problems.

For special problems involving simple scalar nonlinearities, the traditional nonlinear loads (NOLINI option) may be used in both types of transient solutions. These are useful for simulating nonlinear damping mechanisms and rotational coupling not provided by the CGAP elements.

Transient loads define the loadings as functions of time and the location. They can be a load applied at a particular degree-of-freedom, pressure over the surface area, or the body force simulating an acceleration. The time history is provided by the TLOADi Bulk Data entries and the static loads may be converted to dynamic loads using LSEQ entry. The transient dynamic loads are selected in the Case Control Section by the following commands:

DLOAD	Selects TLOADi or DLOAD Bulk Data entries. Each stage of the transient solution may be defined with a separate SUBCASE, each with its own DLOAD request. The DLOAD entry is restricted to act only in the time defined in its subcase.
LOADSET	Selects LSEQ entries. Must be placed above the solution subcases and/or in the Superelement Subcases.
NOLINEAR	Selects NOLINi entries. May not be changed between solution subcases.
NLLOAD	Required for NOLINi output request.

Bulk Data entries associated with the transient loads in SOL 129 are briefly described below:

TLOAD1, TLOAD2	Required data defining the coupling between load factors and functions of time (TABLEDi, DELAY, or coefficients). Specifies the time functions, $F_i(t)$, for dynamic load i . TLOAD1 defines the load history in TABLEDi format while TLOAD2 defines the load history by an analytic expression. Note that each TLOADi entry must have a unique set identification number.
DAREA	Defines load scale factors, A_{ij} , for each degree-of-freedom, u_j requested in set i . The DAREA factors may also be derived from static load entries with the LSEQ Bulk Data entry.
LSEQ	Generates transient load history for static loads and controls the assembly of static load vectors to be used in dynamic analysis. Each set is equivalent to a subcase in Case Control. It will cause a static load vector A_j to be generated and will label it with the EXCITEID identification. This is useful for modeling problems with distributed transient loading such as GRAV loads or pressurized areas.
DLOAD	Combines different TLOADi functions into a single set and scales each by a factor, C_{ki} as shown in Eq. 4-1. Performs the same function as the static LOAD Bulk Data input. Unlike statics, this is the only method to combine loads in dynamic analysis.

$$P_{kj} = C_{ki} \cdot A_j \cdot F_i(t)$$

Equation 4-1.

For more information, see [NLRGAP](#).

TABLEDi, i=1,2,3,4	Used with TLOAD1 entries to specify general time varying tabular functions. Since extrapolation is used beyond the range of data, you should be careful with the end points.
-----------------------	--

DELAY	Specifies the delay time for applying the forcing function defined in TLOADi to each GRID point. The DELAY entry is associated with the degree-of-freedom and not the forcing function in each TLOADi. This option is useful for defining loads that travel across a structure such as an oblique wave along a dam or a bump in the road exciting the wheels of a traveling vehicle. This saves the analyst the effort of defining multiple TLOADi entries for simple time lags.
NOLINi, i=1,2,3,4	<p>Define specific nonlinear scalar loads as functions of velocity and/or displacement. Basic analog nonlinear transfer functions are defined. However, these functions do not generate tangent matrix terms and the Newton iteration method may experience difficulties in convergence. Line search and BFGS solution methods are recommended for these models.</p> <p>NOLIN1 Nonlinear transient load as a tabular function.</p> <p>NOLIN2 Nonlinear transient load as products of two variables.</p> <p>NOLIN3 Nonlinear transient load as a positive variable raised to a power.</p> <p>NOLIN4 Nonlinear transient load as a negative variable raised to a power.</p>
NLRGAP	The NLRGAP is an idealization of the contact between any circular shaft enclosed by a circular housing (Figure 4-2). In this idealization, grid point A represents the outer surface of the shaft while grid point B represents the inner surface of the housing. Grid points A and B are coincident and must have a parallel displacement coordinate system. A rudimentary friction capability is included to account for the friction-induced torque and friction-induced lateral loads that occur with a spinning shaft. However,

friction can be neglected, in which case the solution is consistent with that for a nonrotating shaft in a frictionless housing.

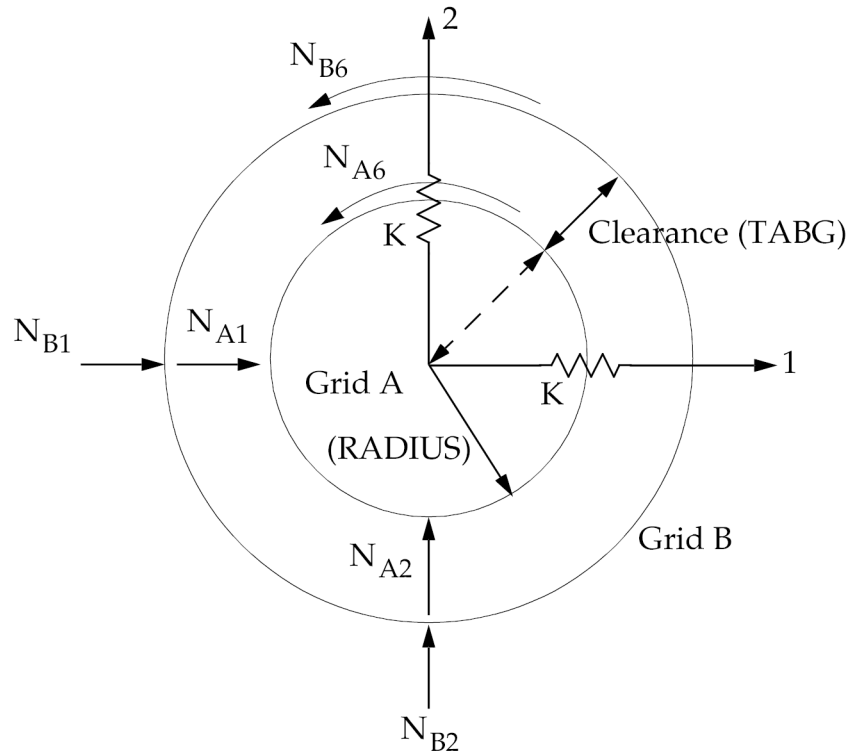


Figure 4-2. Radial Gap Orientations and Nonlinear Load Sign Conventions

NLRGAP

The equations underlying the NLRGAP measure the relative radial displacement between the shaft grid and the housing grid in the displacement coordinate system of grid points A and B. When the relative radial displacement is greater than the clearance (i.e. when contact has occurred), contact forces as well as frictional forces are automatically applied to the grids. The contact and frictional forces are dependent on the contact stiffness and the coefficient of friction.

Definitions

$K(t,u)$	Contact stiffness, function of time or displacement
$G(t)$	Radial clearance, function of time
$\mu(t)$	Coefficient of friction, function of time. Positive value implies shaft rotation from axis 1 towards axis 2.
R	Shaft radius for friction induced torque calculation

Equations

The NLRGAP is internally composed of the following equations. It is helpful to visualize Grid A as belonging to the shaft and Grid B as belonging to the housing. Consider each grid to have DOF 1 and 2 in the plane of action (e.g., DOF 1 = X and DOF 2 = Y for XY orientation).

The element relative displacement in directions 1 and 2 are:

$$\Delta U_1 = U_{A1} - U_{B1}$$

Equation 4-2.

$$\Delta U_2 = U_{A2} - U_{B2}$$

Equation 4-3.

The relative radial displacement is:

$$\Delta r = \sqrt{\Delta U_1^2 + \Delta U_2^2}$$

Equation 4-4.

The force in the gap when it is open ($\Delta r \leq G(t)$) is zero. When the gap is closed ($\Delta r > G(t)$), the penetration is defined as:

$$P = \Delta r - G(t)$$

Equation 4-5.

The nonlinear gap forces are:

$$S = K(t,u) \left(1 - \frac{G(t)}{\Delta r} \right)$$

Equation 4-6.

$$N_{A1} = -\Delta U_1 S + \Delta U_2 S \mu(t)$$

Equation 4-7.

$$N_{B1} = \Delta U_1 S - \Delta U_2 S \mu(t)$$

Equation 4-8.

$$N_{A2} = -\Delta U_2 S - \Delta U_1 S \mu(t)$$

Equation 4-9.

$$N_{B2} = \Delta U_2 S + \Delta U_1 S \mu(t)$$

Equation 4-10.

If the shaft radius R is input, then the following friction induced torque loads are generated.

$$N_{A6} = -R \Delta r S \mu(t)$$

Equation 4-11.

$$N_{B6} = R \Delta r S \mu(t)$$

Equation 4-12.

Benefit

NLRGAP lets you model contact between two coaxial cylinders, for example shaft and housing.

Input

You use the NLRGAP bulk data entry to model the radial gap. The contact stiffness can be input either as a function of time or as a function of penetration. In both cases, the stiffness curves are input on TABLEDi Bulk Data entries which are referenced by the NLRGAP entry. Inputting stiffness as a function of time lets you model special situations where structural changes over time cause the contact stiffness to vary over time. Inputting stiffness as a function of penetration lets you input a nonlinear stiffness curve which might be required, for example, if the shaft or housing were covered by some nonlinear-elastic material.

As with stiffness, both the coefficient of friction and the clearance can be functions of time. The time-varying coefficient of friction might be used where the quality of lubrication varied with time, or if experimental data showed that the surface characteristics of the shaft and housing varied with time. The clearance can be input as a function of time to model situations where an accelerating, spinning rotor expands due to centripetal loads.

The NLRGAP entry can also approximate the torque on the shaft and housing due to friction. If this effect is desired, the RADIUS field of the NLRGAP must be specified. If not specified, then friction effects will still be included in the lateral loads as shown in [Eq. 4-7](#) through [Eq. 4-10](#). The torque loads, [Eq. 4-11](#) and [Eq. 4-12](#), will not be computed nor applied if RADIUS is input as 0.0 or left blank.

As with the NOLINs, the NLRGAP is selected by the NONLINEAR Case Control command.

Output

The output is the nonlinear loads applied to the NLRGAP grids to simulate the contact. The loads can be recovered with the NLLOAD Case Control command.

Guidelines

- There should be mass on both grids of the NLRGAP. Otherwise erratic results and divergence can occur. A small amount of damping (e.g. CDAMPi) between the two grids may improve stability for some problems.
- The two grids listed on the NLRGAP should be coincident and have parallel displacement coordinate systems.
- The friction capability is somewhat arbitrary in sign. Referring to the figure [Radial Gap Orientations and Nonlinear Load Sign Conventions](#), a positive coefficient of friction is consistent with shaft rotation from axis 1 towards axis 2 (counter-clockwise). If the shaft is rotating in the opposite direction, then a negative coefficient of friction must be input in order for the friction induced loads to have the proper signs.
- In SOL 129, setting ADJUST = 0 on the TSTEPNL entry will turn off the adaptive time step. This is useful if large contact stiffness is causing an excessively small time step.
- As with the NOLINs, the forces applied to the NLRGAP grids to simulate the radial contact are based upon the relative displacements of the grids from the previous time step. Therefore, the larger the contact stiffness, the smaller the necessary time steps. The smaller the contact stiffness, the larger the penetration. Use the smallest contact stiffness possible to model the contact to the accuracy desired. The time step may need to be decreased by 10 times or more compared to the same model without NLRGAPs and NOLINs.
- The NLRGAP is based upon small displacement theory. Therefore, for most accurate results, especially with respect to friction induced torque, the clearance should be small relative to the housing diameter. The friction induced torque applied to grids A and B will be equal and opposite in direction.

Limitations

- The NLRGAP can only be positioned in the XY, YZ or ZX plane of the displacement coordinates of the referenced grids.
- The DOF in the plane of the NLRGAP must be in the d-set. This means that these DOF cannot be constrained by SPCs or be dependent DOF on MPCs and RBE type elements.
- The NLRGAP can only be used in SOLution sequences 109 and 129.

A diagram of input data linkages is shown in [Figure 4-3](#). The general form of the transient load, P_{kj} , for each degree-of-freedom, u_j , is:

$$P_{kj} = C_{ki} \cdot A_j \cdot F_i(t)$$

Equation 4-13.

where the factors are defined by various Bulk Data entries as shown in [Figure 4-4](#).

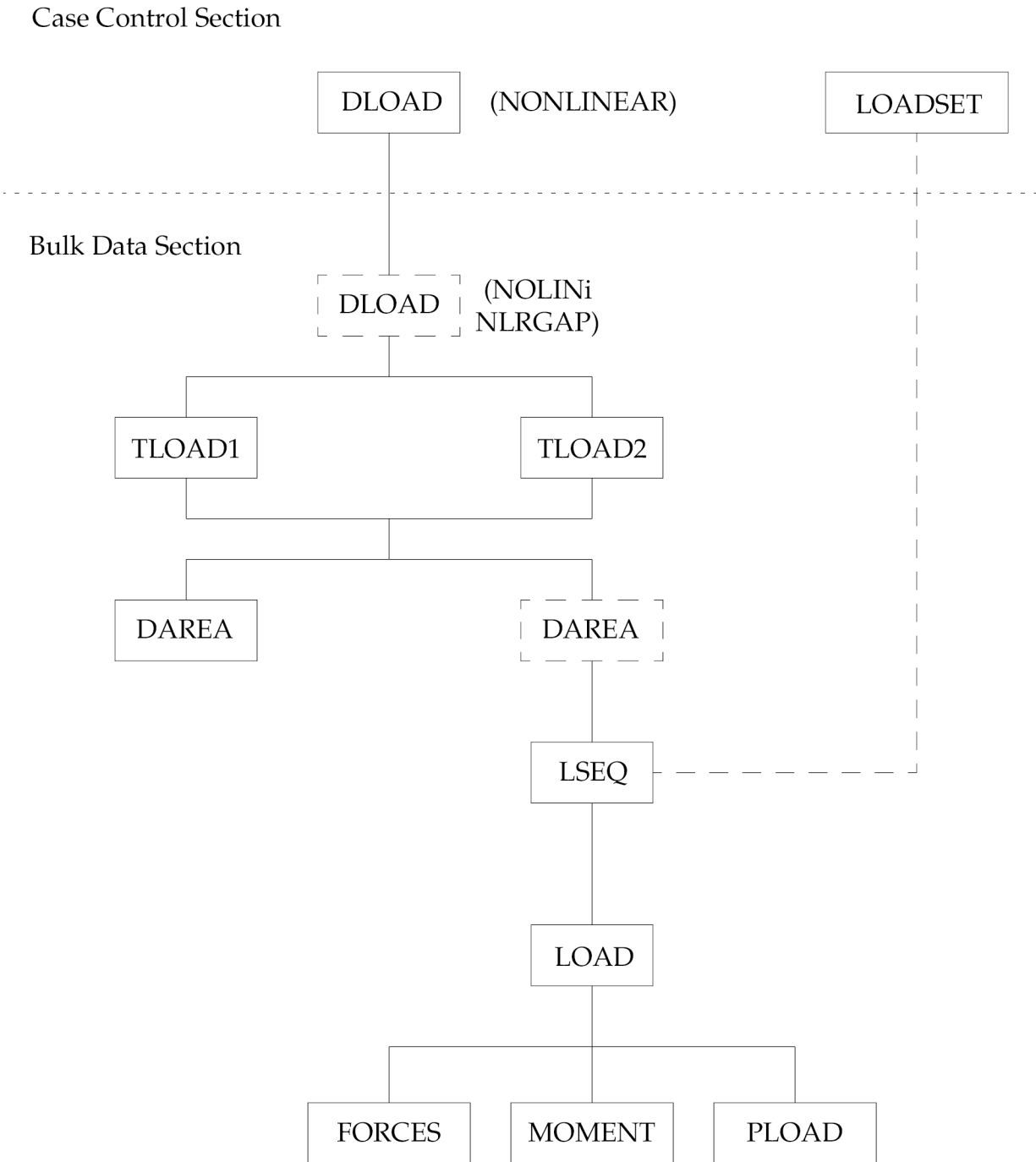
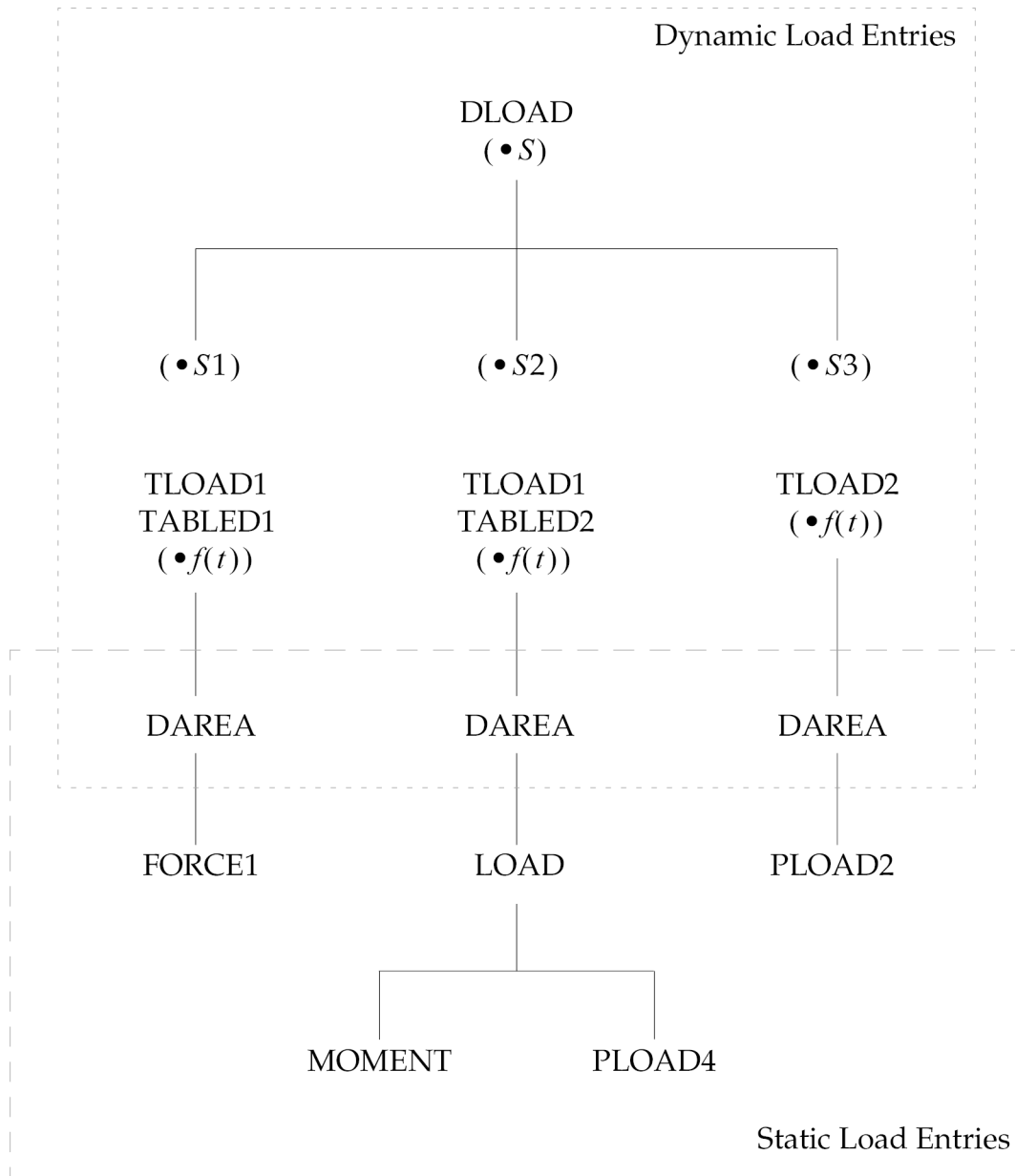


Figure 4-3. Dynamic Loads for Transient Analysis



Note that each load is scaled by $S \cdot S_i \cdot f(t)$ at each time step in dynamic loads.

Figure 4-4. Example of Combining Loads

Chapter 5: Nonlinear Analysis Types

5.1 Overview of Nonlinear Analysis Types

Simcenter Nastran provides the following types of basic nonlinear analysis capabilities:

- 3-D slide line contact
- nonlinear static
- nonlinear buckling
- nonlinear modal
- nonlinear transient response
- nonlinear analysis with superelements

5.2 3-D Slide Line Contact Analysis

Slide line contact lets you model interactions between two deformable bodies. One of the deformable bodies is called the master and the other is called the slave. The modeling of interaction requires that you define contact regions in terms of slide lines. You may specify as many slide lines as you need.

A slide line contact region consists of a master line and a slave line as shown in [Figure 5-1](#). A master line is a list of grid points in the topological order on the master body. A slave line is a list of grid points in the topological order on the slave body. The grid points on the master line are called the master nodes, and on the slave line are called the slave nodes. A line segment joining two consecutive master nodes is called a master segment. Thus a master line, in general, consists of number of master segments. A minimum of one master segment consisting of two master nodes are required for the master line. Similar to master line, a slave line, in general, consist of a number of slave segments. However, a slave line may not have any slave segments; it may have just one slave node.

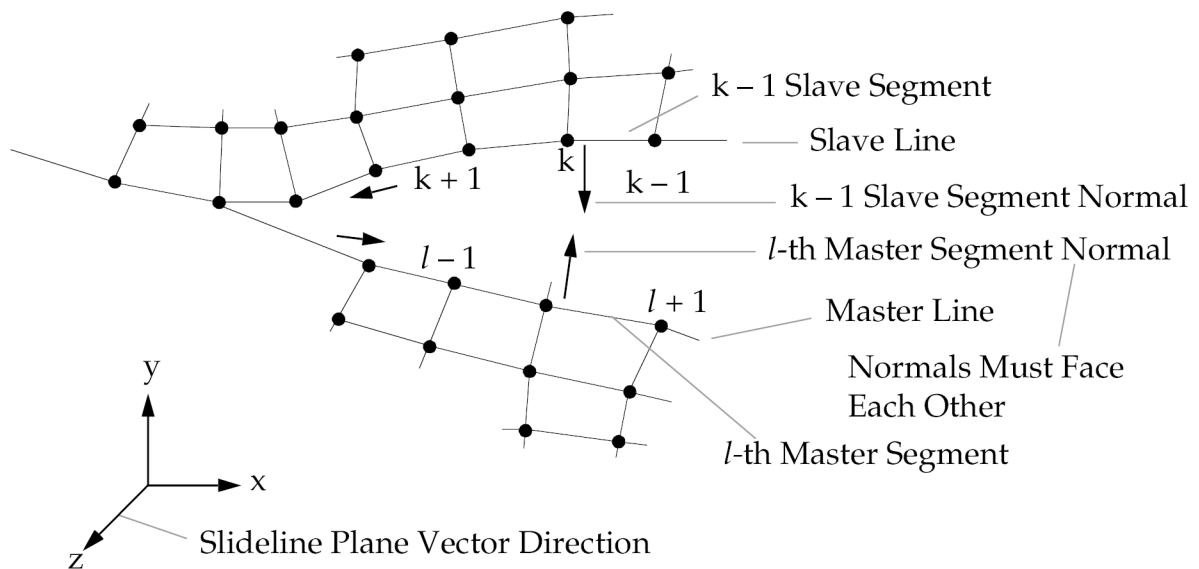


Figure 5-1. Typical Slideline Contact Region

Note

- Arrows show positive direction from ordering nodes, counterclockwise from master line to slave line.
- X-Y plane is the slideline plane. unit normal in the Z-direction is the slideline plane vector.
- Slave and master segment normals must face each other; i.e. the normals must be in the opposite direction.

The master and the slave nodes in a contact region must lie in a plane called the slide line plane. The slide line plane is defined by a vector normal to the plane called the slide line plane vector. The master and slave can have large relative motions within the slide line plane. However, relative motion outside the slide line plane are ignored, therefore, must be small compared to a typical master segment. Thus, slide line contact is ideal for modelling interactions between two bodies which may come in contact in specific planes. Simcenter Nastran checks to make sure that in the initial geometry all the nodes of a contact region lie in the slide line plane. However, no such check is made during analysis.

In general, contact is determined only for slave nodes and the master line. This may result in master nodes penetrating the slave line. However, the error involved depends only on the mesh discretization. At the expense of increase in CPU cost, a user may wish to determine contact between master nodes and the slave line in addition to determining contact between slave nodes and the master line. This is called symmetric penetration. Thus, for symmetric penetration, no distinction is made between slave and master.

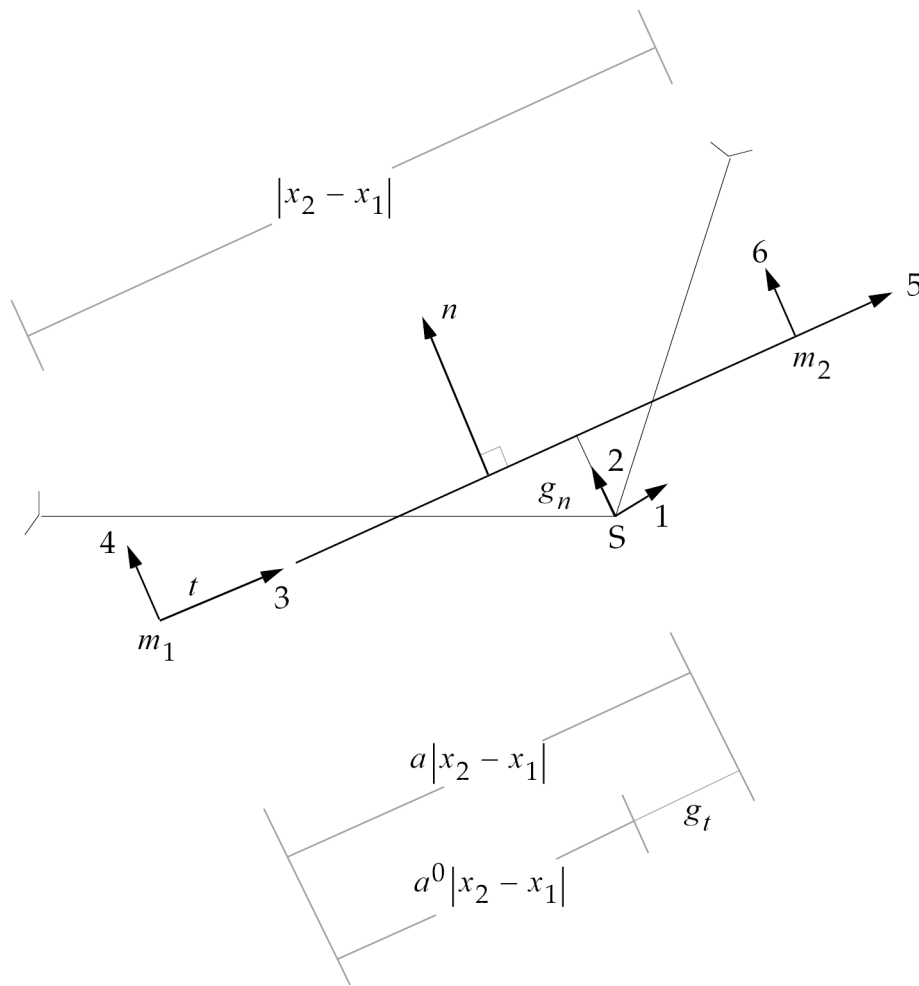
To determine contact between the slave nodes and the master line, a local coordinate system is created for each master segment. The tangent direction for the master segment is given by the vector from its first node to the second node. Remember that the master nodes are specified in the topological order. The normal direction to the master segment is determined by the cross product between the slide line plane vector and the tangent vector. This normal direction must point from

master line to the slave line. This is because Simcenter Nastran determines the contact between slave nodes and the master line by measuring how close the slave nodes are to the master segment in this normal direction.

For symmetric penetration, similar to the master segments, tangent and normal directions are also defined for slave segments. Now, the normals of master segments and slave segments must face each other as shown in [Figure 5-1](#). This is generally accomplished by traversing from master line to slave line in a counter-clockwise fashion, or clockwise fashion depending on whether the slideline plane vector forms a right hand or a left hand coordinate system with the slide line plane. It is your responsibility to make sure that the normals face each other. Simcenter Nastran checks indirectly to make sure that the normals face each other by calculating the initial gap between the two bodies. However, this is not a foolproof check, especially when there is no initial clearance.

Initial penetration between the two bodies is not allowed. The initial gaps between the slave nodes and the master line are calculated based on the coordinates specified for the slave and the master nodes. Thus, if there is no initial gap, Simcenter Nastran may calculate initial penetration if the coordinates are not specified accurately. To avoid having you calculate the coordinates very accurately, Simcenter Nastran adjusts the coordinates automatically and issues a warning message if the initial penetration of a slave node is less than ten percent of the length of the master segment to which it projects. However, if the initial penetration is larger than ten percent, the analysis is terminated and a fatal message is issued. Sometimes, the analysis may be terminated with this message even though there is a large initial gap between the two bodies. This implies that the normals of master or slave segments are not pointing in the correct direction.

Once a contact between a node and a body is detected, a three node slide line element as shown in [Figure 5-2](#) is automatically created. The first node of this element is the slave node and the last two nodes are the master nodes of the master segment with which the slave node is in contact. The slave node is forced to slide on the master line after contact and must remain on the master line until a tensile force develops.



- S, m_1, m_2 = slave, master node 1 and master node 2, respectively
- a, a_0 = current and previous surface coordinate
- g_n = penetration of slave node into the master segment
- g_t = sliding of the slave node on the master segment
- n = normal direction for the master segment

Figure 5-2. Geometry and Definition of a Typical Three Node Slide Line Element

In Simcenter Nastran the contact and traction forces/stresses are always associated with slave nodes. In order to compute the contact and traction stresses, an area is associated with each slave node, based on the contributory length and the width/thickness from the adjacent slave segments. Therefore, you need to specify widths associated with slave line in a three dimensional model or thickness associated with slave line in a two dimensional model.

You can request to output results for any number of slave nodes in any number of slide line contact regions. The output for a slave node consists of (a) the slide line contact region identification number, (b) the master segment to which it projects, (c) the parametric surface coordinate to identify

the exact projection of slave node relative to the two master nodes of the master segment, (d) contact and tangential forces and stresses in the element coordinate system, and (e) a slip ratio to indicate whether the slave node is sticking, slipping or sliding. The slip ratio is defined as the (shear force)/(coefficient of friction * normal force). Thus, a ratio of 0. indicates no shear or no contact. A slip ratio of 1. indicates slipping. A value between 0. and 1. is a normalized value which indicates how close the value of the shear force is to impending slipping.

The slide line contact formulation is based on the penalty method which uses an artificial stiffness value. However, you don't need to provide penalty values. Simcenter Nastran automatically calculates the appropriate numbers. You have the option to scale the calculated penalty values. It is possible that the automatically calculated penalty values, may sometimes, cause convergence difficulties. However, you can overcome the convergence problems by restarting the analysis from any previous converged solution and scaling down the penalty values. Generally, it is recommended to scale down the penalty values by one order of magnitude at a time.

Diagnostic Output

Diagnostic output related to 3-D slide line contact can be obtained by setting the executive control statement DIAG to 35.

DIAG 35 Prints current coordinates of contact node and other information associated with slave nodes to debug slide line contact algorithm for each iteration for slide line contact.

Limitations

The slide line capability is only available for SOL 106 and SOL 129. For all other solution sequences all the Bulk Data entries and Case Control commands related to slide line contact are ignored.

Contact stress or force for any slave node could be plotted using XY PLOT BOUT RESP command. The grid ID is the slave node external grid ID. However, the slave node requested for plotting must also be requested for printout via the BOUTPUT Case Control command.

Defining 3D Slideline Contact in the Input File

Table 5-1 shows how the slideline input data is given to Simcenter Nastran through executive and case control as well as the bulk data entries. For a complete list of these input entries, see the *Simcenter Nastran Quick Reference Guide*.

Table 5-1. Simcenter Nastran Input for Nonlinear Slideline Contact

Slideline Input	Simcenter Nastran Input
Analysis Type and Diagnostics Request	Executive Control Parameters:
<ul style="list-style-type: none"> Nonlinear Statics Nonlinear Dynamics Slideline Diagnostics 	<ul style="list-style-type: none"> SOL 106 SOL 129 DIAG 35 – Provides extended slideline diagnostics output.

Table 5-1. Simcenter Nastran Input for Nonlinear Slideline Contact

Slideline Input	Simcenter Nastran Input
Slideline Output Request	<p>Case Control Parameters:</p> <p>BOUTPUT – Selects the contact regions for history output.</p> <p>Bulk Data Entries:</p>
Contact Parameters	BCONP– Defines the parameters for contact between two bodies.
Boundary Line Segments	BLSEG – Defines the curve, which consists of a number of line segments via grid point numbers that may come in contact with other body.
Boundary Line Segments Width/Thickness	BWIDTH – Defines width/thickness for line segments in 3-D/2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.
Contact Friction	BFRIC – Defines frictional properties between two bodies in contact.
Contact Output	BOUTPUT – Defines the slave nodes at which the output is requested.
User Control and Results Processing; Parameters	ADPCON – (Default = 1.0); Automatically calculated penalty values can be scaled by specifying a real value for the ADPCON. This is useful for changing the penalty values on restart. This will change the penalty value to be equal to a number calculated by Simcenter Nastran times SFAC value specified in BCONP Bulk Data entry times ADPCON . ADPCON is applied to all the contact regions.

The slideline contact formulation is based on the penalty method, which uses an artificial stiffness value. Simcenter Nastran will automatically calculate the appropriate numbers based on local stiffness. You have the option to scale the calculated penalty values. The automatically calculated penalty values may sometimes cause convergence difficulties. However, you can overcome the convergence problems by restarting the analysis from any previous converged solution and scaling down the penalty value. Generally, you should decrease the penalty values by one order of magnitude at a time.

3D Slideline Contact Example

An elastic sphere in contact with a rigid plane is pressed down with a uniform pressure at the top. No friction is assumed between the sphere and the rigid plane. The analytical solution to this problem were first published by Timoshenko, S. P., and Goodier, J. N., *Theory of Elasticity*, McGraw-Hill Book Company, 1970, pgs. 409-420.

For demonstration purposes, a symmetric half of a ten degree sector of the sphere is modeled by CHEXA and CPENTA elements with axisymmetric boundary conditions (see [Figure 5-3](#)). Uniform vertical displacements of the symmetry plane is maintained by tying the vertical displacements of all the grid points on the top plane (loading plane) by MPC relations to the one at the center line (z-axis).

A total load of 2160 is applied at the top of sphere. Because only a ten degree sector is modeled and because of the MPC relationships, a load of only 60 is applied at the top center grid point. This load is applied in three subcases with five load increments per subcase.

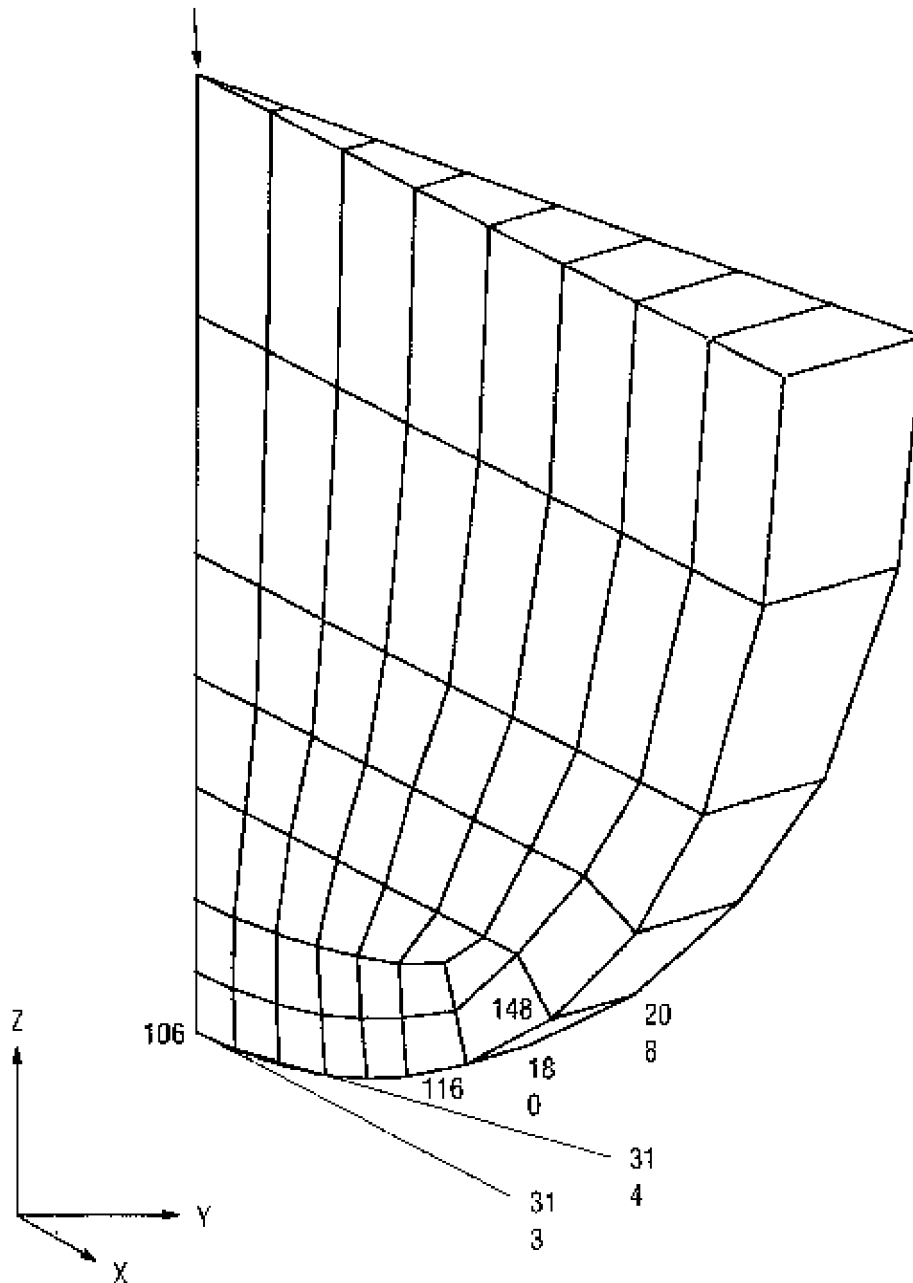


Figure 5-3. Model of a Sphere in Contact with Rigid Plane

To model the contact, two slide line regions are defined via the BCONP Bulk Data entry. The first slide line contact region models the contact between the outer grid points of the sphere at $\theta = 0^\circ$ and the rigid plane as shown in [Figure 5-4](#). The outer grid points of the sphere (148, 116, 114, 112, 110, 108, 105) are used to define the slave line. To define the master line a grid point 313 at $\theta = 0^\circ$ is defined on the rigid plane. The master line is defined by the lower most center grid point of the sphere (grid point 106) and the grid point 313. Notice that the grid points on the master and slave are

defined in topological order traversing in counter-clockwise form the master line to the slave line (see BCONP with ID = 10, BLSEG with ID's = 10 and 20 in the example input). This is because the slide line plane is the R-Z plane at $\theta = 0^\circ$ and the slide line plane vector forms a right hand system with the slide line plane (see CORD2R with ID = 1 in the example listing).

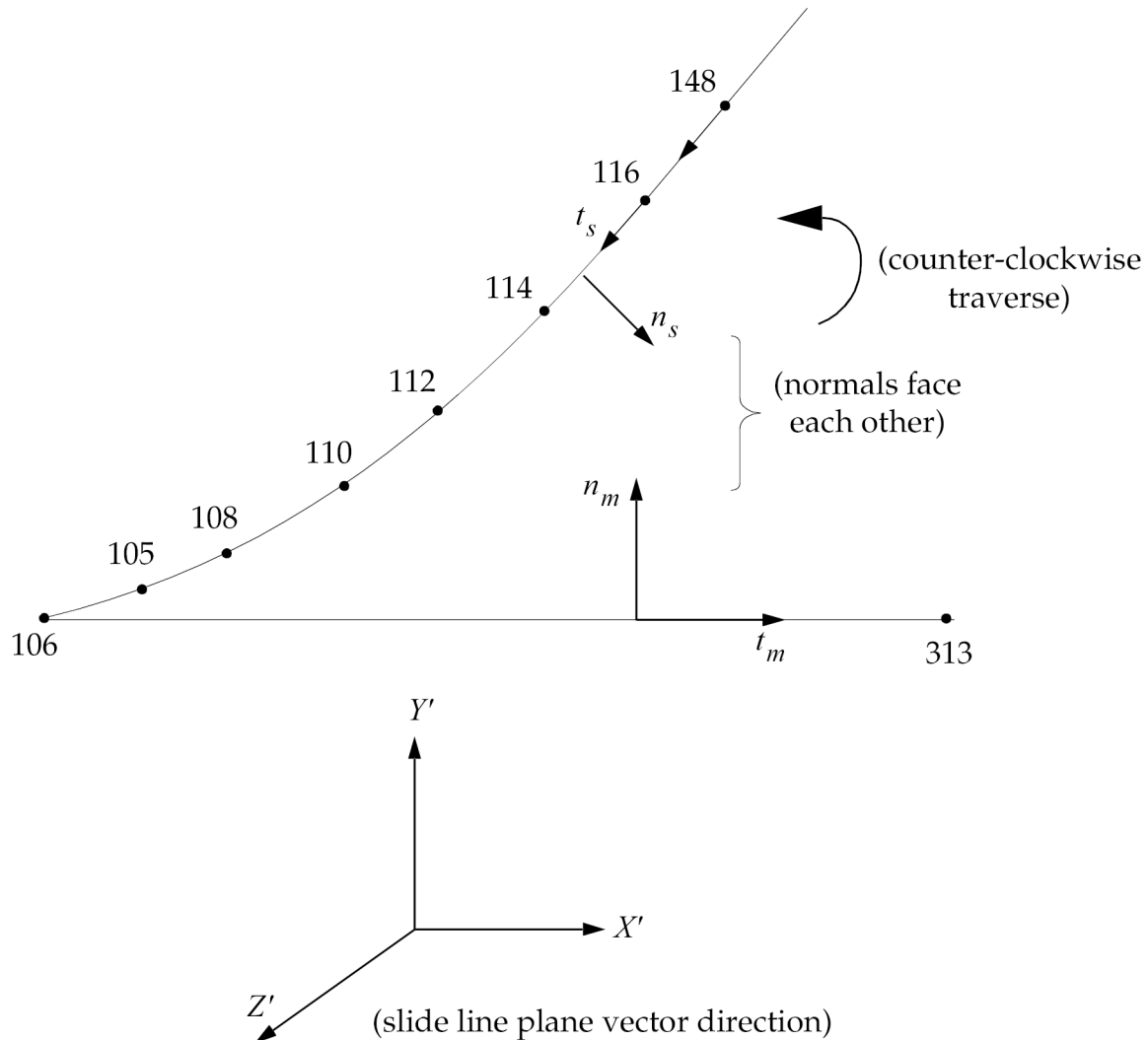


Figure 5-4. Slide Line Contact Region One

The second slide line contact region models the contact between the outer grid points of the sphere at $\theta = 0^\circ$ and the rigid plane as shown in Figure 5-5. Another grid point 314 at $\theta = 10^\circ$ on the rigid plane is defined to model the master line. This time we traverse clockwise form the master line to the slave line in topological order (see BCONP with ID = 20 and BLSEG with ID's = 30 and 40 in the example input) because the slide line plane vector forms a left hand system with the slide line plane (see CORD2R with ID = 3 in the example input).


```

SPC = 4
MPC = 100
SUBCASE 1                                $ 20 LBS
  NLPARM = 410
  LOAD = 100
SUBCASE 2                                $ 40 LBS
  NLPARM = 420
  LOAD = 200
SUBCASE 3                                $ 60 LBS
  NLPARM = 430
  LOAD = 300
OUTPUT (XYPLOT)
  XTITLE = LOAD AFCTOR
  YTITLE = CONTACT NORMAL FORCE
  XYPLOT BOUTPUT RESP/108(6)
  XYPLOT BOUTPUT RESP/105(6)
  YTITLE BOUTPUT ELEMENTS NORMAL STRESS - X
  XYPLOT BOUTPUT RESP/9001(6),9002(6)
BEGIN BULK
$
$ SLIDELINE CONTACT related bulk data entries
$
BCONP,1,10,20,,      ,,1,1
BCONP,2,30,40,,      ,,1,3
BLSEG,10,148,116,114,112,110,108,105 $
BLSEG,20,106,313 $
BLSEG,30,170,172,174,176,178,180,208 $
BLSEG,40,314,106 $
$
$ COORDINATE SYSTEM FOR SLIDE LINE PLANE VECTOR
$
CORD2R,1,2,0.,.0,.0,1.,-90.,.0,+C2
+C2,1.,0.,0.0
CORD2R,3,2,0.,.0,0.,.0,1.,100.,.0,+C3
+C3,1.,0.,.0

```

Listing 5-1. Slideline Example Problem of Sphere Contacting a Flat Plate

Output Formats

Sort1 Output

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID PLANE WITHOUT FRIC											JULY 23, 2003 NX NASTRAN		07/23/03		PAGE 109	
											SUBCASE 3					
LOAD STEP = 3.00000E+00											RESULTS FOR SLIDE LINE ELEMENTS (IN ELEMNT SYSTEM)					
SLAVE GRID	CONTC ID	MASTER GRID1	GRID2	SURFACE COORDINATE	NORMAL FORCE	SHEAR FORCE	NORMAL STRESS	SHEAR STRESS	NORMAL GAP	SLIP RATIO	SLIP CODE					
148	1	106	313	9.7604E-01	.0	.0	.0	.0	-1.1240E+00	1.5218E-02	.0	OPRN				
116	1	106	313	7.3917E-01	.0	.0	.0	.0	-4.1923E-01	-4.1647E-03	.0	OPRN				
114	1	106	313	5.7432E-01	.0	.0	.0	.0	-1.1861E-01	-2.3407E-02	.0	OPRN				
112	1	106	313	4.5907E-01	2.4318E+00	.0	4.1069E+00	.0	1.2792E-04	-3.9671E-02	.0	SLIDE				
110	1	106	313	3.4298E-01	1.1404E+01	.0	1.8499E+01	.0	5.9989E-04	-4.8093E-02	.0	SLIDE				
108	1	106	313	2.1905E-01	1.0069E+01	.0	1.6320E+01	.0	5.2968E-04	-3.5747E-02	.0	SLIDE				
105	1	106	313	1.0478E-01	5.2337E+00	.0	1.7661E+01	.0	2.7531E-04	-1.8078E-02	.0	SLIDE				
170	2	314	106	8.9522E-01	5.2335E+00	.0	1.7660E+01	.0	2.7530E-04	1.8078E-02	.0	SLIDE				
172	2	314	106	7.8095E-01	1.0069E+01	.0	1.6320E+01	.0	5.2968E-04	3.5747E-02	.0	SLIDE				
174	2	314	106	6.5702E-01	1.1404E+01	.0	1.8499E+01	.0	5.9989E-04	4.8093E-02	.0	SLIDE				
176	2	314	106	5.4093E-01	2.4318E+00	.0	4.1069E+00	.0	1.2792E-04	3.9671E-02	.0	SLIDE				
178	2	314	106	4.2568E-01	.0	.0	.0	.0	-1.1861E-01	2.3407E-02	.0	OPRN				
180	2	314	106	2.6083E-01	.0	.0	.0	.0	-4.1923E-01	4.1647E-03	.0	OPRN				
208	2	314	106	2.3956E-02	.0	.0	.0	.0	-1.1240E+00	-1.5218E-02	.0	OPRN				

SORT2 Output

NONLINEAR STATIC CONTACT OF A SPHERE ON A RIGID PLANE WITHOUT FRIC											JULY 23, 2003 NX NASTRAN		07/23/03		PAGE		99	
SLAVE GRID-ID = 110											SUBCASE 1							
RESULTS FOR SLIDE LINE ELEMENTS (IN ELEMENT SYSTEM)																		
STEP	CONTAC	MASTER	SURFACE	NORMAL	SHEAR	NORMAL	SHEAR	NORMAL	SLIP	SLIP	SLIP							
ID	GRID1	GRID2	COORDINATE	FORCE	FORCE	STRESS	STRESS	GAP	RATIO	CODE								
2.000E-01	1	106	313	3.5166E-01	.0	.0	.0	-1.0336E-01	-4.6982E-03	.0	OPEN							
4.000E-01	1	106	313	3.5067E-01	.0	.0	.0	-7.1160E-02	-9.6436E-03	.0	OPEN							
6.000E-01	1	106	313	3.4968E-01	.0	.0	.0	-4.3237E-02	-1.4611E-02	.0	OPEN							
8.000E-01	1	106	313	3.4868E-01	.0	.0	.0	-1.5314E-02	-1.9579E-02	.0	OPEN							
1.000E+00	1	106	313	3.4787E-01	5.6225E-01	.0	9.1206E-01	.0	2.9576E-05	-2.3653E-02	.0	SLIDE						
1.200E+00	1	106	313	3.4727E-01	1.8074E+00	.0	2.9318E+00	.0	9.5072E-05	-2.6643E-02	.0	SLIDE						
1.400E+00	1	106	313	3.4667E-01	3.0525E+00	.0	4.9516E+00	.0	1.6057E-04	-2.9632E-02	.0	SLIDE						
1.600E+00	1	106	313	3.4608E-01	4.2976E+00	.0	6.9713E+00	.0	2.2606E-04	-3.2622E-02	.0	SLIDE						
1.800E+00	1	106	313	3.4548E-01	5.5427E+00	.0	8.9911E+00	.0	2.9156E-04	-3.5612E-02	.0	SLIDE						
2.000E+00	1	106	313	3.4488E-01	6.7878E+00	.0	1.1011E+01	.0	3.5706E-04	-3.8601E-02	.0	SLIDE						
2.200E+00	1	106	313	3.4428E-01	8.0329E+00	.0	1.3031E+01	.0	4.2255E-04	-4.1591E-02	.0	SLIDE						
2.400E+00	1	106	313	3.4368E-01	9.2780E+00	.0	1.5050E+01	.0	4.8805E-04	-4.4580E-02	.0	SLIDE						
2.600E+00	1	106	313	3.4322E-01	1.0329E+01	.0	1.6756E+01	.0	5.4336E-04	-4.6913E-02	.0	SLIDE						
2.800E+00	1	106	313	3.4310E-01	1.0867E+01	.0	1.7628E+01	.0	5.7162E-04	-4.7503E-02	.0	SLIDE						
3.000E+00	1	106	313	3.4298E-01	1.1404E+01	.0	1.8499E+01	.0	5.9989E-04	-4.8093E-02	.0	SLIDE						

5.3 Nonlinear Static Analysis

In Simcenter Nastran, you can perform nonlinear static analysis using SOLs 106, or 153. Modeling options are compatible with linear analysis. For example, you can convert a SOL 101 model to a SOL 106 analysis by adding a few bulk data entries relevant to the nonlinear analysis to your input file.

These nonlinear properties and/or effects are defined by nonlinear material data (e.g., MATS1, MATHP, CREEP, and TABLES1), gap elements (GAP), slideline contact (BCONP, BLSEG and BFRIC) for nonlinear interface, and PARAMeter LGDISP for geometric nonlinearity.

Case Control

Each subcase defines a set of loading and boundary conditions which can be subdivided into a number of increments. Loading conditions are specified using the same loads as in linear static analysis, i.e., FORCEi, MOMENTi, PLOADi, GRAV, TEMP, etc. Constraints can be specified using SPCs and MPCs, which can be changed from subcase to subcase. The enforced motion can be applied using SPCD or nonzero value in SPC. The specified value for SPCD or SPC is a measure of displacement with reference to the undeformed position. The enforced motion is also subdivided in the subcase like the incremental loads. The unique data required for SOL 106 is supplied on the NLPARM entry, which controls the incremental and iterative solution processes. The snap-through analysis or post-buckling analysis can be performed using arc-length methods which are controlled by parameters in the NLPCI Bulk Data entry.

The load step is labeled by the cumulative load factor. The load factor varies from 0 to 1 in each subcase. Then the load step will end with 1, 2, 3, etc. for the first, the second and the third subcase, respectively. There are advantages to divide the entire loading history into many subcases so that each subcase does not have excessive number of increments, e.g., not to exceed 20 steps. The data blocks containing solutions can be generated at each increment or the end of each subcase depending on the intermediate output option specified in the INTOUT field of NLPARM data entry. Data blocks are stored in the database for output process and restarts. As such, the database size or the output quantities are better controlled with multiple subcases. Furthermore, the nonproportional loading can only be achieved by using multiple subcases.

Output requests for each subcase are processed independently. Requested output quantities for all the subcases are appended after the computational process for actual output operation. Available

outputs are DISPLACEMENT, OLOAD, STRESS, FORCE, GPSTRESS, and SPCFORCE. However, FORCE and GPSTRESS output are available only for linear elements. It should be noted that the STRESS or STRESS(PLOT) command will cause the nonlinear stress to be printed. In other words, the STRESS(PLOT) command suppresses the combined linear/nonlinear stress output only.

Restarts are controlled by parameters (LOOPID and SUBID) which can be specified either in the Case Control Section or Bulk Data Section. Some optional parameters (ADPCON, BUCKLE, LANGLE, MAXLP, NMLOOP) are provided for additional control or capabilities. All the superelement model generation options and matrix reduction options are allowed for the linear portion of the structure.

Restrictions and Limitations in SOL 106

- Only CROD, CONROD, CTUBE, CBEAM, CGAP, CQUAD4, CTRIA3, CTRIA6, CHEXA, CPENTA, and CTETRA elements may be defined with material or geometric nonlinear properties. All other elements will be treated as having small displacements and linear materials.
- The CBEAM is assumed to have its nonlinear region concentrated at the ends. Taper is allowed but pin joints are illegal if nonlinear material conditions are defined with the MATS1, MATTi, or CREEP entries. Pin joints can be used if the only nonlinearity is geometric. The specification of offset vectors is not recommended. When geometry nonlinear conditions exist (PARAM,LGDISP,1), the offset vectors remain parallel to their original orientation when computing the differential stiffness. Also, when material nonlinear conditions are defined with the MATS1 or CREEP entries, the offset vectors may produce incorrect results. As a result, the specification of offset vectors is not permitted in these solutions. However, setting SYSTEM(463) to 1 will disable the FATAL error message and the analysis will be allowed to continue (with offset vectors remaining parallel to their original orientation). The use of this system cell may generate incorrect results. Loading conditions that generate follower forces should not be used when SYSTEM(463) = 1. Use PARAM,FOLLOWK,NO if follower boundary conditions are specified.
- Constraints apply only to the non-rotated displacements at a grid point. In particular, multipoint constraints and rigid elements may cause problems if the connected grid points undergo large motions. However, also note that replacement of the constraints with overly stiff elements can result in convergence problems.
- Large deformations of the elements may cause non-equilibrium loading effects. The elements are assumed to have constant length, area, and volume, except for hyperelastic elements.
- All restarts with grid point or element changes must start over at the first subcase.
- When TEMPi bulk entries are used to define thermal loads, you have the following restart options:
 - o If TEMPi bulk entries are defined in the cold-start run only, you can restart from any LOOPID.
 - o If TEMPi bulk entries are defined in the restart-start run only, you must restart from the beginning of the solution.
 - o If TEMPi bulk entries are defined in both runs and they have the same SID, a fatal error occurs.
 - o If TEMPi bulk entries are defined in both runs and they have different SIDs, you must restart from the beginning of the solution.

- For large displacement analysis, there are two different approaches for the angular motions: gimbal angle and rotation vector approach. In Gimbal angle approach angular motions are treated as three ordered rotations about the z, y, and x axes. In rotation vector approach the three angular motions are treated as a vector. The rotation is about the rotation axis and the magnitude of rotation is equal to the amplitude of rotation vector.
- The CGAP element contact planes do not rotate as a function of displacement. The actual shapes of the contacting surfaces would be required for such calculations.
- Since the solution to a particular load involves a nonlinear search procedure, solution is not guaranteed. Care must be used in selecting the search procedures on the NLPARM data. You can override nearly all iteration control restrictions.
- Follower force effects are calculated for loads which change direction with grid point motion.
- All cold-start runs are assumed to start in a stress-free condition with zero displacements and rotations. Prestressed and pre-loaded cases must be simulated by running them as the initial load case, followed by the desired solution subcase.
- Every subcase must define a new total load or enforced boundary displacement. This is necessary because error ratios are measured with respect to the load change.
- All superelements, except for the residual (SE = 0), are assumed to be linear. All existing superelement restrictions also apply here.
- Any changes to a non-residual superelement requires the use of the appropriate SEMG, SEMA, SELA, etc., case control request for efficient restarts (see also [“Performing Nonlinear Analysis With Superelements”](#)).
- Only the residual superelement has automatic output data processing. Upstream superelements may be loaded only via LSEQ and CLOAD data.
- In postbuckling analysis with arc-length methods (NLPCI Bulk Data entry), the iteration algorithm may jump between different buckling modes when multiple bifurcation points are present in the structure. It is recommended to check all possible buckling modes with the linear buckling solution sequence SOL 105 before making a run with SOL 106. You have to introduce initial imperfections (loads or enforced displacements) to keep the structure in the intended buckling mode.

Controlling Iterations with the NLPARM Entry

The most crucial data for successful nonlinear static solutions are contained in the NLPARM bulk data entry. NLPARM defines strategies for the incremental and iterative solution processes. It is difficult to choose the optimal combination of all the options for a specific problem. However, based on a considerable number of numerical experiments, the default option was intended to provide the best workable method for a general class of problems. Therefore, if you have little insight or experience in a specific application, you should start with the default option. However, if you have some experience and insight in a specific problem, you can change the default values, if you keep in mind the following guidelines:

- Computing cost for each line search is comparable to that of an iteration.

- The SEMI method usually provides better convergence than the AUTO method at the expense of higher computing cost.
- Default tolerances for the convergence criteria may be somewhat conservative. However, loose tolerances may cause difficulties in the subsequent steps.
- The quasi-Newton method is effective in most problems. However, it seems to have adverse effects in some problems, e.g., creep analysis.
- The line search method is effective to cope with difficulties in convergence in some problems. More extensive line searches may be exercised by a large value of MAXLS and/or a smaller value of LSTOL. On the other hand, line searches may have adverse effects in some problems, e.g., plane stress plasticity.
- The arc-length method should be used if the problem involves snap-through or postbuckling deformation. Then the Bulk Data entry NLPCI must be attached.

The NLPARM entry format is shown below with default values wherever applicable:

1	2	3	4	5	6	7	8	9	10
\$NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
NLPARM			0.0	AUTO	5	25	PW	NO	

\$	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	1.0E-3	1.0E-3	1.0E-7	3	MAXITER	4	0.2	0.5	

\$	MAXBIS				MAXR		RTOLB		
	5				20.		20.		

The ID field is referenced by the NLPARM Case Control command. The NINC field is an integer which specifies the number of increments to be processed in the subcase. The total load specified in the subcase minus the load specified in the preceding subcase is equally divided by this integer (NINC) to obtain the incremental load for the current subcase. Another subcase should be defined to change constraints or loading paths. However, multiple subcases may be required in the absence of any changes in constraints or loads to use a moderate value (e.g., not to exceed 20) for NINC. Use of a moderate value has advantages in controlling database size, output size, and restarts.

The DT field requires a real number specifying the time increment for each load step in the case of creep analysis. The time unit must be consistent with the unit used in the CREEP data to define the creep characteristics. The creep time increment should be determined properly for efficiency and accuracy based on the creep rate represented by the creep law to be used. The larger the creep rate, the smaller the time increment should be. No creep deformation is considered with the default value of 0. It is noted that a creep analysis should be preceded by a static analysis with at least one subcase, because the creep deformation requires stresses in the material.

Stiffness matrix update strategies are determined by a combination of the data specified in the two fields KMETHOD and KSTEP. Options for KMETHOD are AUTO, SEMI, or ITER. The KSTEP field, which is an auxiliary to the KMETHOD field, should have an integer equal to or greater than 1. With the AUTO option, the program automatically determines when to reevaluate the stiffness matrix based on the rate of convergence. At each iteration, the computing time for convergence without the stiffness matrix update is estimated and compared with the computing time for the matrix update

in order to determine whether the update is more efficient. This decision is deferred in the first two iterations after a new stiffness is obtained. If the solution tends to diverge, however, the update decision will be made effective immediately. The stiffness matrix will be updated upon convergence if the number of iterations required for convergence is greater than KSTEP. The SEMI option is identical to the AUTO option except for one additional stiffness update after the first iteration which always occurs unless the solution converges in a single iteration. With the ITER option, the stiffness matrix is updated at every KSTEP iterations. Thus, the full Newton-Raphson iteration is exercised if KSTEP is 1. If $KSTEP > MAXITER$, the stiffness will never be updated.

The MAXITER field is an integer representing the number of iterations allowed for each load increment. If the number of iterations reaches MAXITER without convergence, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is reached or $MAXBIS = 0$) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated.

Simcenter Nastran performs the convergence test at every iteration with the criteria you specify in the CONV field. You can specify any combination of U (for displacement), P (for load), and W (for work). All the specified criteria must be satisfied to achieve convergence, except for an absolute convergence condition, under which the solution is converged regardless of criteria. The convergence tolerances are specified in the fields EPSU, EPSP, and EPSW for U, P, and W criteria, respectively.

Default nonlinear tolerances in Solutions 106 and 153 will vary according to the model, analysis type, and user-defined precision. Although you can override default tolerances by entering values of CONV, EPSU, EPSP, and EPSW on the NLPARM entry, you can apply default tolerances with a single parameter, leaving the above fields on the NLPARM entry blank. Depending on the desired level of accuracy and run times, various default tolerance sets may be selected. These tolerances are designated as “very high,” “high,” “engineering design,” and “preliminary design” and are controlled with the parameter:

```
PARAM, NLTOL, ITOL
```

where ITOL is an integer value ranging from 0 (very high) to 3 (preliminary design). The default is 2 (engineering accuracy) for nonlinear static analysis and 0 (very high) for nonlinear heat transfer and dynamic analyses.

See Also

- NLTOL in the *Simcenter Nastran Quick Reference Guide*

The *Simcenter Nastran Quick Reference Guide* lists the tolerances set by Simcenter Nastran if you leave the CONV, EPSU, EPSP, and EPSW fields on the NLPARM entry blank. See Remark 16 of the NLPARM entry for further details.

To use these default tolerances, leave the CONV, EPSU, EPSP, and EPSW fields on the NLPARM entry blank. In addition, the NINC field must also either be blank or have a value of 10 or larger. There are two reasons for this:

- In general, approximately 10 load steps or more are required for proper accuracy and convergence with the default tolerances.
- Some users may have already “calibrated” a model as to the number of load steps necessary for good results the “very high” accuracy tolerance.

Therefore, if you set NINC between 1 and 9, the software uses the “very high” accuracy tolerance.

If you don't specify PARAM,NLTOL, then the default convergence tolerances are the same as if you had entered PARAM,NLTOL,2 except for heat transfer problems, for which the default is PARAM,NLTOL,0. The type of analysis will be detected automatically by Simcenter Nastran and the default tolerances adjusted accordingly.

Dynamic problems using SOL 129 and 159 retain the default tolerances as described in:

- "TSTEPNL" in the *Simcenter Nastran Quick Reference Guide*

If you have a good idea of what to use for the default tolerances and/or number of increments, then you should use these. If the problem is nearly linear, a value of NINC=1 sometimes yields acceptable results. For small problems, you should use the "very high" tolerance option because run times are generally not significant.

Because nonlinear tolerances depend on NLTOL, NINC (on the NLPARM entry), and other possible NLPARM overrides, convergence tolerances actually used by the code are echoed in the output file (right after the Bulk Data echo, if any) as shown in the following example:

```

DEFAULT NONLINEAR TOLERANCES
  NLPARM ID      10
  NINC           10
  EPSU           1.00000E-03 (not normally used)
  EPSP           1.00000E-03
  EPSW           1.00000E-05
BASED ON
MODEL HAS GAPS AND/OR LINE CONTACT
(VALUES YOU ENTER OVERRIDE THE ABOVE DEFAULTS)

```

Examples of Convergence Tolerances

A very large nonlinear model consisting of approximately 950,000 DOFs was studied extensively to determine the effect of various tolerances on run time and accuracy. This particular model did not employ heat transfer, gaps, or contact. Results of the study showed that compared to the present "very high accuracy" solution, acceptable accuracy can be achieved for both the "high accuracy" and "engineering accuracy" designations. The "preliminary design" accuracy tolerances produces results with the same general trends as other reasonable tolerances, but the answers were considered too inexact for analyses other than preliminary design trade-off studies. The following table summarizes the effect the tolerances for this large model.

Accuracy Type	Tolerances		Run time CPU hours	Accuracy for the large model
	EPSP	EPSW		
Very high	0.001	1.0E-7	163.2	Very Good
High	0.01	0.001	10.3	5% or less accurate
Engineering Design	0.01	0.01	7.0	10% or less accurate
Preliminary Design	0.1	0.1	2.9	10% to 20% less accurate

Many test cases—including standard nonlinear QA test cases—were also analyzed using the default tolerances. For all models run to date, satisfactory results were obtained. For those models that invoked the new "engineering design" tolerances (because NINC, CONV, EPSU, EPSP and EPSW were blank) results changed as expected. Typically, such changes were largest, on a percentage basis, for the smallest responses in the model. Maximum response values experienced little change.

For example, if the largest displacement was 0.01, then displacements with values around 0.01 changed by about 1% or less, but displacements that were originally 0.0001 might change by 20% or even more. If the maximum stress was 20,000, then stresses around 20,000 would change by 1% or less, but stresses around 1000 might change by 20% or more. If the problem produced primarily displacements in the Y direction, then using the “engineering tolerances” gives larger cross-axis response in the X and Z directions than do the “very high” accuracy tolerances. Stresses behave in a similar manner. However, deformed contour plots of most models using the two accuracy values show little or no difference. If cross-axis responses or if stresses away from the most highly stressed regions are important, or if the nonlinear material properties of the model are known with great precision, the “very high” accuracy tolerances should be used. However for most analyses, the “engineering design” tolerances will often suffice.

The INTOUT field requires a specification of YES, NO or ALL. If YES is specified, the output requests (DISPLACEMENT, FORCE, STRESS, etc.), specified in the Case Control Data, will be processed for every computed converged solution in the subcase. If NO is specified, the output requests will be processed only for the last load step of the subcase. If ALL is specified, the output requests are processed for every computed and user-specified load increment. For Newton's iteration methods (i.e., without NLPCI), the option ALL is equivalent to the option YES, since the computed load increment is equal to load increment you specified. For arc-length methods (i.e., NLPCI command is specified), the computed load increment in general is not the same as the user-specified load increment, and is not known in advance. The option ALL allows you to obtain solutions at the desired intermediate load increments. With a small database option, the database will retain only those data blocks corresponding to the incremental load steps for which output is requested by an INTOUT field. Then, the restarts will be restricted to those starting from these steps. However, if the job is terminated with an incomplete subcase for some reason (e.g., diverging, time expiration, etc.), data blocks for all the interim steps of that subcase are stored in the database to allow flexible restarts.

The MAXDIV field requires an integer to specify a limit on the probable divergence conditions allowed for each iteration to continue. There are two classifications in the divergence condition: probable and absolute. The absolute divergence is treated as two occurrences of the probable divergence. When the probable divergence occurs MAXDIV times or more and if MAXDIV is positive, the current solution base (displacements) is retracted and the stiffness matrix is updated in reference to the preceding iteration step. If the solution diverges again in the same load increment while MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated on second divergence.

The MAXQN field requires an integer to specify the maximum number of quasi-Newton vectors to be saved in the database. Quasi-Newton vectors are accumulated, if desired, until MAXQN is reached. If the number of accumulated QN vectors is greater than or close to MAXQN upon convergence, the stiffness matrix will be updated before the next increment is processed. All the QN vectors already accumulated are purged upon a stiffness update and the accumulation is reinitiated. The quasi-Newton update may be suppressed with a value of 0 for MAXQN. Details of the quasi-Newton method will be presented in the following section.

The line search is controlled by the fields MAXLS and LSTOL. The maximum number of line searches allowed for each iteration is specified in the MAXLS field. The line search process may be suppressed with a value of 0 for MAXLS. The LSTOL field requires a real number between 0.01 and 0.9 to specify a tolerance for the line search operation. The line search operation will be conducted if the error defining the divergence rate is greater than LSTOL. If the line search convergence is not achieved, the line search continues until the number of line searches reaches MAXLS.

The FSTRESS field requires a real number ($0.0 < \text{FSTRESS} < 1.0$), representing a fraction of the effective stress ($\bar{\sigma}$) used to limit the subincrement size in the material routines. The number of subincrements in the material routines for elasto-plastic or creep deformation processes is determined such that a subincrement in the effective stress is approximately $\text{FSTRESS} \cdot \bar{\sigma}$. This data is also used to establish an error tolerance in the yield function to alleviate accumulation and propagation of the error. If the error in the yield function exceeds $\text{FSTRESS} \cdot \bar{\sigma}$ at the converging state, the program will terminate the job with a fatal error message "ERROR EXCEEDED 20% OF CURRENT YIELD STRESS." If the bisection option is selected, bisection will be activated under this condition.

The MAXBIS field requires an integer to specify the number of bisections allowed for a load or arc-length increment. When the solution diverges, different actions are taken depending on the sign of MAXBIS. If MAXBIS is positive, the stiffness matrix is updated on the first divergence and the load increment is bisected on the second divergence. If MAXBIS is negative, the load increment is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after $|\text{MAXBIS}|$ bisections, the analysis is continued or terminated depending on the sign of MAXDIV.

The MAXR field requires a real number to specify the bounds on the ratio of the adjusted arc-length increment relative to the initial value. In the adaptive load/arc-length increment, the overall upper and lower bounds on the load/arc-length increment in the subcase are defined as

$$\frac{1}{\text{MAXR}} \leq \frac{\Delta l_n}{\Delta l_0} \leq \text{MAXR}$$

Equation 5-1.

where Δl_n is the arc-length at step n and Δl_0 is the original arc-length. The arc-length method for load increments is selected by the NLPCI Bulk Data entry, which must have the same ID as the NLPARM Bulk Data entry.

The RTOLB field requires a real number to specify the incremental rotation (in degrees) allowed per iteration. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$ or $\Delta\theta_z$) exceeds the value specified for RTOLB. This bisection strategy based on the incremental rotation is controlled by the MAXBIS field.

User Interface for Arc-Length Methods: NLPCI Entry

Most of the parameters in the NLPARM Bulk Data entry are used to control the iteration strategy for arc-length methods. However, MAXLS is not applicable because the line search procedure is not coupled with arc-length methods currently. The parameters which are applicable only to the arc-length methods are specified in the Bulk Data entry NLPCI, in connection with the NLPARM data by the same ID. The NLPCI entry is shown below with default values, followed by brief description of the fields:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE		DESITER	MXINC	
NLPCI		CRIS	0.25	4.	0.0		12		

ID Must be identical to ID of NLPARM to be active.

TYPE Constraint type (CRIS for Crisfield, RIKS for Riks, or MRIKS for modified Riks method)

MINALR, MAXALR For variable arc-length method, the arc-length is bounded by

$$\text{MINALR} \leq \frac{\Delta l_{new}}{\Delta l_{old}} \leq \text{MAXALR}$$

where $\text{MINALR} \leq 1$. and $\text{MAXALR} \geq 1$. The automatic adjustment is not activated if $\text{MAXALR} = \text{MINALR} = 1$.

SCALE Scaling factor for the magnitude of the load term in arc length. Real > 0 Default = 0. The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that you can make constraint equation unit-independent by a proper scaling of the load factor, μ . As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite w , the arc-length method would be degenerated to the conventional Newton's method.

DESITER Desired number of iterations for convergence to be used for adaptive arc-length adjustment.

$$\Delta l_{new} = \Delta l_{old} \sqrt{\frac{I_d}{I_{max}}}$$

MXINC Maximum number of controlled increments allowed within the specified load step. There can be cases where the structure never snaps back and collapses to failure. In these cases, the analysis cannot be completed without bounding the maximum number of increments. For such cases, this value should be confined to a reasonable limit, e.g., less than 100, to prevent inadvertent depletion of the computing resources.

Iteration Related Output Data

At the end of every iteration, the relevant data from the iteration process are printed under the following heading. The section numbers in parentheses refer to sections in the *Simcenter Nastran Handbook for Nonlinear Analysis*.

ITERATION	iteration count i .
EUI	relative error in terms of displacements (see Section 3.6.3).
EPI	relative error in terms of loads (see Section 3.6.3).
EWI	relative error in terms of work (see Section 3.6.3).
LAMBDA	rate of convergence (defined in Section 3.3.1).
DLMAG	absolute norm of the load error vector.
FACTOR	final value of the line search parameter (see Section 3.4).
E-FIRST	divergence rate, initial error before line search (see Section 3.4.3).
E-FINAL	error at the end of line search (see Section 3.4.3).

N-QNV	number of quasi-Newton vectors appended (see Section 3.5.3).
N-LS	number of line searches performed during the iteration.
ENIC	expected number of iterations for convergence (see Section 3.3.1).
NDV	number of occurrences of probable divergence during the iteration (see Section 3.3.2).
MDV	number of occurrences of bisection conditions in the material routine (excessive stress increment) or in the rotation angle (excessive rotation) during the iteration.

Loads Overview

The static loads in nonlinear analysis consist of a subset of the total Simcenter Nastran static load set. Most of the relevant loads data applicable to the linear static analysis are also applicable to nonlinear static analysis except for the Bulk Data entry DEFORM. The enforced displacements can be applied as loads using the Bulk Data entry SPCD, which defines an enforced motion of a grid in a specific fixed direction. Each grid point with an enforced displacement must also appear on an SPC or SPC1 entry.

The loads are selected in each subcase by a Case Control command LOAD. The load for a subcase is subdivided into the number of increments specified for the subcase. The solution strategy in nonlinear is to apply the loads in an incremental fashion until the desired load level is reached. The algorithms “remember” the loads from one subcase to the next. If the load reaches the desired level in a subcase and if the load description is left out in the subsequent subcase inadvertently, the nonlinear solution algorithm will begin in an incremental fashion to remove the load. The loads described below make up the valid subset applicable for nonlinear static analysis.

Concentrated Loads

The concentrated loads consist of two categories: those which are stationary in direction through out the analysis and those which follow the grid motion. The first category consists of the following: FORCE, MOMENT, and SLOAD.

The second category consists of the following: FORCE1, FORCE2, MOMENT1, and MOMENT2. The three FORCE entries differ only in the manner in which the direction of the force is specified. FORCE uses the components of a vector. FORCE1 uses two grid points, not necessarily the same as the loaded grid points. FORCE2 defines the direction of the force as a vector which is the vector product of two other vectors. The distinctions between the three MOMENT entries are similar. These loads are follower loads in nonlinear and produce follower stiffness terms.

Distributed Loads

The distributed loads consist of two categories: those which are stationary in direction through out the analysis and those which follow the element motion. The first category consists of the PLOAD1.

The second category consists of the following: PLOAD, PLOAD2, and PLOAD4.

Loads may be applied to the interiors of finite elements. The PLOAD1 entry defines concentrated and linearly distributed forces and moments to the CBEAM element. However, the stress calculation for nonlinear CBEAM element ignores the distributed loads, but lumps them onto the nodal points. The PLOAD entry is exceptional in that it references three or four arbitrarily located grids rather than an element. The PLOAD2 entry defines a uniform static pressure load applied to the CQUAD4 or CTRIA3 elements. The PLOAD4 defines a pressure load on surfaces of CHEXA, CPENTA, CTRIA3,

and CQUAD4 elements which need not be normal to the surface. When normal to the surface, the PLOAD4 is a follower load.

Mass Related Loads

The mass related loads are gravity and centrifugal loads, which require specification of mass data. The gravity loads which can be specified in the GRAV Bulk Data entry are stationary in direction through out the analysis. It is used to define the direction and magnitude of the gravity vector in some identified coordinate system. The components of the gravity vector are multiplied by the mass matrix to obtain the components of the gravity force at each grid point.

The centrifugal loads to be specified in the RFORCE Bulk Data entry is used to define the components of a spin vector which is used internally to compute centrifugal forces. Each component of the spin vector is multiplied by a scale factor.

Thermal Loads

You select thermal loads in each subcase with the TEMP(Load) Case Control command. Thermal loading which is a follower loading is accomplished through the following entries: TEMP, TEMPD, TEMPP1, TEMPP3, and TEMPRB. The TEMP and TEMPD provide for grid point temperature specification which are then interpolated to points within elements. For the nonlinear CQUAD4 and CTRIA3 elements the TEMPP1 provides for an average temperature and thermal gradient in the thickness direction and the TEMPP3 provides for temperature at points through the thickness. The TEMPRB provides for average temperature at ends A and B for the CROD and CBEAM element. Average temperature specified directly for an element will take precedence over the temperature interpolated from the elements connected grid points.

The Case Control TEMP(INIT) is required above the subcase level. For nonlinear static analysis TEMP(MAT) or TEMP(BOTH) are not applicable. The use of TEMP(MAT) or TEMP(BOTH) will cause a fatal error. Subcases that do not contain a TEMP(Load) will default to TEMP(INIT), which implies no thermal load. Thermal loads are often used to provide initial strains.

Loads Combination

Various load specifications can be combined using LOAD and (CLOAD, LSEQ).

The LOAD provides for the linear combination of static load sets. The static loads (LOAD, FORCE, etc.) applied to the upstream superelements cannot be referenced by a Case Control command LOAD. The CLOAD entry is designed to apply static loads to upstream superelements by combining loads defined in LSEQ entries. The (CLOAD, LSEQ) combination allows the nonlinear algorithms to apply in an incremental fashion on the upstream superelement loads to the boundary of the residual structure.

The CLOAD defines a static load combination for superelement loads acting on residual boundary and the LSEQ defines a static load combination for superelement upstream loads. The LSEQ assigns load vectors to the superelements and numerically labels them via the DAREA field entry value. The LID and TID field entries point to the appropriate load and temperature distribution entries. The CLOAD defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ. The IDVi fields correspond directly to the numeric labels defined via the DAREA fields of the LSEQ entries.

The CLOAD must be selected in the residual solution subcase via the case control CLOAD entry. Only one LOADSET may appear in Case Control. The hierarchy of the loads data is shown schematically in [Figure 5-6](#). An example Case Control Section is shown below:

```
.  
.   
.   
SEALL=ALL $  
LOADSET = 1000 $ Selects LSEQ 1000 for upstream loads  
SUPER = ALL $ Identify SEs to process  
  
.   
.   
.   
  DISP = ALL  
.   
.   
.   
.   
.   
.   
.   
.   
$ NONLINEAR SOLUTIONS FOR RESIDUAL SUPERELEMENTS  
SUBCASE 10  
  CLOAD = 1001 $ Refers to CLOAD Bulk Data  
  NLPARM = 12 $ Iteration control  
  
.   
.   
.   
SUBCASE 20  
  CLOAD = 1002  
  LOAD = 10 $ Residual SE load  
  NLPARM = 22  
  
.   
.   
.   
.
```

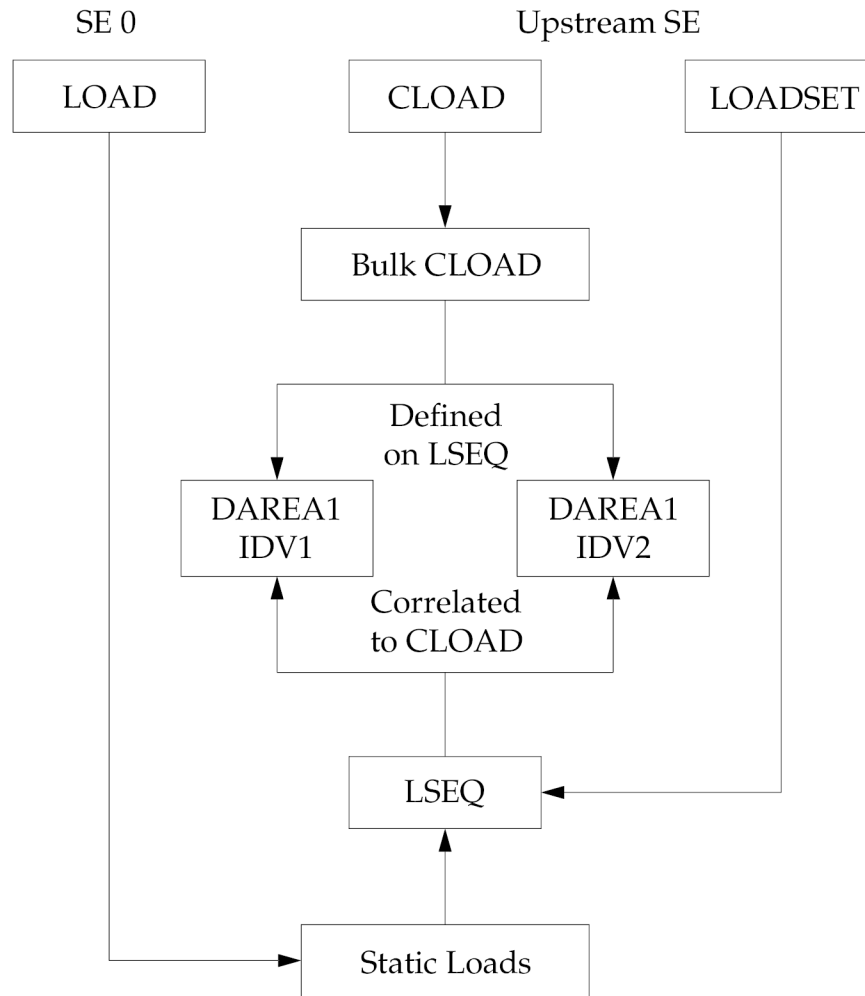



Figure 5-6. Hierarchy of Load Data

Restarts for Nonlinear Static Analysis

Any restart in SOL 106 must have the FMS command RESTART and the two parameters PARAM,SUBID,value and PARAM,LOOPID,value.

The SUBID is the sequential number of a subcase. It is recommended that the SUBID value always be incremented by one and point to a new subcase. The new Subcase should contain any standard subcase commands required.

The LOOPID is the identifier of the converged solution from which the restart is to take place.

Any converged and saved solution always gives the messages:

```

*** USER      INFORMATION MESSAGE 6186,
*** SOLUTION HAS CONVERGED ***

      SUBID 7  LOADINC 5  LOOPID 67  LOAD STEP  7.000  LOAD FACTOR  1.00000

^^^ DMAP INFORMATION MESSAGE 9005 (NLSTATIC) - THE SOLUTION FOR LOOPID= 67
    IS SAVED FOR RESTART
  
```

To restart from the above run, the restart might look like:

```

RESTART
TIME 10000
SOL 106
DIAG 50
CEND
TITLE = SAMPLE RESTART
PARAM, SUBID, 8
PARAM, LOOPID, 67
SPC = 200
MPC = 250
SET 1 = 1118
DISPLACEMENT = 1
SPCF=1
SUBCASE 1
LOAD = 101
NLPARM = 51
.
.
.
SUBCASE 7
LOAD = 107
NLPARM = 57
SUBCASE 8
LOAD = 108
NLPARM = 58
BEGIN BULK
NLPARM 58      5      AUTO      20      20      PW      YES
             1-3    1-3    1-7    -3
SPCD  108      1118    3      0.025
FORCE 108      1118    0.     0.     0.     1.
ENDDATA

```

For a full data recovery restart after the completion of a complete analysis proceed as above but do not actually add the new Subcase in Case Control. For example, assume the analysis has ended in subcase 8 above with the following messages:

```

*** USER      INFORMATION MESSAGE 6186,
*** SOLUTION HAS CONVERGED ***

      SUBID 8  LOADINC 5  LOOPID 72  LOAD STEP  8.000  LOAD FACTOR  1.00000

^^^ DMAP INFORMATION MESSAGE 9005 (NLSTATIC) - THE SOLUTION FOR LOOPID= 72
    IS SAVED FOR RESTART

```

Perform a full data recovery restart as follows:

```

RESTART
TIME 10000
SOL 106
DIAG 50
CEND
TITLE = FULL DATA RECOVERY RESTART
MAXLINES = 10000000
$ POINT TO A NEW SUBCASE BUT DO NOT ACTUALLY ADD A NEW SUBCASE
PARAM, SUBID, 9
PARAM, LOOPID, 72

```

```

SPC = 200
MPC = 250
SET 1 = 1118
$DISPLACEMENT = 1
$ ADD DISPLACEMENT AND STRESS REQUESTS FOR DATA RECOVERY
DISPLACEMENT=ALL
STRESS=ALL
SPCF=1
SUBCASE 1
LOAD = 101
NLPARM = 51
.
.
.
SUBCASE 7
LOAD = 107
NLPARM = 57
SUBCASE 8
LOAD = 108
NLPARM = 58
BEGIN BULK
ENDDATA

```

Note that in the above, there is no SUBCASE 9 added.

The result of this last restart will be the complete displacement and stress results output for each converged solution saved on the data base starting with the first converged solution and ending with the results of the converged solution at LOOPID = 72.

5.4 Nonlinear Buckling Analysis

A simple approach to nonlinear buckling analysis exists in SOL 106 by using a quadratic extrapolation. A similar approach to the limit point calculations was published by Fujikake. The distinction is that Simcenter Nastran uses two converged solutions in the vicinity of the buckling point to form a differential stiffness $[\Delta K]$, whereas Fujikake uses a converged point and an estimated data point beyond the buckling point. To find an instability point within a small range of nonlinear domain, two methods of idealization can be considered:

1. The tangent stiffness matrix is proportional to the external loads, which implies that the critical load may be linearly interpolated by $P_{cr} = P_n + \lambda \cdot \Delta P$.
2. The tangent stiffness matrix is proportional to the displacement increments, which implies that the critical displacements may be obtained by extrapolating from the current state, i.e., $U_{cr} = U_n + \lambda \cdot \Delta U$.

It appears that Fujikake uses the first approach. Simcenter Nastran uses the second approach. Since the tangent matrix is assumed to change linearly, the internal loads are a quadratic function of the displacements. The distinction between the two approaches disappears in the linear bifurcation problem.

Nonlinear Buckling Formulation

The eigenvalue problem for a nonlinear buckling analysis is approximated by

$$[K_n + \lambda \Delta K] \{\phi\} = \{0\}$$

with

$$\Delta K = K_n - K_{n-1}$$

where K_n and K_{n-1} are the stiffness matrices evaluated at the known solution points in the vicinity of the instability.

The critical displacements upon instability are estimated as

$$\{U_{cr}\} = \{U_n\} + \lambda \{\Delta U\}$$

with

$$\{\Delta U\} = \{U_n\} - \{U_{n-1}\}$$

Based on the virtual work principle,

$$\{\Delta U\}^T \{P_{cr}\} = \{\Delta U\}^T \{F_{cr}\}$$

where

$$F_{cr} = F_n + \int_{U_n}^{U_{cr}} K du = F_n + \int_0^\lambda K(\lambda) \Delta U d\lambda \simeq F_n + \lambda [K_n + \frac{1}{2} \lambda \Delta K] \Delta U$$

The critical buckling load may be estimated as

$$\{P_{cr}\} = \{P_n\} + \alpha \{\Delta P\}$$

where P_n is the latest load value which resulted in convergence, and

$$\{\Delta P\} = \{P_n\} - \{P_{n-1}\}$$

where P_{n-1} is the previous load value which resulted in convergence, and

$$\alpha = \frac{\lambda \{\Delta U\}^T [K_n + \frac{1}{2} \lambda \Delta K] \{\Delta U\}}{\{\Delta U\}^T \{\Delta P\}}$$

where λ is a linear buckling load factor computed from a linear buckling analysis in which P_n was used as the applied load. The linear buckling analysis can be requested in any nonlinear buckling subcase, and occurs after all of the nonlinear load steps in the subcase have been applied.

Restart is required for a nonlinear buckling analysis after the non-positive definite stiffness matrix is detected. Since the modified Newton's method is employed with quasi-Newton updates in the general purpose program, tangential stiffness matrices in the two successive increments are not immediately available in general.

Nonlinear buckling is requested with the parameter BUCKLE set to +1. The operational steps are then:

1. Run SOL 106 for static analysis and create a database for restarts. It is recommended to continue the nonlinear static analysis with small steps until a negative determinant of the stiffness matrix is encountered.

2. Make a restart run as follows:
 - a. Select PARAM entries SUBID and LOOPID for restart.
 - b. Provide two small loading steps (below buckling). This may be provided by adding a new subcase, skipping the rest of the increments of the subcase in which the stiffness matrix becomes singular.
 - c. Keep "KSTEP = 1" in the NLPRAM entry for two load steps to be solved so that the stiffness matrix is updated for each solution.
 - d. Include a EIGB Bulk Data entry with a METHOD command in the Case Control Section for the eigenvalue analysis. The SINV method is recommended for the eigenvalue extraction.
 - e. Provide mode shape PLOT data if desired.
 - f. Include PARAM,BUCKLE,1.

To provide a more convenient and versatile capability for nonlinear buckling analysis, an option for a buckling analysis using a cold-start run with PARAM,BUCKLE,2 was introduced. This option allows buckling analysis in any subcase that has a METHOD command specified for an eigenvalue analysis.

Example (NLSOLIDB.DAT)

The following input file shows how a nonlinear buckling analysis can be performed in a cold start run. The complete input file nlsolidb.dat for this example is included at *installation_path/nxnr/nast/tpl*.

```

ID NLSOLIB
$
$ DESCRIPTION - ELASTIC-PLASTIC BUCKLING OF IMPERFECT SPHERICAL SHELL
$               HYDROSTATIC PRESSURE APPLIED,PERIPHERY CLAMPED
$               Material definition: ELASTIC-PLASTIC
$               Element types: CHEXA
$
$ SOLUTION     - 106
$
$ OUTPUT       - DISPLACEMENT, OLOAD
$
$ HISTORY      - Aug-5-2009 MRG, Genesis
$
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
SOL 106
CEND
TITLE=ELASTIC-PLASTIC BUCKLING OF IMPERFECT SPHERICAL SHELL
SUBTITLE=HYDROSTATIC PRESSURE APPLIED,PERIPHERY CLAMPED
SET 10 =100,200,109,209,119,219,131,231
SET 20 =10,11,15,20,25
set 30 = 1000
ECHO=UNSORT
DISP=10
oload=30
SPC=10
PARAM BUCKLE 2
SUBCASE 1
LOAD=10
NLPRAM=10
SUBCASE 2
LOAD=20
NLPRAM=20
METHOD=30
SKIPON
SUBCASE 3
LOAD=30
NLPRAM=30

```

```

skipoff
SUBCASE 4 $ SUBCASE FOR BUCKLING ANALYSIS
  LOAD=40
  NLPARM=40
  METHOD=30
SKIPOFF
BEGIN BULK
$ FOR BUCKLING ANALYSIS
EIGB 30 SINV 0.0 2. 20 2 2
$ PROPERTIES
MAT1 1 10.8+6 0.3 +MAT1
MATS1 1 PLASTIC 1.225+6 1 2 7.8+4
PSOLID 1 1 100 BUBBLE
$ PARAMETERS
$param post 0
PARAM LGDISP 1
NLPARM 10 2 AUTO 7 30 yes +NL10
+NL10 5 20 3
NLPARM 20 5 AUTO 1 30 yes +NL20
+NL20 5 20 3
NLPARM 30 8 AUTO 7 30 +NL30
+NL30 5 20 3
NLPARM 40 2 AUTO 1 yes
$ BOUNDARY CONDITION
SPC1 10 123456 131 132 231 232
GRDSET 100 3456
$ LOADING
LOAD 10 1. 2. 1 2. 2
LOAD 20 1. 3. 1 3. 2
LOAD 30 1. 3.8 1 3.8 2
LOAD 40 1. 3.5 1 3.5 2
PLOAD4 1 10 1000. 100
...
$ GEOMETRY AND CONNECTIVITY
CORD2S 100 0. 0. 0. 0. 0. 1. +C2S1
+C2S1 1. 0. 1.
CORD2S 200 0. 0. 0. -.32908 0. 0. 1. +C2S2
+C2S2 1. 0. 1.
GRID 1000 0. 0. 0. 123456
force 2 1000 1000. 0. 0. 1.
GRID 100 200 1.16315 0. 0. 0 12456
...
CHEXA 11 1 101 103 104 102 201 203 +HX11
+HX11 204 202
...
ENDDATA

```

The buckling analysis is requested in Subcases 2 and 4 in the example. Notice that the KSTEP field on each of the NLPARM entries for both Subcases 2 and 4 are set to 1 to force a stiffness matrix update at every solution step. The PARAM,BUCKLE,2 is placed above Subcase 1, but it could be placed in the Bulk Data Section or under the desired subcases. The buckling solutions in Subcases 2 and 4 will be different because the nonlinear buckling solution is an approximate solution based on extrapolation of two consecutive incremental solutions. The buckling solution tends to be more accurate when these two consecutive incremental steps are closer to the buckling point. However, you don't know where the buckling point is until the analysis is performed. For this reason, you may want to divide the load into a number of subcases and perform buckling analysis in multiple subcases.

In this example, the first subcase brings the pressure load up to 2000.0 in 2 increments. In the .f06 output, the printed load factor indicates the incremental load steps for a particular subcase. Since there are 2 increments defined on the NLPARM for subcase 1, the load factors in the .f06 output for this subcase are .5 and 1.0 corresponding to the loads 1000.0 and 2000.0, respectively.

The second subcase begins with the 2000.0 pressure load from subcase 1 and brings the pressure load up to 3000.0. Since there are 5 increments defined on the NLPARM for subcase 2, the load factors in the .f06 output for this subcase are .2, .4, .6, .8, and 1.0 corresponding to the loads 2200.0, 2400.0, 2600.0, 2800.0, and 3000.0, respectively.

At the end of the second subcase, an eigenvalue analysis is performed due to PARAM,BUCKLE,2, which results in $\alpha = 0.3240$. Since $P_n = 3000.0$ and $P_{n-1} = 2800.0$, ΔP for the second subcase is 200.

The critical buckling load is calculated as:

$$P_{cr} = P_n + \alpha \Delta P = 3000 + 0.3240 \cdot 200 = 3064.8$$

Equation 5-2.

The fourth subcase applies a pressure load of 3500 in two increments, followed by a buckling analysis.

The buckling analysis resulted in $\alpha = .06547$, and the buckling load from this second eigenvalue analysis yields

$$P_{cr} = P_n + \alpha \Delta P = 3500 + .06547 \cdot 250 = 3516.37$$

Equation 5-3.

If this buckling analysis is repeated by a restart run into subcase 4 using PARAM,BUCKLE,1, the eigenvalue analysis should give an identical solution within round-off error.

Discussion

Let us return to the example list of the input data for a review. Notice the OLOAD request for grid point 1000 that is fixed in space. However, a force of 1000 is applied to this dummy node with the same scaling in the LOAD Bulk Data as the pressure load. This OLOAD request prints the applied load for GRID 1000 at the eigenvalue solution step under the heading LOAD VECTOR. This applied load at buckling is identical to the critical buckling load calculated above. One can use this value as the critical buckling load instead of performing the above calculation.

There are two more ways to estimate the critical buckling load. One method is the arc length method that can provide solutions past the critical buckling load into the post-buckling state. Using the arc length method, the nodal displacement of a point with the most noticeable motion should be traced to detect the peak load. The applied load at this peak resembles the critical buckling load. Another method is to use the Newton's method until the solution cannot be obtained due to divergence, in which case adaptive bisection method is activated in the vicinity of the critical buckling load and stops at the limit load close to the critical buckling load. When the example problem was analyzed using an arc length method (CRIS), the solution went through the peak load of 3559.2. When the same problem was analyzed using the Newton's method with bisection procedure, the solution traced up to the maximum load of 3540.63 before diverging due to instability. When the model was remeshed with about 100 times more elements, the buckling analysis procedure predicted a critical buckling load at 3545.43. In this refined model, Newton's method with bisection converged up to 3546.88 and the arc length method peaked at 3555.72.

Concluding Remarks

The following conclusion can be drawn from this example. More refined models generally result in more accurate solutions as the eigenvalue analysis predicts a more accurate buckling point as the two solution points for extrapolation approach the actual buckling point. Comparison of the results confirms that the limit load obtained from Newton's method with adaptive bisection process closely predicts the critical buckling load. The arc length method tends to predict a slightly higher value for the buckling load.

5.5 Nonlinear Modal Analysis

You can request prestressed normal modes at the end of each subcase in SOL 106 by adding the following items to the input file.

- Add METHOD=SID in the subcase of interest. A prestressed normal mode analysis will then be performed at the end of this subcase. The METHOD command points to an EIGRL or EIGR Bulk Data entry which then selects an eigenvalue method.
- For multiple normal mode analyses in SOL 106, the METHOD command may appear in more than one subcase or above all subcases. In the latter case, a normal mode analysis is performed at the end of each subcase. The stiffness used for modal analysis corresponds to the last step of the subcase. Modal analysis cannot be performed at intermediate solution step.
- A PARAM,NMLOOP,loopid command must appear in the Case Control or Bulk Data Section to request a normal mode analysis at the end of those subcases with a METHOD command. The actual value of loopid is unimportant as long as it is a positive integer. This alleviates the cumbersome task of figuring out the exact loopid.
- Add the appropriate EIGRL or EIGR entry in the Bulk Data Section of the *Simcenter Nastran Quick Reference Guide*.

Example (TURBINE1.DAT)

The following example is a normal mode analysis of a prestressed turbine blade. (See [Figure 5-7](#)) The blade is modeled with CHEXA elements and loaded with centrifugal forces (RFORCE). The default iteration strategy is selected on the NLPARM Bulk Data entry. The software automatically updates the stiffness matrix at the end of the subcase if a METHOD Case Control command appears in the subcase.

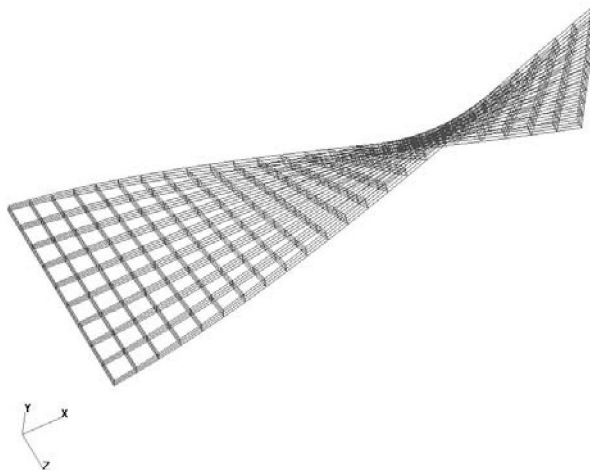


Figure 5-7. Prestressed Modes of a Turbine Blade

An excerpt of the input file is shown below.


```

SOL 106
DIAG 8
CEND
TITLE = Turbine Blade
ECHO = SORT,PUNCH
SPC = 200
LOAD = 300
NLPAR = 400
METHO = 500
DISP(PLOT) = ALL
ELST(PLOT) = ALL
NLSTRESS = NONE
BEGIN BULK
PSOLID 100 100
MAT1 100 1.+7 .3 1.
RFORCE 300 .5 1. 2
NLPARM 400 1 200 +N
+N 1.-6 1.-12 +B
+B 0
EIGRL 500 6
PARAM COUPMASS1
PARAM LGDISP 1
PARAM NMLOOP 1
PARAM POST 0
$
CHEXA 1 100 10000 10001 10101 10100 11000 11001 +M00000
+M00000 11101 11100
GRID 19324 44. 4.5 -.2 456
SPC1 200 123 10000 10100 10200 10300 11000 11100 +M00648
+M00648 11200 11300 12000 12100 12200 12300 13000 13100 +M00649
+M00649 13200 13300 14000 14100 14200 14300 15000 15100 +M00650
+M00650 15200 15300 16000 16100 16200 16300 17000 17100 +M00651
+M00651 17200 17300 18000 18100 18200 18300 19000 19100 +M00652
+M00652 19200 19300
$
$ PLUS THE REST OF THE MODEL
$
ENDDATA

```

Restarting from SOL 106 into SOL 103 or into Another Linear Solution Sequence

For the purpose of a prestressed normal modes analysis, you can restart from a SOL 106 run into another SOL 106 run to perform the prestressed normal modes calculation.

Restarts from SOL 106 into linear solution sequences are not recommended to the novice user because of several limitations. The results of the linear restart are incremental values with respect to the preload, not total values. However, some experienced users restart from SOL 106 into SOL 103 to perform prestressed modal analysis with changing boundary conditions, or restart into another linear solution sequence to perform a perturbed linear solution on a preloaded structure. In the SOL 106 cold start run, `KMETHOD='ITER'` or `'AUTO'` and `KSTEP=1` must be specified on the `NLPARM` Bulk Data entry in order to restart from the end of the subcase. With these parameters, all nonlinear information is stored on the database so that the nonlinear tangent stiffness can be recovered in the linear restart run. The following command must be included in the linear restart run:

```

PARAM,NMLOOP,loopid in the Case Control or
Bulk Data Section, where loopid is the load step id at
the end of a subcase from the SOL 106 cold start run.

```

Restarts are also allowed at intermediate steps within a subcase. The parameter `INTOUT` on the `NLPARM` entry must be set to `'YES'` or `'ALL'` in the SOL 106 cold start run to store intermediate steps on the database for restart purposes. However, the follower force stiffness may not be updated at intermediate steps. Therefore, you should restart from the end of a subcase if the effect of follower force stiffness is important.

Differential Stiffness and Follower Force Stiffness in Linear and Nonlinear Solution Sequences

For prestressed normal mode analysis, it is useful to know which stiffness contributions are included. The table below summarizes the action of the parameters LGDISP and FOLLOWWK in SOL 106 as compared to the action of FOLLOWWK in linear solution sequences. In some cases, the default settings for linear and nonlinear solution sequences are not the same. In nonlinear static analysis (SOL 106), for the default of LGDISP=-1 (geometric linear), the parameter FOLLOWWK (follower force stiffness) is ignored. For LGDISP=1, FOLLOWWK is set to YES by default. In the linear solution sequences, FOLLOWWK=YES is the default, see Case No. 3 in [Table 5-2](#).

In theory, the follower force stiffness is unsymmetric. The linear solution sequences always symmetrize the follower force stiffness. By default, SOL 106 uses the symmetrized follower force stiffness to improve performance. You can use the parameter FKSYNFAC to specify unsymmetric follower force stiffness.

See Also

- “FKSYMFAC” in the *Simcenter Nastran Quick Reference Guide*

In general, you should use the symmetrized follower force stiffness because it gives sufficient accurate results with substantial performance improvement.

In SOL 129, follower forces are included, however, the follower force stiffness is never calculated.

	Parameters		SOL 106 Cold Start or Restart		Restart from SOL 106 into SOL 103	SOL 103 Cold Start Using STATSUB
	LGDISP	FOLLOWWK	Nonlinear Element Force	Stiffness	Stiffness	Stiffness
1	1	YES	L+M+D+F	L+M+D+F	L+M+D+F	L+D'+F'
2	1	NO	L+M+D+F	L+M+D	L+M+D	L+D'
3	-1	YES	L+M	L+M	L+M+D'+F'	L+D'+F'
4	-1	NO	L+M	L+M	L+M+D'	L+D'
5	2	YES	L+M+D	L+M+D+F	L+M+D+F	L+D'+F'
6	2	NO	L+M+D	L+M+D	L+M+D	L+D'

where:

- L = linear part of element force or stiffness
- M = material nonlinear part of element force or stiffness
- D = differential or geometric nonlinear part of element force or stiffness
- D' = differential part of stiffness for small deformations
- F = follower force part of element force or stiffness
- F' = follower force part of stiffness for small deformations

All other linear solution sequences (e.g., SOL 105, 108, etc.) have the same default as SOL 103.

5.6 Nonlinear Transient Response Analysis

The input data required for SOL 129 is a combination of direct time integration control data, similar to SOL 109 (for direct linear transient with superelements), and nonlinear modeling data similar to SOL 106 (for nonlinear statics). The nonlinear properties and/or effects are defined by nonlinear material data (MATS1, MATHP, and TABLES1), gap elements (GAP) or slideline contact (BCONP, BLSEG, and BFRIC) for nonlinear interface, and PARAMETER LGDISP for geometric nonlinearity. The transient effects are produced by time-dependent loading functions (TLOADi, DAREA, LSEQ, etc.), damping (parameters, elements and material data), and mass properties. Massless degree-of-freedom should be avoided for numerical stability and the small damping is also recommended.

The unique data required for SOL 129 is supplied on the TSTEPNL Bulk Data entry. The TSTEPNL entry in itself is a combination of the TSTEP entry for direct time integration and the NLPARM entry for nonlinear iteration control. Restarts are controlled by parameters (LOOPID, STIME, SLOOPID, and SDATA) which can be specified either in the Case Control Section or Bulk Data Section. Some optional parameters (TSTATIC, NDAMP) are provided for additional control or capabilities.

Case Control

Each subcase defines a time interval starting from the last time step of the previous subcase and the interval is subdivided into small time steps. The output time is labeled by the cumulative time, including all previous subcases. There are advantages to divide the total duration of analysis into many subcases so that each subcase does not have excessive number of time steps, e.g., not to exceed 200 steps. The data blocks containing solutions are generated at the end of each subcase to store in the database for output process and restarts. As such, converged solutions are apt to be saved at many intermediate steps in case of divergence and more flexible control becomes possible with multiple subcases.

The input loading functions may be changed for each subcase or continued by repeating the same DLOAD command. However, it is recommended to use the same TLOADi Bulk Data for all the subcases in order to maintain the continuity between subcases, because TLOADi entries define the loading history as a function of cumulative time. Static loads (PLOADi, FORCEi, MOMENTi) may be associated with time-dependent functions by the LSEQ Bulk Data entry that can be selected by a Case Control command LOADSET. However, thermal loads or enforced displacements (SPCD) are not allowed in the nonlinear transient analysis. Nonlinear forces as function of displacements or velocities (NOLINi) may be selected and printed by Case Control commands NONLINEAR and NLLOAD, respectively. Each subcase may have a different time step size, time interval, and iteration control selected by the TSTEPNL request. The Case Control commands which may not be changed after the first subcase are: SPC, MPC, DMIG, and TF.

Output requests for STRESS, SDISPLACEMENT, SVELOCITY, SACCELERATION, and NONLINEAR may be specified within a subcase. DISPLACEMENT, VELOCITY, ACCELERATION, OLOAD, FORCE, and SPCFORCE must be specified above the subcase level. However, FORCE output and GPSTRESS output are not available for nonlinear elements.

Initial conditions (displacement or velocity) can be specified by the Bulk Data TIC selectable by the Case Control command IC. If initial conditions were given, all of the nonlinear element forces and stresses must be computed to satisfy equilibrium with the prescribed initial conditions. On the other hand, initial conditions could be a part of the nonlinear analysis by applying static analysis for the

preload using PARAM, TSTATIC in the first subcase. Then the transient analysis can be performed in the ensuing subcases. Associated with the adaptive time stepping method, the PARAMeter NDAMP is used to control the stability in the ADAPT method. The parameter NDAMP represents the numerical damping (recommended value for usual case is 0.01) which often required to improve the stability and convergence in the contact problems.

All the superelement model generation options and matrix reduction options are allowed for the linear portion of the structure. The component mode synthesis and Guyan reduction may be performed for upstream superelements. The residual superelement may contain scalar degree-of-freedom representing linear modal formulations.

Limitations of Nonlinear Transient Response Analysis

1. No constraint set changes are allowed between subcases or in restarts.
2. No initial preloads such as closed gaps or terminal loads are allowed in a cold start. See PARAM,LOOPID under “Restarts” in this section.
3. Overall structure damping (PARAM,G) is not provided in nonlinear elements. Use the damping field in the material (MATi) data entry and the PARAMeter W4 to supply damping based on the linear properties.
4. Element forces are not computed for nonlinear elements. Output requests will be ignored.
5. When multiple subcases are specified, it is recommended that a single time function (see the TLOADi Bulk Data entry description) be assigned to an applied load throughout the entire time span of the solution. This time function can be used for all subcases. If different time functions are to be used between subcases, ensure that the time functions are continuous between subcases.
6. Element stresses for nonlinear elements are computed and printed during the nonlinear iteration phase only. They are not merged with those of the linear elements.

Direct Time Integration

If the effects of inertia and damping forces are to be included in the nonlinear analysis, the nonlinear transient response is analyzed by a step-by-step integration of an equation of motion, i.e.,

$$M\ddot{u} + C\dot{u} + F(u) = P(t)$$

Equation 5-4.

By direct numerical integration, the equilibrium of Eq. 5-4 is satisfied at discrete time steps with an interval of Δt . The equilibrium is ensured by iterations until the solution converges to the preset error tolerance.

Procedures, such as the quasi-Newton update and line search process, provide users with additional options to adjust incremental and iterative processes in search of more efficient and effective strategy for obtaining solutions. A self-adaptive time stepping method that adjusts time step automatically during analysis is available.

Newmark's direct time integration method is implemented using the two-point recurrence (or one-step) formula as a foundation of this self-adaptive time stepping algorithm. The optimal time step

size, which is required for accuracy and efficiency, changes continuously in the transient dynamic environment. The primary concept of automatic time step adjustment is that the proper size of the time step can be predicted based on the dominant frequency in the incremental deformation pattern at the previous time step. This concept presents a deficiency of time lag involved in the prediction process. Furthermore, changes in nonlinearity cannot be predicted from the deformation pattern at the previous time step. This deficiency is overcome by the bisection process, which is activated when any difficulties arise in terms of convergence during the iteration.

Frequent decomposition of the dynamic stiffness matrix is inevitable in adaptive time stepping using the implicit integration method. Efforts have been made to minimize the thrashing in the algorithm (e.g., too frequent or repetitive adjustment of the time step size) and yet to provide a solution reliably to any type of problem. The iteration process for the equilibrium employs expeditious methods such as BFGS updates and line searches as well as an adaptive stiffness matrix update strategy. The present self-adaptive time stepping algorithm for the nonlinear transient analysis is proving to be a robust and practical method, aimed at efficiency, effectiveness, and user friendliness.

The transient response analysis capability is provided in the self-contained solution sequence SOL 129. SOL 129 is a structured DMAP. Although this solution sequence is developed for nonlinear transient response analysis, linear transient response analysis can be performed in this solution sequence in order to take advantage of the automatic time stepping and restart capabilities. The core parts of SOL 129 are the nonlinear modules (NLTRD2 and NLTRD) which perform the incremental and iterative processes for implicit direct time integration.

Implicit Integration Control: TSTEPNL Data Entry

The input fields of the TSTEPNL Bulk Data entry specify the time step size, the number of steps, and the output interval as well as the nonlinear iteration options. The TSTEPNL Bulk Data is selectable by the Case Control command TSTEPNL. Although the same TSTEPNL Bulk Data may be selected by more than one subcase, it is recommended to select different TSTEPNL entry for each subcase in preparation for changes in the restarts.

The choice of time step size is determined primarily by the frequency content of the input load and the highest frequency mode-of-interest. A general guideline is that seven or more steps per cycle be provided for reasonable accuracy. Modes with shorter periods (higher frequency) will be attenuated by the numerical process. Highly nonlinear problems may require smaller step size. However, the adaptive time stepping capability is the vital part of 129 and is recommended for any problem (linear or nonlinear). Nevertheless, you should use these principles to estimate the initial time step size.

A caution is necessary in using the automatic time step adjustment if the forcing function is a short duration pulse. Since the automatic time step adjustment algorithm does not consider the loading history, short pulses could be skipped if the time step is automatically adjusted to a larger value than the pulse duration. It is advised that a drastic change in the time step size between subcases be avoided. The drastic change, e.g., a ratio exceeding 1000, could cause a loss of accuracy upon subcase switch. In such case, an intermediate subcase should be provided for a transition period of short interval to reduce the ratio.

The parameters for controlling the accuracy and stability of the incremental and iterative process can be specified in the TSTEPNL Bulk Data entry. The controls are applicable to the automatic time step adjustment and bisection process in addition to stiffness matrix updates, BFGS updates and line searches similar to those on the NLPARM Bulk Data entry. Since default values are resulted from numerous test runs, the analysis should be started with the default setting and changed if necessary. The TSTEPNL data format is shown below with default values:

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO		KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB		

Example With Defaults:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	250	100	.01	1		2	10	PW	
	1.E-2	1.E-3	1.E-6	2	10	2	0.2		
	5	5		0.75	16.0	0.1	20.		

The TSTEPNL Bulk Data entry is selected using ID by the Case Control command TSTEPNL. Each subcase (residual superelement solutions only) requires a TSTEPNL entry. Multiple subcases are assumed to occur sequentially in time. Therefore, the initial conditions of each subcase are defined by the end conditions of the previous subcase.

The NDT field specifies the number of time steps with DT as the size of each time step. The total duration for the subcase can be assessed by multiplying NDT with DT (i.e., $NDT \cdot DT$). The time increment (Δ, t) remains constant during the analysis in AUTO and TSTEP options, and is equal to DT. However, the time increment (Δ, t) changes during the analysis in the ADAPT option and the actual number of time steps will not be equal to NDT. In ADAPT option DT is used as an initial value for Δ, t .

The NO field specifies the time step interval for output, i.e., every NOth step solution is saved for output. The data will be output at steps 0, NO, ..., etc., and the last converged step for printing and plotting purposes. The Case Control command OTIME may also be used to control the output points.

The program automatically adjusts the incremental time and uses the bisection algorithm in case of divergence. During the bisection process the stiffness is updated at every KSTEPth successful bisection. The stiffness matrix is always updated for a new subcase or restart, irrespective of the option selected.

The number of iterations for a time step is limited to MAXITER. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process, i.e., either a bisection or stiffness matrix update takes place based on the value of MAXBIS. The sign of MAXITER provides a control over the ultimate recourse (reiteration) in case of failure in convergence or bisection. If the MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best attainable solution and continues the analysis on second divergence.

The convergence test is controlled by convergence test flags (U for displacement error test, P for load equilibrium error test, W for work error test) and the error tolerances (EPSU, EPSP and EPSW) which define the convergence criteria. All requested criteria (combination of U, P, and/or W) are satisfied upon convergence. It should be noted that at least two iterations are necessary to check the displacement convergence criterion.

The MAXDIV field provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions (NDIV) is incremented by 1 or 2. The solution is assumed to be divergent when NDIV reaches MAXDIV during the iteration. If the bisection option is used, the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new

stiffness based on the converged state at the beginning of the time step. If NDIV reaches MAXDIV twice within the same time step, the analysis is terminated with a fatal message.

The BEGS quasi-Newton updates and the line search process work in the way as in static analysis except for the default setting. The MAXQN field defines the maximum number of quasi-Newton vectors to be saved on the database and the MAXLS defines the number of line searches allowed per iteration. Nonzero values of MAXQN and MAXLS activate the quasi-Newton update and the line search process, respectively.

The FSTRESS field defines a fraction of the effective stress ($\bar{\sigma}$) which is used to limit the subincrement size in the material routine. The number of subincrements in the material routines is determined such that the subincrement size is approximately $FSTRESS \cdot \bar{\sigma}$ (equivalent stress). FSTRESS is also used to establish a tolerance for error correction in the elasto-plastic material, i.e.,

$$Error \text{ in yield function} < FSTRESS \cdot \bar{\sigma}$$

Equation 5-5.

If the limit is exceeded at the converging state, the program will exit with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface.

MAXBIS is the maximum number of bisections allowed for each time step ($-9 \leq MAXBIS \leq 9$). The bisection process is activated when divergence occurs and $MAXBIS \neq 0$. The number of bisections for a time increment is limited to $|MAXBIS|$. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in $|MAXBIS|$ bisection, the analysis is terminated.

The parameter ADJUST lets you control the automatic time stepping in the ADAPT option. A value of zero for ADJUST turns off the automatic adjustment completely. If ADJUST is positive, the time increment is continually adjusted for the first few steps until a good value of Δt is obtained. After this initial adjustment, the time increment is adjusted every ADJUST time steps only. A value of ADJUST an order greater than NDT will turn off adjustment after the initial adjustment. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is your responsibility to ensure that the loading history is properly traced with ADJUST option. The ADJUST option should be suppressed for the duration of short pulse loading. If unsure, start with DT which is much smaller than the pulse duration in order to properly represent the loading pattern.

MSTEP defines the desired number of time steps to obtain the dominant period response accurately ($10 \leq \text{Integer} \leq 200$). RB defines bounds for maintaining the same time step for the stepping function in the automatic time step adjustment method ($0.1 \leq \text{Real} \leq 1.0$). Parameters MSTEP and RB are used to adjust the time increment during the analysis in the ADAPT option. The adjustment is based on the number of time steps desired to capture the dominant frequency, ω_n , in the incremental deformation. The dominant frequency is estimated by the software at every converged time step using the displacement increment and the internal forces by

$$\omega_n^2 = \frac{\{\Delta U_n\}^T K \{\Delta U_n\}}{\{\Delta U_n\}^T M \{\Delta U_n\}} \approx \frac{\{\Delta U_n\}^T \{F_n - F_{n-1}\}}{\{\Delta U_n\}^T M \{\Delta U_n\}}$$

where $\{\Delta U_n\} = \{U_n - U_{n-1}\}$ is the change in the displacements, and $\{F_n - F_{n-1}\}$ is the change in the internal forces.

Then the time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r) \Delta t_n$$

Equation 5-6.

where:

$$r = \frac{1}{\text{MSTEP}} \left(\frac{2\pi}{w_n} \right) \left(\frac{1}{\Delta t_n} \right)$$

with:

$$\begin{aligned} f &= 0.25 \text{ for } r < 0.5 \cdot RB \\ f &= 0.5 \text{ for } 0.5 \cdot RB < r < RB \\ f &= 1.0 \text{ for } RB < r < 2 \\ f &= 2.0 \text{ for } 2. \leq r < 3./RB \\ f &= 4.0 \text{ for } r > 3./RB \end{aligned}$$

The recommended value of MSTEP for nearly linear problems is 20. A larger value (e.g., 40) is required for highly nonlinear problems. In the default option, the program automatically computes the value of MSTEP based on the changes in the stiffness.

The MAXR field defines the maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment ($1.0 \leq \text{Real} \leq 32.0$). MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\text{MIN} \left(\frac{DT}{2^{\text{MAXBIS}}}, \frac{DT}{\text{MAXR}} \right) \leq \Delta T \leq \text{MAXR} \cdot DT$$

Equation 5-7.

The UTOL field defines the tolerance on displacement increment below which there is no time step adjustment ($0.001 < \text{Real} \leq 1.0$). UTOL is used to filter undesirable time step adjustment, i.e., no time step adjustment is performed if

$$\frac{\|\dot{U}_n\|}{\|\dot{U}_n\|_{\max}} < \text{UTOL}$$

Equation 5-8.

The RTOLB field defines the maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection ($\text{Real} > 2.0$). The bisection is activated if the incremental rotation for any

degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$ or $\Delta\theta_z$) exceeds the value specified for RTOLB. This bisection strategy based on the incremental rotation is controlled by MAXBIS field.

Iteration Related Output Data

During the incremental and iterative computation, the process information consisting of iteration data is printed at the end of each iteration or time step. They are printed under the following heading:

TIME	Cumulative time for the duration of the analysis.
ITER	Iteration count for each time step.
DISP	Relative error in terms of displacements defined as

$$E_U^i = \lambda^i \frac{\|u^i - u^{i-1}\|}{(1 - \lambda^i)u_{max}} \quad \text{for ADAPT method}$$

Equation 5-9.

$$E_U^i = \frac{\|u^i - u^{i-1}\|}{(1 - \lambda^i)u_{max}} \quad \text{for AUTO or TSTEP}$$

Equation 5-10.

where $u_{max} = \max(\|u_1\|, \|u_2\|, \dots, \|u_n\|)$

LOAD	Relative error in terms of load vectors defined as
------	--

$$E_p^i = \frac{\|R^i\|}{\max(\|F_n\|, \|P_n\|)} \quad \text{for ADAPT method}$$

Equation 5-11.

$$E_p^i = \frac{\|R^i\|}{\max(\|\bar{F}\|, \|\bar{P}\|)} \quad \text{for AUTO or TSTEP}$$

Equation 5-12.

where \bar{F} and \bar{P} are three-point average values for internal and external forces, respectively.

WORK Relative error in terms of work defined as

$$E_w^i = \frac{\{u^i - u^{i-1}\}^T R^i}{\max(\{u_n\}^T \{F_n\}, \{u_n\}^T \{P_n\})}$$

Equation 5-13.

LAMBDA(I) Rate of Convergence in iteration where

$$\lambda^i = \frac{E_p^i}{E_p^{i-1}}$$

Equation 5-14.

LAMBDA(T) Ratio of the load error for two consecutive time steps computed only for AUTO or TSTEP method

$$\lambda_n^n = \frac{E_p^n}{E_p^{n-1}}$$

Equation 5-15.

LAMBDA-BAR Average of LAMBDA(T) over the last 3 steps, computed only for AUTO or TSTEP method.

DLMAG Absolute norm of the load vector, $\|R\|$. The absolute convergence is defined using DLMAG by $\|R\| < 10^{-12}$.

FACTOR Final value of the line search parameter.

E-FIRST Divergence rate, error before line search.

E-FINAL Error at the end of line search.

NQNV Number of quasi-Newton vectors appended.

NLS Number of line searches performed during the iteration.

ITR DIV Number of occurrences of divergence detected during the adaptive iteration by the iteration module NLTRD2.

MAT DIV Number of occurrences of bisection conditions in the material routine (excessive stress increment) or in the rotation angle (excessive rotation) during the iteration using the ADAPT method.

NO. BIS Number of bisections executed for the current time interval.

ADJUST Ratio of time step adjustment relative to DT within a subcase.

Restarts

Since SOL 106 and SOL 129 share the same database storage for nonlinear tables and matrices, the restart system for transient analysis can use either a previous static or transient nonlinear analysis as its initial conditions. A restart from SOL 129 into SOL 106 is not available.

Restarting from SOL 106 into SOL 129

The options for a restart from SOL 106 into SOL 129 are static to static and static to dynamic analysis. For a restart from a previous static analysis, only the first subcase is affected. Simply provide a database created in SOL 106 (or 129) and specify the parameter PARAM,SLOOPID,N where N is the printed value of LOOPID for the desired static solution. The initial transient load should be identical to static loads at the restart state. Constraint sets, direct input matrices, mass, and damping may be changed.

Restarting within SOL 129

Restarting within SOL 129 allows static to static, static to dynamic, and dynamic to dynamic analysis. Restarts from a previous nonlinear transient execution are available for a number of cases. If the same model is to be re-executed, only the residual superelement needs to be reassembled. If the final results from the previous transient run are to be used as the initial conditions at $t = 0$, add N dummy SUBCASE commands to start the residual Case Control execution and STIME = 0.

The normal restart for a transient run is to be continued from the last step of a previous subcase with different loads and/or TSTEPNL data. For the normal restart provide the following parameters:

```
LOOPID = N   : Start from the Nth subcase
STIME = t    : Start from time t
```

Note that constraint sets should not be changed to avoid incompatible matrix sizes. The values of LOOPID and STIME, which are printed with the iteration information for each subcase, can be directly read from the printout of the previous run. If the adaptive time stepping algorithm is not activated, the value of t may also be calculated by the following equation:

$$t = \left(\sum_{i=1}^N NDT_i \right) \cdot DT_i$$

Equation 5-16.

where NDT_i and DT_i are the number of time steps and the time increment of the i -th subcase, respectively.

If a SOL 129 run is terminated abnormally in the middle of a subcase, it may or may not be restartable depending upon the cause of the abnormal exit. If the job is stopped due to a diverging solution, it can be restarted either from the end of a previous subcase or from the last saved solution step. The restart procedure for the former is identical to that for the normal restart as described in the preceding paragraph. The latter case also requires parameters LOOPID and STIME, however, the input value for STIME differs depending on the value of METHOD specified on the TSTEPNL entry. If METHOD = AUTO or TSTEP (the NLTRD module), STIME is the time corresponding to the last output step which may be calculated based on the output skip factor (i.e., the NO on the TSTEPNL entry). If METHOD = ADAPT (NLTRD2 module) the last converged solution is treated as an output step and is

always saved for the restart so that STIME can be the time of the last converged step. The values of STIME and LOOPID can also be found in the printout, if the ADAPT method is used. Once STIME and LOOPID are known, determine the number of remaining time steps in the subcase and create a new TSTEPNL entry for the remaining time. Insert a new subcase which references the new TSTEPNL entry prior to the remaining subcases in the Case Control Section.

A solution may be terminated in the middle of a subcase due to insufficient CPU time: (1) the CPU time specified in the Executive Control Section is insufficient so the run is forced to exit by the program, or (2) the CPU time exceeds the limit specified in the computer system which leads to a sudden job abortion by the system. In the first case, Simcenter Nastran is able to detect the specified CPU time in the Executive Control Section and automatically activate the wrapping-up procedure before time expiration. When completed, the solution can be restarted from the termination point as in the solution diverging case. In the second case the solution can only be restarted from the end of a subcase.

Restarts may also be performed solely for data recovery by providing the following parameters:

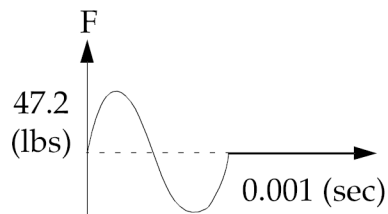
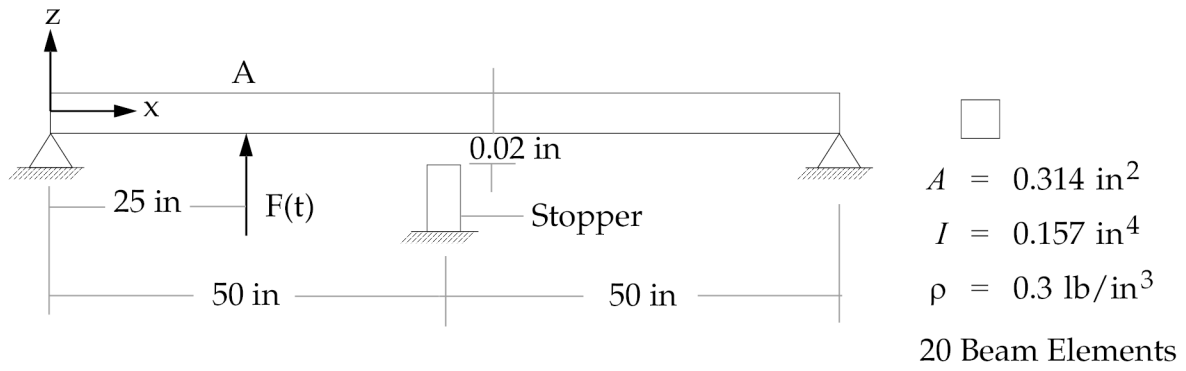
SDATA = -1 : Recover data without running the solution module
 LOOPID = N : from the 1st through the N-th subcases.

Note that solution sets DISP, VELO, ACCE, OLOAD, SPCF (printout and plotting) and NLLOAD (plotting only) are recoverable while STRESS, SDISP, SVELO and SACCE sets are not.

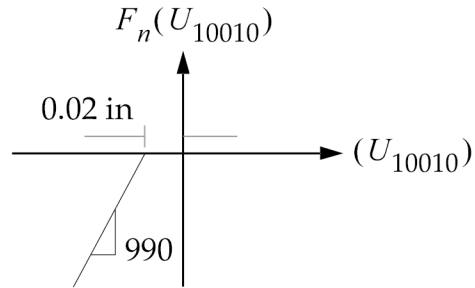
Example

A simply supported elastic beam is subjected to a single cycle of a sinusoidal forcing function at the quarter span as illustrated in [Figure 5-8](#). The beam is modeled using 20 linear CBEAM elements. A stopper (snubber) is present underneath the center of the beam with a clearance of 0.02 inches. This gap is simulated by a displacement dependent force which is active only when the beam is in contact with the snubber. The problem is linear except for the effect of the stopper. The input data for this problem is given in [Listing 5-2](#).

One subcase with 100 time steps and an initial time step size of 0.1 msec was analyzed in the cold start run. Since the adaptive time stepping algorithm was activated, the job ran to completion at 9.7 msec instead of the specified time span, $100 \times 0.1 = 10$ msec. A new subcase with 600 time steps was added in the first restart run to extend the time span for another 30 msec. The adaptive time stepping method was turned off and the execution time was specified to be 0.8 min. As expected, the run was aborted at 36 msec due to insufficient time. In the second restart run, the execution time was changed to 5.0 min. and another subcase with 164 time steps and a time increment of 0.1 msec was added to extend the analysis time span up to 40 msec. The third restart was simply to recover the DISP, VELO, ACCE, OLOAD, NLLOAD and SPC output. The response at the loading point of the beam is shown in [Figure 5-9](#).



Forcing Function



NOLIN1 Representing GAP

Figure 5-8. Simply Supported Beam With a Restrained Motion

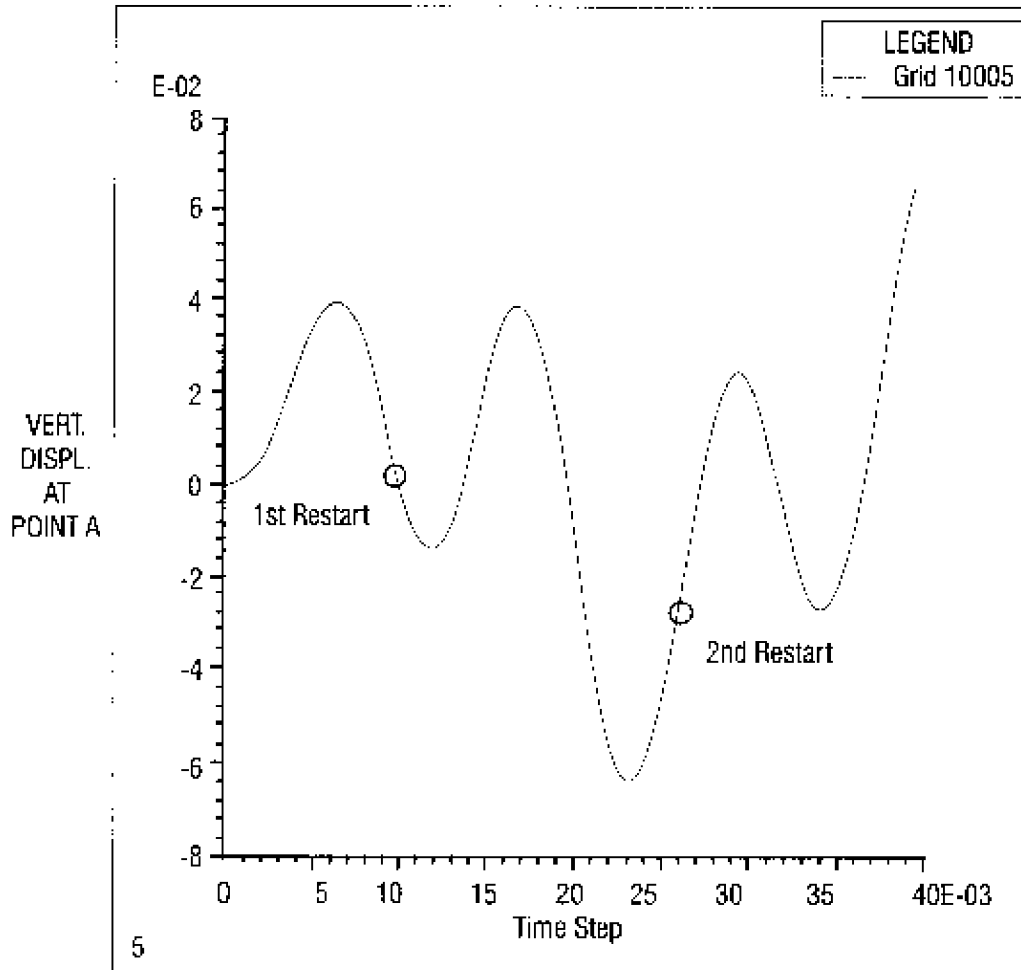


Figure 5-9. Simply Supported Beam: Loading Point Response

```

$RESTART $ ***added for the restart runs
ID TNOLIN, V66 $ SSH 7/10/91
TIME $ $
SOL 99 $ NONLINEAR TRANSIENT ANALYSIS
DIAG 8,50 $ PRINT MATRIX TRAILERS AND ITERATIONS
CEND
TITLE = DYNAMICS HANDBOOK DEMONSTRATION PROBLEM
SUBTITLE = DIRECT TRANSIENT RESPONSE, NONLINEAR FORCE
LABEL = NOLIN IN SOL 99
SEALL = ALL
ECHO = UNSORT
  SET 1 = 10005
  SET 2 = 10010
  SET 3 = 10000
  SPC = 1002
$-----
$PARAM,LOOPID,1 *** used for the 1st restart
$PARAM,STIME,0.0093
$-----
$PARAM,LOOPID,2 *** used for the 2nd restart
$PARAM,STIME,0.0236
$-----
$ DISPL=2
$ VELO=2 *** used for the 3rd restart
$ ACCEL=2
$ OLOAD=1
$ NLLOAD=2

```

```

$ SPCF=3
$PARAM,LOOPID,3
$PARAM,SDATA,-1
$-----
SUBCASE 1
  DLOAD = 30
  TSTEPNL = 10
  NONLINEAR = 13 $ SELECT NONLINEAR FORCE
SUBCASE 2
  DLOAD = 30
  TSTEPNL = 20
  NONLINEAR = 13 $ SELECT NONLINEAR FORCE
SUBCASE 3
  DLOAD = 30
  TSTEPNL = 30
  NONLINEAR = 13 $ SELECT NONLINEAR FORCE
BEGIN BULK
$ MODELING INFORMATION FOR BEAM ONLY
CBAR 101 100 10000 10001 0. 0. 1.
CBAR 102 100 10001 10002 0. 0. 1.
CBAR 103 100 10002 10003 0. 0. 1.
CBAR 104 100 10003 10004 0. 0. 1.
CBAR 105 100 10004 10005 0. 0. 1.
CBAR 106 100 10005 10006 0. 0. 1.
CBAR 107 100 10006 10007 0. 0. 1.
CBAR 108 100 10007 10008 0. 0. 1.
CBAR 109 100 10008 10009 0. 0. 1.
CBAR 110 100 10009 10010 0. 0. 1.
CBAR 111 100 10010 10011 0. 0. 1.
CBAR 112 100 10011 10012 0. 0. 1.
CBAR 113 100 10012 10013 0. 0. 1.
CBAR 114 100 10013 10014 0. 0. 1.
CBAR 115 100 10014 10015 0. 0. 1.
CBAR 116 100 10015 10016 0. 0. 1.
CBAR 117 100 10016 10017 0. 0. 1.
CBAR 118 100 10017 10018 0. 0. 1.
CBAR 119 100 10018 10019 0. 0. 1.
CBAR 120 100 10019 10020 0. 0. 1.
CONM2 12 10010 .1
GRID 10 50. -1.
GRID 10000 0. 0. 0. 1246
GRID 10001 5. 0. 0. 1246
GRID 10002 10. 0. 0. 1246
GRID 10003 15. 0. 0. 1246
GRID 10004 20. 0. 0. 1246
GRID 10005 25. 0. 0. 1246
GRID 10006 30. 0. 0. 1246
GRID 10007 35. 0. 0. 1246
GRID 10008 40. 0. 0. 1246
GRID 10009 45. 0. 0. 1246
GRID 10010 50. 0. 0. 1246
GRID 10011 55. 0. 0. 1246
GRID 10012 60. 0. 0. 1246
GRID 10013 65. 0. 0. 1246
GRID 10014 70. 0. 0. 1246
GRID 10015 75. 0. 0. 1246
GRID 10016 80. 0. 0. 1246
GRID 10017 85. 0. 0. 1246
GRID 10018 90. 0. 0. 1246
GRID 10019 95. 0. 0. 1246
GRID 10020 100. 0. 0. 1246
MAT1 1000 3.+7 .3 .3
PARAM GRDPNT 10010
PARAM WTMASS .002588
PBAR 100 1000 .31416 .15708 +PB1
+PB1 1. 0.
SPC 1002 10 123456
SPC 1002 10020 3 10000 3
$
$ MODELING INFORMATION FOR CENTER SPRING
CROD 10 10 10 10010
MAT1 10 10. 0.
PROD 10 10 1.
MATS1 10 PLASTIC 0. 1 1 3.+8
$ LOADING AND SOLUTION INFORMATION
TLOAD2 30 33 0. .011451 87.33 -90.
DAREA 33 10005 3 47.2

```

```

TSTEPNL 10 100 .0001 1 ADAPT
TSTEPNL 20 600 .00005 1 ADAPT +T521
+T521 +T522
+T522 0
TSTEPNL 30 164 .00010 1 ADAPT +T531
+T531 +T532
+T532 0
$
$ MODELING INFORMATION FOR NONLINEAR SPRING
NOLIN1 13 10010 3 1. 10010 3 13
TABLED1 13 +TAB
+TAB -2.5-2 4.95 -2.0-2 0. 0. 0. ENDT
$
ENDDATA

```

Listing 5-2. Input Data Listing for Simply Supported Beam

5.7 Nonlinear Analysis With Superelements

In nonlinear analysis with superelements, the nonlinear model must be in the residual model (Superelement = 0). All basic superelement rules apply with the exception that a nonlinear element can not have any of its degrees-of-freedom in the “o-set”.

The unique feature pertaining to load application for superelements are listed in [“Performing a 3-D Slide Line Contact Analysis”](#)

5.8 Grid Point Force Balance and Element Strain Energy in Nonlinear Static Analysis

In nonlinear analysis, the strain energy, E , for an element is defined by integrating the specific energy rate, the inner product of strain rate and stress, over element volume and time

$$E = \int_0^t \int_V \dot{\varepsilon}^T \sigma dV d\tau$$

Equation 5-17.

where:

- σ = stress tensor
- $\dot{\varepsilon}$ = strain rate
- V = element volume
- t = actual time in the load history

The integration over time leads to the following recursive formula using the trapezoidal rule

$$E_{n+1} = E_n + \Delta E_{n+1} = E_n + \frac{1}{2} \int \Delta \varepsilon_{n+1}^T (\sigma_n + \sigma_{n+1}) dV$$

Equation 5-18.

where:

$n + 1$	=	current load step
n	=	previous load step
ΔE	=	strain energy increment
Δ	=	strain-increment

The steps n and $n + 1$ are converged solution steps.

By integrating [Eq. 5-18](#) over the element volume, we get

$$E_{n+1} = E_n + \frac{1}{2} \Delta u_{n+1}^T (f_n + f_{n+1})$$

Equation 5-19.

where:

f	=	internal element forces
Δu	=	displacement increment from n to $n + 1$

For computational convenience, Simcenter Nastran uses [Eq. 5-19](#) to calculate the element strain energy. The internal element forces are readily available in every step because they are needed for the force equilibrium. Note that temperature effects are included in the internal element forces.

When loads from temperature differences or element deformation are present, the default definition of element strain energy for linear elements differ from the definition for nonlinear elements. For linear elements, the element strain energy is defined as

$$E = \frac{1}{2} u^T K_e u - u^T P_{et}$$

Equation 5-20.

where P_{et} is the element load vector for temperature loads and element deformation. [Eq. 5-20](#) assumes that the temperatures are constant within a subcase. For nonlinear elements, the definition of [Eq. 5-17](#) is used. In the case of linear material and geometry, [Eq. 5-17](#) becomes

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et}$$

Equation 5-21.

Eq. 5-21 assumes that the temperature varies linearly within a subcase. You may request the definition of Eq. 5-21 to be applied to linear elements by adding `PARAM,XFLAG,2` to the input file.

User Input

The output of grid point force balance and strain energy in nonlinear analysis is requested with the existing `GPFORCE` and `ESE` Case Control commands, respectively. The Case Control commands remain unchanged.

Printed Output

The output is the same as in linear analysis. An example appears below.

Example

The following Nastran input deck represents a simplified model of a mechanical clutch that consists of springs, beams, rigid elements, and gap elements. A geometric nonlinear analysis is performed. Both `GPFORCE` and `ESE` output requests are applied above all subcases. This is a good example to show the grid point force balance with both linear and nonlinear elements, that includes:

- applied loads
- element forces
- SPC forces
- MPC forces.

The model is shown in Figure 5-10. For clarity, only the elements are displayed.

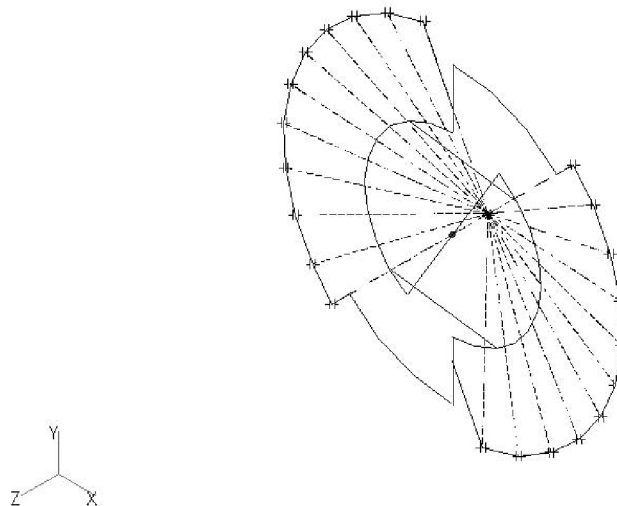


Figure 5-10.

The dashed lines represent rigid elements, the rest are gap and beam elements. Spring elements are located at the intersection point of all dashed lines and are not visible in the figure because their connection points are coincident.

Grid Point Force Output

```

1  GPF05B - FORD ELECTRO-MECHANICAL CLUTCH          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 248
   DISPLACEMENT IN MM., FORCE IN NEWTON, STRESS IN MPA
0
   LOAD STEP = 2.00000E+00                                SUBCASE 2
                                GRID POINT FORCE BALANCE
POINT-ID  ELEMENT-ID  SOURCE      T1      T2      T3      R1      R2      R3
1         1          APP-LOAD    .0      .0      .0      .0      .0      -2.400000E+02
1         1          F-OF-SPC    .0      .0      .0      .0      .0      2.400000E+02
1         1          BEAM       .0      .0      .0      .0      .0      .0
1         1          BEAM       .0      .0      .0      .0      .0      .0
1         1          *TOTALS*   .0      .0      .0      .0      .0      .0
0        100         F-OF-SPC    -5.366200E-25  -4.514879E-24  .0      4.535559E-23  -6.400176E-24  .0
100       100         F-OF-MPC    5.366200E-25  4.514879E-24  .0      -4.535559E-23  6.400176E-24  1.292178E-24
100       100         ELAS2      .0      .0      .0      .0      .0      1.079983E-24
100       100         *TOTALS*   .0      .0      .0      .0      .0      2.363161E-24
0        101         F-OF-SPC    .0      .0      -3.078859E-28  -3.540490E-26  2.535672E-24  .0
101       101         BEAM       -3.701448E-08  2.907432E-07  .0      .0      .0      -5.474456E-09
101       101         BEAM       -3.888788E-07  -1.144153E-07  3.078859E-28  3.540490E-26  -2.535672E-24  -1.777338E-23
101       101         GAP       1.270000E-10  1.438901E-27  .0      .0      .0      .0
101       101         *TOTALS*   -4.257663E-07  1.763280E-07  .0      .0      .0      -5.474456E-09
0        102         BEAM       -2.715682E-08  -2.918291E-07  .0      .0      .0      -2.549396E-09
102       102         BEAM       -1.637746E-08  1.273060E-07  .0      .0      .0      -1.374512E-25
102       102         GAP       1.270000E-10  -9.144001E-27  .0      .0      .0      .0
    
```

Element Strain Energy Output

```

1  GPF05B - FORD ELECTRO-MECHANICAL CLUTCH          JULY 23, 2003 NX NASTRAN 07/23/03  PAGE 231
   DISPLACEMENT IN MM., FORCE IN NEWTON, STRESS IN MPA
0
   LOAD STEP = 1.00000E+00                                SUBCASE 1
                                ELEMENT STRAIN ENERGIES
ELEMENT-TYPE = BEAM
SUBCASE 1
* TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM = 4.422530E+02
* TOTAL ENERGY OF ALL ELEMENTS IN SET   -1 = 4.422530E+02
0
ELEMENT-ID  STRAIN-ENERGY  PERCENT OF TOTAL  STRAIN-ENERGY-DENSITY
101         1.211380E+01  2.6928  6.680606E-02
102         1.049874E+00  .2334  4.923244E-03
103         1.323984E-01  .0294  6.208644E-04
104         1.323428E-01  .0294  6.206040E-04
105         1.274425E-01  .0306  6.445181E-04
106         1.407000E-01  .0313  6.597969E-04
107         1.478413E-01  .0329  6.932822E-04
108         1.483820E-01  .0330  6.958218E-04
109         1.032130E-01  .0229  1.936014E-04
110         3.163312E-01  .0703  7.020313E-04
111         4.605607E-01  .1024  1.927717E-03
112         7.067609E-01  .1571  2.958209E-03
113         4.186314E-01  .0931  1.752469E-03
121         3.068841E+00  6.822  2.905759E-02
    
```

Remarks

- Linear and nonlinear elements can be mixed. The output for grid point force and element strain energy in nonlinear analysis follow the same format as in linear analysis.
- The reference system for the grid point force output is the grid point global coordinate system. You can't specify another output coordinate system. In nonlinear, the grid point forces are not aligned with element edges. Therefore, PARAM,NOELOF and PARAM,NOELOP are ignored in nonlinear analysis.
- In nonlinear analysis, the element strain energy must be calculated for each intermediate load step even if the output is requested only in the last load step. To save computations, the element strain energy is only calculated upon user request. The Case Control commands, GPFORCE or ESE must be present to activate grid point force output or element strain energy calculations and output. If you want to save the grid point forces and element strain energy on the database and want an output later in an eventual restart run, you must specify the Case Control commands, GPFORCE(PLOT) or ESE(PLOT) in the cold start run.
- By default, if GPFORCE or ESE is present, then the output of grid point forces or element strain energy is only available at the end of each subcase. For output of intermediate steps, the parameter INTOUT on the NLPARM Bulk Data entry must be set to YES or ALL.
- For a model with superelements, the grid point force balance is not in equilibrium along the boundary of the residual and upstream superelements. This is because the applied load in the

residual does not include the load contributed from upstream superelements when the grid point force balance is computed. The same situation also occurs in each upstream superelement whose applied load does not include the contribution from other superelements and the residual. When the total of the grid force balances of all superelements is added together, equilibrium is reached.

For example, grid point 123 is located at the boundary between the residual (superelement ID = 0) and an upstream superelement (ID = 10) and is connected to two QUAD4 elements (ID 22 and 21). The output of the grid point force balance is shown below.

```

1 GPF006A - NONLINEAR GPFORCE TEST PROBLEM              JULY 23, 2003 NX NASTRAN 07/23/03 PAGE 26
04 ELEMENTS + MULTIPLE S.E.'S                          SUPERELEMENT 0
0 NONLINEARITY ANALYSIS - AUTO/HLPCI + INTOUT=YES      SUBCASE 1
LOAD STEP = 1.00000E+00

      GRID POINT FORCE BALANCE

POINT-ID  ELEMENT-ID  SOURCE      T1      T2      T3      R1      R2      R3
0 123      APP-LOAD    -3.287216E-01  1.006007E-02 -5.592173E-03 .0      .0      .0
   123      P-OF-SPC     .0      .0      5.127304E+00 -2.28779E-04 -1.193804E-02 .0
   123      21 QUAD4      -6.766498E-01  8.791827E+00 -2.111628E+00  8.897618E-04  1.142736E-02 -3.166363E-02
   123      *TOTALS*    -1.005271E+00  8.801888E+00  3.010083E+00  6.609823E-04 -5.106755E-04 -3.166363E-02

1 GPF006A - NONLINEAR GPFORCE TEST PROBLEM              JULY 23, 2003 NX NASTRAN 07/23/03 PAGE 33
04 ELEMENTS + MULTIPLE S.E.'S                          SUPERELEMENT 10
0 NONLINEARITY ANALYSIS - AUTO/HLPCI + INTOUT=YES      SUBCASE 1
LOAD STEP = 1.00000E+00

      GRID POINT FORCE BALANCE

POINT-ID  ELEMENT-ID  SOURCE      T1      T2      T3      R1      R2      R3
123      22 QUAD4      1.005275E+00 -8.801861E+00 -3.010083E+00 -6.609823E-04  5.106755E-04  3.166376E-02
   123      *TOTALS*    1.005275E+00 -8.801861E+00 -3.010083E+00 -6.609823E-04  5.106755E-04  3.166376E-02

```

Listing 5-3.

Obviously, neither the residual nor the upstream superelement has the balanced grid point force. However, if both sides are added together, the balance will be reached.

- Computational time and memory requirement increase when grid point force balance or element strain energy output is requested. For a large nonlinear model, allocation of sufficient memory is important. Spill logic is built in the code, but it may increase the overall computational time.

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 Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing, and electronics design meet tomorrow. Our solutions help companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities, and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit [siemens.com/software](https://www.siemens.com/software) or follow us on [LinkedIn](#), [Twitter](#), [Facebook](#), and [Instagram](#).

*Siemens Digital Industries Software —
Where today meets tomorrow.*

© 2019 Siemens. A list of relevant trademarks can be found [here](#). All other trademarks belong to their respective owners.